SINUMERIK 840D/810D/FM-NC

Short Guide

Operation 12.98 Edition

User Documentation

Overview of SINUMERIK 840D/810D/FM-NC Documentation (12.98)

General Documentation

SINUMERIK 840D/810D/ FM-NC
Brochure

SINUMERIK 840D/810D/ FM-NC/611
Catalog Ordering Info NC 60.1 ***)**
Technical Info. NC 60.2

SINUMERIK SIROTEC SIMODRIVE Accessories
Catalog Accessories NC-Z

User Documentation

SINUMERIK 840D/810D/ FM-NC
AutoTurn
– Short Guide
– Programming (1)
– Setup (2)

SINUMERIK 840D/810D
Operator's Guide
– **Unit Operator Panel**
– **HPU**

SINUMERIK 840D/810D/ FM-NC
Diagnostics Guide *)

SINUMERIK 840D/810D/ FM-NC
Operator's Guide
– Short Guide
– Operator's Guide ***)**

User Documentation

SINUMERIK 840D/810D/ FM-NC
Program. Guide
– Short Guide
– Fundamentals ***)**
– Advanced ***)**
– Cycles
– Measuring Cycles

SINUMERIK 840D/810D
Operator's Guide
– **ManualTurn**
– Short Guide ManualTurn
– **ShopMill**
– Short Guide ShopMill

Manufacturer / Service Documentation

SINUMERIK 840D/810D
Description of Functions
– **ManualTurn**
– **ShopMill**

SINUMERIK 840D/810D
Description of Functions **Synchronized Actions** Wood, Glass, Ceramics

SINUMERIK 840D/810D
Descr. of Functions
– **Computer Link**
– **Tool Data Information System**

SINUMERIK
Configuring (HW) *)
– FM-NC
– 810D
– 840D

Manufacturer / Service Documentation

SINUMERIK 840D/810D/ FM-NC
Operator Components (HW) ***)**

SINUMERIK SIMODRIVE 611D 840D/810D
Description of Functions **Drive Functions *)**

SINUMERIK 840D/810D/ FM-NC
Description of Functions
– Basic Machine ***)**
– Extended Functions
– Special Functions

SINUMERIK 840D/810D/ FM-NC
Configuring Kit MMC100/101
– Configuring Syntax
– Development Kit

SINUMERIK 840D/810D/ FM-NC
Screen Kit MMC100/101 SW Update and Configuration

SINUMERIK 840D/810D
Description of Functions **Tool Management**

SINUMERIK 840D/810D/ FM-NC
Description of Functions Operator Interface **OP 030**

Manufacturer / Service Documentation

SINUMERIK SIMODRIVE
Description of Functions SINUMERIK **Safety Integrated**

SINUMERIK 840D
Description of Functions **Digitizing**

SINUMERIK SIMODRIVE
Installation & Start-up Guide *)
– FM-NC
– 810D
– 840D/611D
– MMC

SINUMERIK SIMODRIVE 840D/810D FM-NC 611D
Lists *)

SINUMERIK SIMODRIVE 840D 611D
Description of Functions **Linear Motor**

SINUMERIK SIMODRIVE 840D 611D
Description of Functions **Hydraulics Module**

SINUMERIK SIMODRIVE SIROTEC
EMC Guidelines

Electronic Documentation

SINUMERIK SIMODRIVE 840D/810D/ FM-NC 611, Motors
DOC ON CD *)
The SINUMERIK System

Manufacturer / Service Documentation

SINUMERIK 840D/810D
Description of Functions **Fanuc Programs** for SINUMERIK

***)** These documents are a minimum requirement for the control

SINUMERIK 840D/810D/FM-NC

Short Guide
Operation

Valid for

Control	*Software version*
SINUMERIK 840D	5
SINUMERIK 840DE (Export version)	5
SINUMERIK 810D	3
SINUMERIK 810DE (Export version)	3
SINUMERIK FM-NC	3

12.98 Edition

SINUMERIK® documentation

Printing history

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

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

Status code in the "*Remarks*" column:

A .... New documentation
B .... Unrevised reprint with new Order No.
C .... Revised edition with new status.
If factual changes have been made on the page since the last edition, this is indicated by a new edition coding in the header on that page.

Edition	**Order No.**	**Remarks**
11.94	6FC5298-0AA10-0BP0	A
04.95	6FC5298-2AA10-0BP0	C
03.96	6FC5298-3AA10-0BP0	C
08.97	6FC5298-4AA10-0BP0	C
12.98	6FC5298-5AA10-0BP0	C

This manual is included in the documentation on CD-ROM (**DOCONCD**)

Edition	**Order No.**	**Remarks**
02.99	6FC5298-5CA00-0BG0	C

Further information is available on the Internet under:
http://www.ad.siemens.de/sinumerik

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

The reproduction, transmission or use of this document or its contents is not permitted without express written authority. Offenders will be liable for damages. All rights, including rights created by patent grant or registration of a utility model or design, are reserved.

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

Subject to change without prior notice.

© Siemens AG 1994 - 1998 All Rights Reserved.

Introduction

MMC 100.2 MMC 103

Validity

Unless stated otherwise, this operator's guide applies to the SINUMERIK 840D/810D and SINUMERIK FM-NC controls

How to use this booklet

This booklet is an operator's guide describing all the main programming steps.

The aim is to provide the operator with some quick help and a memory aid especially for commands that are used infrequently or to offer a quick reference on various parameters.

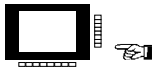
This guide therefore has little text!

First familiarize yourself with the symbols below so that you understand them better whenever they occur on the following pages.

The symbols

Gives you a tip or background information

Screen displays



Operation via softkeys

Input through the operator keyboard

Input through the machine control panel

© Siemens AG 1998. All rights reserved.

© Siemens AG 1998 All rights reserved

List of Sections

Operator Control 1

Set-up 2

Manually Controlled Operation 3

Part Programming 4

Program Management 5

Automatic Mode 6

Alarms and Messages 7

© Siemens AG 1998. All rights reserved.

© Siemens AG 1998 All rights reserved

1. Operator Control

1.1 Key assignments slimline operator panel OP 031 1-10

1.2 Key assignments slimline operator panel OP 032 1-11

1.3 Key assignments operator panel 1-12

1.4 Key assignments external machine control panel 1-13

1.5 Graphical user interface 1-16

1.6 Operating areas 1-18

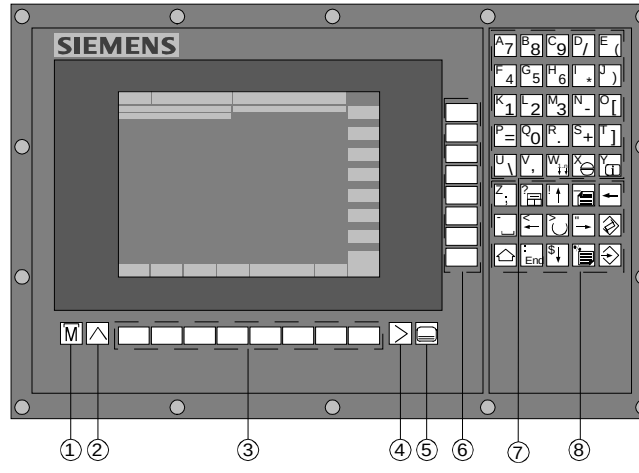
1.7 Operating principle 1-19

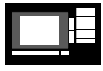
© Siemens AG 1998 All rights reserved.

1.1 Key assignments slimline operator panel OP 031

MMC 100.2

MMC 103



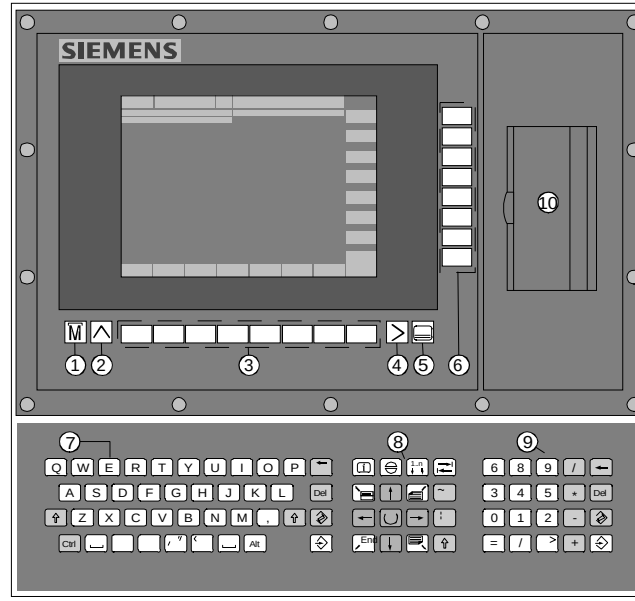


1	Machine area key
2	Recall
3	Softkey bar (horizontal)
4	ETC key (menu extension)
5	Area changeover key
6	Softkey bar (vertical)
7	Alphanumeric key
8	Correction/cursor block with control keypad and Input key

© Siemens AG 1998 All rights reserved.

1.2 Key assignments slimline operator panel OP 032

MMC 100.2 MMC 103



1	Machine area key
2	Recall
3	Softkey bar (horizontal)
4	ETC key (menu extension)
5	Area changeover key
6	Softkey bar (vertical)
7	Alphablock
8	Correction/cursor block with control keypad and Input key
9	Numeric block
10	Slot for PCMCIA adapter

© Siemens AG 1998 All rights reserved.

1.3 Key assignments operator panel

MMC 100.2 MMC 103

Meaning of the key symbols:
The keys marked with * correspond to the US layout.

Switch over channel

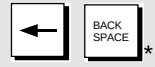
Acknowledge alarm

Information

Select window

Move cursor

Page up



Delete character

Blank

Selection key/toggle key

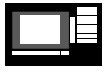
Edit/Undo

Shift

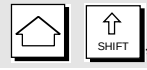
End of line

Page down

Input (accept)



For keys with double assignments:

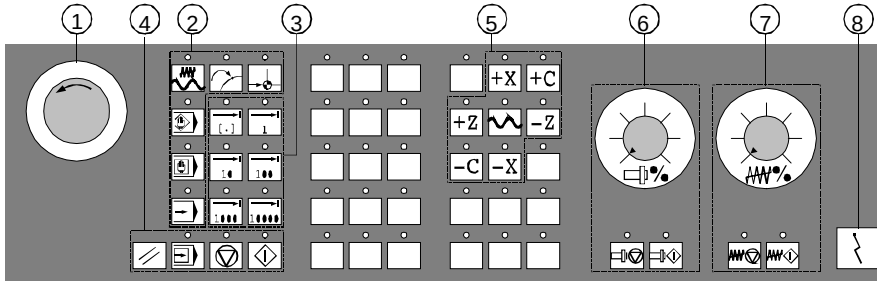


Shift with key "Shift"

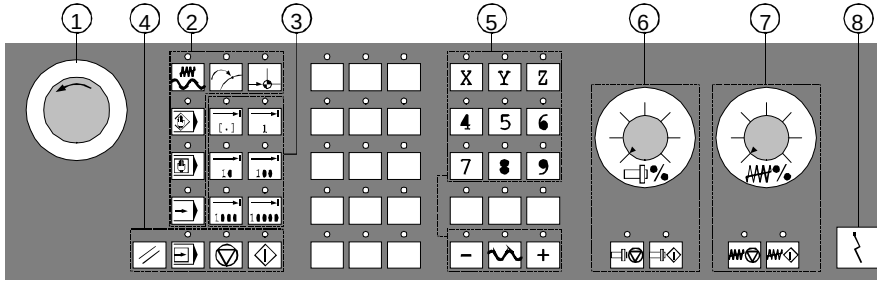
© Siemens AG 1998 All rights reserved.

1.4 Key assignments external machine control panel

MMC 100.2 MMC 103



Machine control panel for turning machines

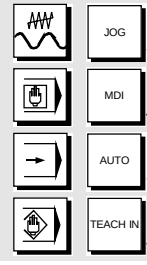


Machine control panel for milling machines

The keys marked with * correspond to the US layout.

1 Emergency stop pushbutton

2 Mode and machine functions



Jog

MDA

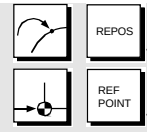
Automatic

Teach In

© Siemens AG 1998 All rights reserved.

1.4 Key assignments external machine control panel

MMC 100.2 MMC 103

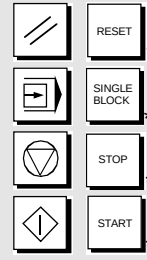


Repos

Ref

3 Incremental travel

4 Program control



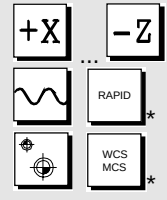
Reset

Single block

NC Stop

NC Start

5 Axis keys for turning machines

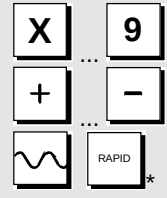


Axis keys (with direction)

Rapid traverse override

MCS/WCS

Axis keys for milling machines



Axis keys

Direction keys

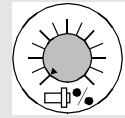
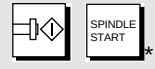
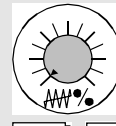
Rapid traverse override

MCS/WCS

© Siemens AG 1998 All rights reserved.

1.4 Key assignments external machine control panel

MMC 100.2 MMC 103

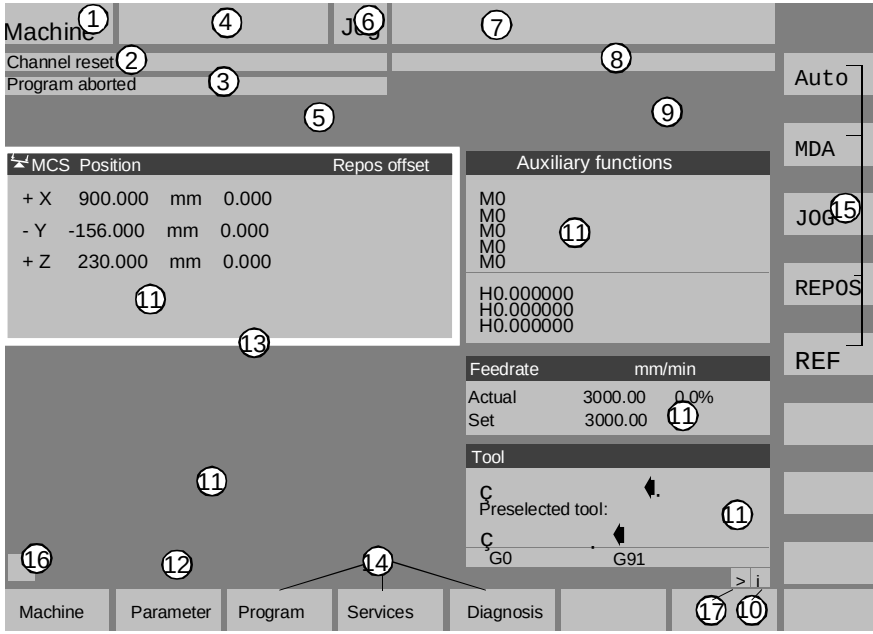
6	Spindle control	
		Spindle override
		Spindle stop
		Spindle start
7	Feed control	
		Feed/rapid traverse override
		Feed stop
		Feed start
8	Keyswitch	

© Siemens AG 1998 All rights reserved.

1.5 Graphical user interface

MMC 100.2

MMC 103



1	Operating area
2	Channel status
3	Program status
4	Channel and mode group
5	Alarm and message line

© Siemens AG 1998 All rights reserved.

1.5 Graphical user interface

MMC 100.2 MMC 103

6	Mode
7	Program name
8	Channel operational messages
9	Channel status display
10	Information relating to the menu bar
11	Work windows and NC displays
12	Dialog line with notes for operator
13	Focus
14	Horizontal softkey bar
15	Vertical softkey bar
16	Recall
17	etc.

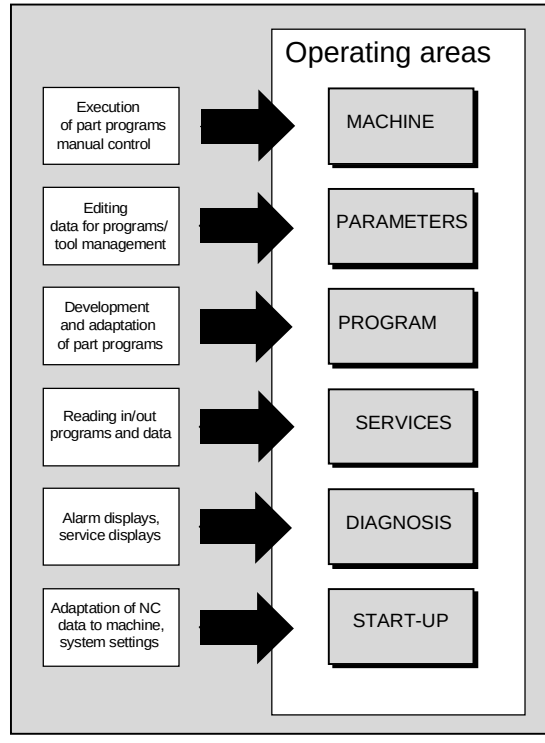
© Siemens AG 1998 All rights reserved.

1.6 Operating areas

MMC 100.2

MMC 103

The standard functions in the control are organized into the following operating areas:



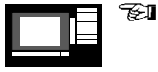
When you press the "Area changeover" key, the operating areas in the main menu appear in the horizontal softkey bar. You can use this key to switch from any menu to the main menu.

© Siemens AG 1998 All rights reserved.

1.7 Operating principle

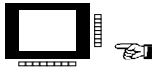
MMC 100.2 MMC 103

The MMC 100.2 and MMC 103 provide a range of key and softkey assignments which have the same function in all operating areas and menus.



Area changeover:

Switch back from any menu to the main menu of your control.



Softkeys:

Horizontal softkeys:
These keys subdivide each operating area into further menu levels. Each horizontal menu item has a vertical menu bar/softkey assignment.

Vertical softkeys:
Assigned with functions for the currently selected horizontal softkeys.

© Siemens AG 1998 All rights reserved.

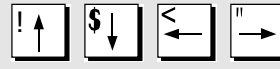
1.7 Operating principle

MMC 100.2 MMC 103

Navigation in the menu window:

Change menu windows:
Change the focus to the selected menu window.

Scroll in the menu window:
Scroll one screen page down or up.



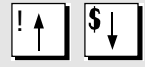
Position cursor in the menu window:
Position the cursor at the desired point in the menu window.

© Siemens AG 1998 All rights reserved.

1.7 Operating principle

MMC 100.2 MMC 103

Navigation in the directory tree:



Select directory/file:
Position the cursor on the desired directory/file.

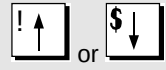
Open/close directory:
Open or close the selected directory.

Open file:
Open the desired file, e.g. if you want to edit the file in the ASCII editor (in this case the editor is opened automatically).

Select file (MMC103):
Select the desired file.

Select several files (MMC103):
Hold down the "Shift" and "Cursor Down" keys.

Select start of block.



Neighboring files are selected when you hold down the "Cursor Up" or "Cursor Down" keys.

Deselect selected files.

Cancel all selections.

© Siemens AG 1998 All rights reserved.

1.7 Operating principle

MMC 100.2 MMC 103

Edit inputs/values:

If you want 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 fields** (radio buttons or check boxes): Activate or deactivate the option field.

2. **Input fields:**
Switch to input mode.

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

You automatically switch to input mode if you first position the cursor on the input field.

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

© Siemens AG 1998 All rights reserved.

1.7 Operating principle

MMC 100.2

MMC 103

You can use the "Toggle" key to select one of several default values.

3. **Selection list (MMC 103):** Display the preselection of possible values.

Open selection list

Position the cursor on the desired values.

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

Switch to the next value in the selection list without displaying the entire list.

© Siemens AG 1998 All rights reserved.

1.7 Operating principle

MMC 100.2

MMC 103

Confirm/cancel input:

OK	**Confirm input:** Save the inputs and exit the current menu (you automatically return to the calling menu).
Cancel	**Cancel input:** Reject the inputs and quit the current menu (you automatically return to the calling menu).
	Reject the inputs and quit the current menu (you automatically move up one level).
	Clear the current input but remain in the current menu.

© Siemens AG 1998 All rights reserved.

1.7 Operating principle

MMC 100.2 MMC 103

Operation with the mouse:

If you have installed a mouse, operation is facilitated as follows:

1 "click"

1 click means:

- Activate the menu window.
- Position the cursor on the desired input field.
- Select directory.
- Activate softkey.
- Activate/deactivate radio button/check box.
- Activate input field.
- Open selection list.

2 "clicks"

2 clicks mean:

- Accept value/input.
- Open directory.

© Siemens AG 1998 All rights reserved.

© Siemens AG 1998 All rights reserved.

2. Set-up

2.1 Approach reference point 2-28

2.2 Enter tool offsets 2-29

2.3 Calculate tool offsets 2-30

2.4 Enter/alter zero offset (MMC100.2) 2-31

2.5 Enter/alter zero offset (MMC103) 2-32

2.6 Scratching method/determine zero offset (MMC100.2) 2-34

2.7 Scratching method/determine zero offset (MMC103) 2-35

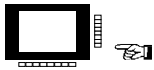
2.8 Program the setting data 2-36

© Siemens AG 1998 All rights reserved.

2.1 Approach reference point

MMC 100.2 MMC 103

Caution:
Position the axes if they are not already located at a safe position.
Always follow the axis movements directly on the machine.
Ignore the actual value display as long as the axes are not referenced (the software limit switches are not effective).



Machine — Select "Machine" operating area.

Jog or MDA — Choose "Jog" or "MDA".

Select "Ref" machine function.

Turning machine:
Traverse axes.

Milling machine:
Select axes and

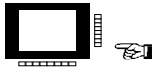
traverse axes.

Stop axis before reference point is reached.

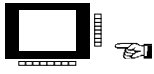
© Siemens AG 1998 All rights reserved.

2.2 Enter tool offsets

MMC 100.2 MMC 103



Parameter	Select "Parameter" operating area.
Tool offset	Select "Tool offset" menu.



Choose the desired function:

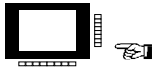
T no. +	Select tool (T no.).
T no. -	
D no. +	Select tool edge (D no.).
D no. -	
Delete	– Delete tool/tool edge
Go to	– Find tool
Overview...	List existing tools.
New...	Create new tool/tool edge.

Enter the new values.

© Siemens AG 1998 All rights reserved.

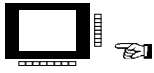
2.3 Calculate tool offsets

MMC 100.2 MMC 103



Parameter	Select "Parameter" operating area.
Tool offset	Select "Tool offset" menu.

Position cursor on desired tool parameter.

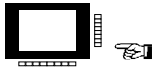


Determine compensa.	Select axis and enter reference value.
Calculate	The current position and the reference value of the selected tool parameter are calculated automatically.

© Siemens AG 1998 All rights reserved.

2.4 Enter/alter zero offset (MMC100.2)

MMC
100.2



Parameter

Select "Parameter" operating area.

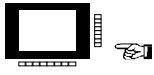
Zero offsets

Select "Zero offsets" menu.

Overview

Select zero offsets:
- settable
- active settable
- active programmable
- sum of active ones
- external ones
- basic offsets

Position the cursor on the desired field and enter/alter the value, or
use the "Toggle" key to choose a new value.



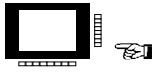
Save

Save the values.

© Siemens AG 1998 All rights reserved.

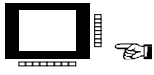
2.5 Enter/alter zero offset (MMC103)

MMC
103



Parameter	Select "Parameter" operating area.
Zero offset	Select "Overview of settable zero offsets" menu.

		Select zero offset:
Axes +		Changes to the defined zero offsets of the next or previous axis.
Axes -		
Offsets	Rotation, scal., mirr.	Change display mode of the currently displayed zero offset. The display shows either the absolute offsets or the proportional values of rotation, scaling or mirroring.



	Enter/change zero offset:
Basic ZOs	Display all the defined basic zero offsets.
Sett. ZOs	Display all the defined settable zero offsets.

© Siemens AG 1998 All rights reserved.

2.5 Enter/alter zero offset (MMC103)

MMC
103

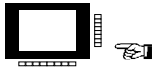
Position the cursor on the desired field and enter/alter the value.

Please note:
Changes in the zero offset are updated at entry.

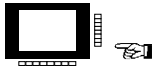
© Siemens AG 1998 All rights reserved.

2.6 Scratching method/determine zero offset (MMC100.2)

MMC
100.2



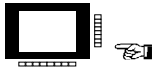
Parameter	Select "Parameter" operating area.
	Select "Jog" mode.
Zero offsets	Select "Zero offsets" menu.
Determine ZO	"Determine ZO (scratching)" window, enter • the tool (T no.), • the tool edge (D no.), • the corresponding length parameters (1, 2, 3) and the direction (+, –, none), • the allowance and direction of the radius (+, –, none), • the allowance and direction of a user-definable offset (+, –, none).
	You can select values using the "Toggle" key.
+ +X	Traverse axes (scratching).
OK	With "OK" all the values are entered into the selected zero offset.



© Siemens AG 1998 All rights reserved.

2.7 Scratching method/determine zero offset (MMC103)

MMC
103



Machine	Select "Machine" operating area.
Scratching	Active level, active zero offset and active tool are selected.
	Select axis which shall be traversed, with the cursor.
or	Traverse axis to the workpiece, enter desired set position and confirm with "Input". The offset is calculated.
OK	With "OK" all the values are entered into the selected zero offset.

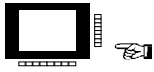
Please note:
The calculation of the offset always refers to the current workpiece coordinate system (WCS).

	In order to take account of the tool geometry, position the cursor in the column "Geometry + wear" on the axis which shall be traversed and select with "Toggle" key, how the tool offset shall be calculated.

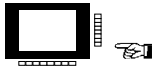
© Siemens AG 1998 All rights reserved.

2.8 Program the setting data

MMC 100.2 MMC 103



Parameter	Select "Parameter" operating area.
Setting data	Select the "Setting data" menu.



Define operating states using setting data:

Work area limit.	• Change working area limitation.
Jog data	• Change jog data.
Spindle data	• Change spindle data.
Feedrate DRY	• Change dry run feedrate for DRY test mode.
Starting angle	• Change starting angle for thread cutting.
Misc.	• Display miscellaneous setting data.
Protection zones	• Choose level for protection zone.

© Siemens AG 1998 All rights reserved.

2.8 Program the setting data

MMC 100.2

MMC 103

Position the cursor on the desired field and change the value, or

select a new value using the "Toggle" key.

© Siemens AG 1998 All rights reserved.

© Siemens AG 1998 All rights reserved.

3. Manually Controlled Operation

3.1 Jog/Jog-Inc mode 3-40

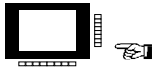
3.2 MDA mode 3-42

3.3 Teach In mode 3-43

© Siemens AG 1998 All rights reserved.

3.1 Jog/Jog-Inc mode

MMC 100.2 MMC 103



Machine

Select "Machine" operating area.

Jog

Select "Jog" mode.

"Reference point approach" is deactivated.

+X ... -Z

Turning machine:
Traverse axes.

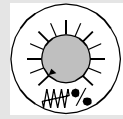
X ... 9

Milling machine:
Select axis and

\+ −

traverse.

The traversing velocity is stored in the "setting data".



If necessary, use the override to set the velocity.

If necessary, move axes with rapid traverse.

© Siemens AG 1998 All rights reserved.

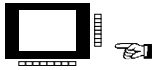
3.1 Jog/Jog-Inc mode

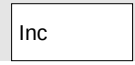
MMC 100.2

MMC 103

Select/enter increment value for traversing position ("Inc"):

Traverse axes in defined increments, or





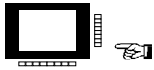
enter the desired increment.

Traverse axis with the entered increment.

© Siemens AG 1998 All rights reserved.

3.2 MDA mode

MMC 100.2 MMC 103



Machine

Select "Machine" operating area.

MDA

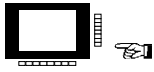
Select "MDA" mode.

In "Teach In" mode, it is only possible to insert blocks at points in the program which have not yet been executed.

Enter NC block (one or several consecutive program blocks).

Confirm input.

Execute NC block.



MDA-Prog. sichern

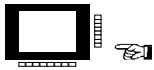
Save program in MDA buffer.

If no program name is entered, the program is automatically stored in the MDA buffer with the name OSTORE.MPF (MMC100.2) or MDAX.MPF (MMC103).

© Siemens AG 1998 All rights reserved.

3.3 Teach In mode

MMC 100.2 MMC 103



Machine — Select "Machine" operating area.

MDA — Select "MDA" mode.

Select "Teach In" submode.

In "Teach In" mode, it is only possible to insert blocks at points in the program which have not yet been executed.

1. Manual positioning:

Turning machine:
Traverse axes.

Milling machine:
Select axis and

traverse.

The axis name and the traversed path are displayed in the "Teach In program" window.

© Siemens AG 1998 All rights reserved.

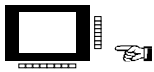
3.3 Teach In mode

MMC 100.2

MMC 103

2. Manual input of coordinates:

- Enter coordinates of traversing positions and
- enter additional functions (preparatory conditions, miscellaneous functions, etc.) in the "MDA program" window.



MMC100.2

Delete block — Delete the current block.

Insert block — Insert an empty line before the current block.

Save block — Save new axis positions and additional functions.

Executed the entered program blocks and display them in the current block window.

MMC103

Enter and delete blocks on keyboard.
Positions are automatically saved at entry.

© Siemens AG 1998 All rights reserved.

4. Part Programming

4.1 Select/enable workpiece/part program 4-46

4.2 Open/edit part program 4-47

4.3 Create new workpiece/part program 4-49

4.4 Execute workpiece/part program 4-50

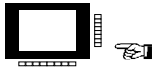
4.5 Copy/paste, rename, delete, enable 4-51

4.6 Load/unload program (MMC 103) 4-53

© Siemens AG 1998 All rights reserved.

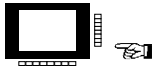
4.1 Select/enable workpiece/part program

MMC 100.2 MMC 103



Program

Select "Program" operating area.



Select workpiece/part program:

Select the level:

Work-pieces
- Workpieces

Part programs
- Part programs

Sub-routines
- Subroutines

Standard cycles
- Standard cycles (MMC100.2)

User cycles
- User cycles

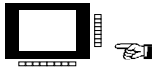
Clip-board
- Clipboard

Position the cursor on the desired file in the directory tree.

© Siemens AG 1998 All rights reserved.

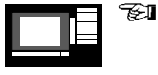
4.2 Open/edit part program

MMC 100.2 MMC 103



Program

Select "Program" operating area.

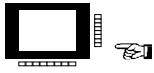


Open part program:

Position the cursor on the desired file in the directory tree.

Open program in ASCII editor:

Press the "Input" key.



Activate interactive programming (MMC103):

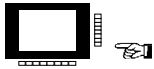
Interactive progr.

Press softkey "Interactive progr.".

© Siemens AG 1998 All rights reserved.

4.2 Open/edit part program

MMC 100.2 MMC 103



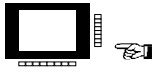
Edit part program:

Overwrite	Toggle between insert and overwrite mode.
Mark block	Mark the beginning of block.
Copy block	Save the block in the Clipboard.
Delete block	Delete block.
Insert block	Paste the block from the Clipboard.
Search/ go to ...	Position on block no., beginning of file, end of file or search string.
Search	Enter search string.
Substitute	Enter substitute.
Continue search	Next occurrence of search string.
Go to ...	Enter block no.
Save file	Save part program.
Close editor	Close text editor.

© Siemens AG 1998 All rights reserved.

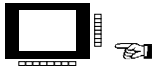
4.3 Create new workpiece/part program

MMC 100.2 MMC 103



Program

Select "Program" operating area.



Work-pieces

Part programs

Sub-routines

Select "Workpieces", "Part programs" or "Subroutines".

Open directory.

New ...

Enter name of workpiece, main program or subroutine and

select "File type".

MMC103:

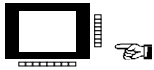
Select key

- text editor
- activate interactive programming.

© Siemens AG 1998 All rights reserved.

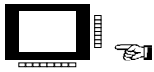
4.4 Execute workpiece/part program

MMC 100.2 MMC 103



Program

Select "Program" operating area.



Work-pieces

Select workpiece:

Mark the desired workpiece with the cursor in the workpiece overview.

Selection

Select the workpiece.

Part programs

Select part program:

Mark the desired workpiece with the cursor in the part program overview.

Selection

Select the part program.

Start the workpiece/part program.

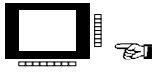
Clip-board

You can also run part programs directly from the Clipboard.

© Siemens AG 1998 All rights reserved.

4.5 Copy/paste, rename, delete, enable

MMC 100.2 MMC 103



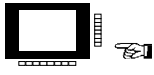
Program

Select "Program" operating area.

Select the level and position the cursor on the desired file in the directory tree.

Manage programs

MMC103:
Select "Manage programs".



Copy/paste program:

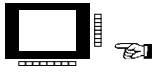
Copy

Select the source file.

Paste

Save the source file in the target directory.

If necessary, use the "Toggle" key to select the file type.



Rename the program:

Rename

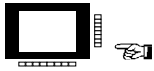
Press the "Rename" softkey and enter new name.

Use the "Toggle" key to select the file type.

© Siemens AG 1998 All rights reserved.

4.5 Copy/paste, rename, delete, enable

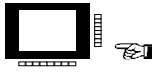
MMC 100.2 MMC 103



Delete

Delete program:

Delete the program from the directory.



Change enable

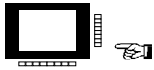
Change enable:

Set (x) or cancel ( ) enable.

© Siemens AG 1998 All rights reserved.

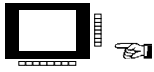
4.6 Load/unload program (MMC 103)

MMC
103



Program	Select "Program" operating area.

Position the cursor on the desired file in the directory tree.



Load	Load the program into the CNC memory.
Unload	Save the program to hard disk.

Please note:
When you activate "Load", the program is automatically deleted from the hard disk. When you activate "Unload", the program is automatically cleared from the CNC memory.

© Siemens AG 1998 All rights reserved.

© Siemens AG 1998 All rights reserved.

5. Manage data

5.1 Transfer data (MMC 100.2) 5-56

5.2 Transfer data (MMC 103) 5-59

5.3 Create new file/directory (MMC 103) 5-61

5.4 Load/unload program (MMC 103) 5-62

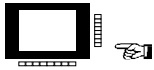
5.5 Copy/paste, delete (MMC 103) 5-63

5.6 Directory/file/archive properties (MMC 103) 5-64

© Siemens AG 1998 All rights reserved.

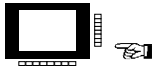
5.1 Transfer data (MMC 100.2)

MMC
100.2



Services

Select "Services" operating area.



Read out data:

Data out

Select "Data out" menu.

Select interface:

V.24-user

- V.24 user

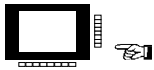
V.24-printer

- V.24 printer

V.24-PG/PC

- V.24-PG/PC

Position the cursor on the desired file in the directory tree.



Start

Start "Data out".

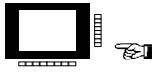
Stop

Interrupt "Data out".

© Siemens AG 1998 All rights reserved.

5.1 Transfer data (MMC 100.2)

MMC
100.2



Read in data:

Data in

Select " Data in" menu.

Select interface:

V.24
user

- V.24 user

V.24-
PG/PC

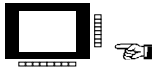
- V.24-PG/PC

Position the cursor on the desired file in the directory tree.

© Siemens AG 1998 All rights reserved.

5.1 Transfer data (MMC 100.2)

MMC
100.2

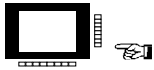


Start	Organize data according to the existing path:
☒	Path/workpiece from archive file
	Store all files in the directory, irrespective of archived path:
☐	Path/workpiece from archive file
	Store all files in the Clipboard, irrespective of archived path:
☒	Read into the Clipboard
OK	Start "Data in".
Stop	Interrupt "Data in".

© Siemens AG 1998 All rights reserved.

5.2 Transfer data (MMC 103)

MMC
103



Services

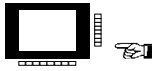
Select "Services" operating area.

Data out

Read out data:

Select "Programs/data" menu.

Select data in the directory tree.



Select target area:

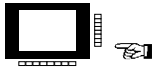
V.24
- V.24 interface

PG
- Programming device

Diskette...
- Diskette drive

Archive...
- "Archive" on hard disk

NC Card
- NC Card



Start transfer:

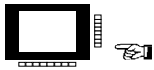
Start

Initiate data transfer on disk/archive (softkey label changes to "Stop").

© Siemens AG 1998 All rights reserved.

5.2 Transfer data (MMC 103)

MMC
103

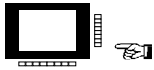


Data in

Read in data:

Select "Programs/data" menu.

Select data in the directory tree.



Select source area:

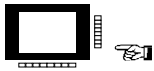
V.24
- V.24 interface

PG
- Programming device

Diskette...
- Diskette drive

Archive...
- "Archive" on hard disk

NC Card
- NC Card



Start

Start transfer:

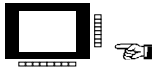
Initiate data transfer on disk/archive (softkey label changes to "Stop").

The data transfer is initiated automatically. You can interrupt data transfer at any time by pressing the softkey again.

© Siemens AG 1998 All rights reserved.

5.3 Create new file/directory (MMC 103)

MMC
103

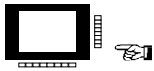


Services

Select "Services" operating area.

Manage data

Select "Manage data" menu.



New...

Enter program name and select type of program.

By selecting the "End" softkey, you can switch between the areas "Name" and "Type of file".

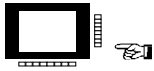
OK

New file/directory will be created.

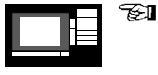
© Siemens AG 1998 All rights reserved.

5.4 Load/unload program (MMC 103)

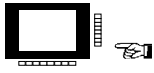
MMC
103



Services	Select "Services" operating area.
Manage data	Select "Manage data" menu.



Position the cursor on the desired file in the directory tree.

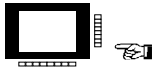


Load	Load the file from the hard disk into the CNC memory.
Unload	Unload the file from the CNC memory to the hard disk.

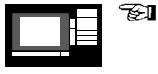
© Siemens AG 1998 All rights reserved.

5.5 Copy/paste, delete (MMC 103)

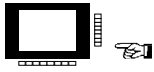
MMC 103



Services	Select "Services" operating area.
Manage data	Select "Manage data" menu.

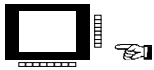


Position the cursor on the source file in the directory tree.



Copy/paste program:

Copy	Select target directory.
Programs data	Close target window.
Paste	Copy source file to target directory.
Clipboard	Copy to or from Clipboard.
Diskette	Copy to or from diskette.



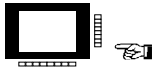
Delete program:

Delete	Delete selected file.

© Siemens AG 1998 All rights reserved.

5.6 Directory/file/archive properties (MMC 103)

MMC
103

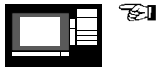


Services

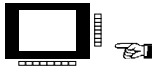
Select "Services" operating area.

Manage data

Select "Manage data" menu.



Position the cursor on the desired file in the directory tree.



Properties

Select "Properties" menu.

- Rename file
- Convert file type
- Change access rights for file/directory
- Display contents of all files

The access level can only be changed with the appropriate access rights.

© Siemens AG 1998 All rights reserved.

6. Automatic Mode

6.1 Select workpiece/program 6-66

6.2 Start/stop/cancel program 6-67

6.3 Repos – Reposition after interruption 6-68

6.4 Overstore 6-70

6.5 Display program level 6-71

6.6 Program correction 6-72

6.7 Execution from V.24 (MMC 100.2) 6-73

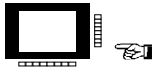
6.8 Load/unload program from hard disk (MMC 103) 6-74

6.9 Execution from hard disk (MMC 103) 6-75

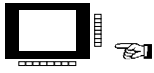
© Siemens AG 1998 All rights reserved.

6.1 Select workpiece/program

MMC 100.2 MMC 103



Machine	Select "Machine" operating area.
AUTO	Select "Automatic" mode.



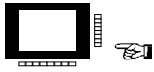
Program overview	Select "Program overview" menu.
Work-pieces	Select desired workpiece/part program with the cursor in the workpiece/part program overview.
Part programs	
Selection	Select workpiece/part program.

Change enable	Set (x) or cancel ( ) enable.

© Siemens AG 1998 All rights reserved.

6.2 Start/stop/cancel program

MMC 100.2 MMC 103



Machine

Select "Machine" operating area.

AUTO

Select "Automatic" mode.

Please check that:
No alarms are active.
The program is selected.
The feed enable is active.
The spindle enable is active.

Start part program.

Stop part program.

Cancel part program.

A part program which has been interrupted by "NC Stop" can be resumed with "NC Start".

A part program which has been interrupted by "Reset" runs from the start when you press "NC Start".

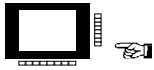
© Siemens AG 1998 All rights reserved.

6.3 Repos – Reposition after interruption

MMC 100.2

MMC 103

After a program interrupt ("NC Stop") you can move the tool away from the contour in the manual mode. The control saves the coordinates of the interruption point. The traversed path differences of the axes are displayed.



Machine	Select "Machine" operating area.
Jog	Select "Jog" mode.

	Reposition after program interruption.
	Turning machine:
+X ... –Z	Traverse axes up to the point of interruption.
	Milling machine:
X ... 9	Select axis and
+ ... –	traverse axes up to the interruption point.

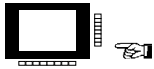
Caution:
Travel beyond the interruption point is inhibited. The feed override switches are active.

© Siemens AG 1998 All rights reserved.

6.3 Repos – Reposition after interruption

MMC 100.2

MMC 103



Machine

Select "Machine" operating area.

AUTO

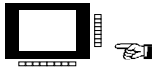
Select "Automatic" mode.

Continue machining.

© Siemens AG 1998 All rights reserved.

6.4 Overstore

MMC 100.2 MMC 103



Machine

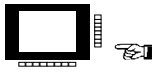
Select "Machine" operating area.

AUTO

Select "Automatic" mode.

Overstore with single block:
The program automatically stops at the next block boundary.

Overstore without single block:
Stop part program.



Overstore

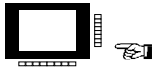
Enter the values and functions to be executed.

Run the entered block.

© Siemens AG 1998 All rights reserved.

6.5 Display program level

MMC 100.2 MMC 103

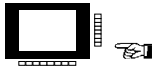


Machine

Select "Machine" operating area.

AUTO

Select "Automatic" mode.



Program level

Display block numbers for main programs and subroutines with the respective pass counts (P).

Current block

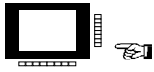
Display the blocks of the current program.

Pressing the "Program level" softkey automatically changes the level to and from "current block".

© Siemens AG 1998 All rights reserved.

6.6 Program correction

MMC 100.2 MMC 103



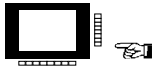
Machine

Select "Machine" operating area.

AUTO

Select "Automatic" mode.

The control has detected a system error in the part program. Execution of the part program is automatically interrupted.



Correct program

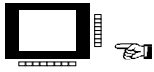
Correct the program block with the error.

Continue machining.

© Siemens AG 1998 All rights reserved.

6.7 Execution from V.24 (MMC 100.2)

MMC
100.2



Machine	Select "Machine" operating area.
Program overview	Select "Program overview" menu.
Execution from extern.	Select "Execution from external source" menu.

Position the cursor on the desired file in the directory tree.

Start execution from V.24.

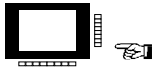
Executed blocks appear in the current block window.

Alternatively, it is possible to start execution in the "Services" operating area.
You can change the transfer parameters in the "Services" operating area. The transfer is recorded in the error log.

© Siemens AG 1998 All rights reserved.

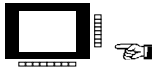
6.8 Load/unload program from hard disk (MMC 103)

MMC
103



Machine	Select "Machine" operating area.
Program overview	Select "Program overview " menu.

Position the cursor on the desired file in the directory tree.



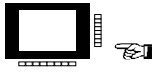
Load	Load the program into the CNC memory.
Unload	Save the program on the hard disk.

Please note:
When you activate "Load", the program is automatically deleted from the hard disk. When you activate "Unload", the program is automatically cleared from the CNC memory.

© Siemens AG 1998 All rights reserved.

6.9 Execution from hard disk (MMC 103)

MMC
103



Machine

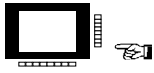
Select "Machine" operating area.

Program overview

Select "Program overview" menu.

Position the cursor on the desired file in the directory tree.

Press the "NC Start" key.



Execution fr. hard disk

Start/cancel read-in and execution.

The program remains stored with "Execution from hard disk".

© Siemens AG 1998 All rights reserved.

© Siemens AG 1998 All rights reserved.

7. Alarms and Messages

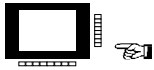
7.1 Alarms/messages 7-78

© Siemens AG 1998 All rights reserved.

7.1 Alarms/messages

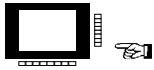
MMC 100.2

MMC 103



Diagnosis

Select "Diagnosis" operating area.



Alarms

Display alarms:

Display alarms with "alarm number", "date", "explanation" and "cancel criterion".

Delete alarm:

Switch the control off and on again.

Press "Reset".

Press "Acknowledge alarm".

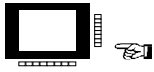
Alarm is canceled with "NC Start".

Alarm is canceled with "Recall".

© Siemens AG 1998 All rights reserved.

7.1 Alarms/messages

MMC 100.2 MMC 103

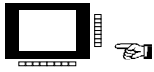


Messages

Display messages:

Display PLC error messages and PLC operational messages.

Caution:
PLC error messages must always be acknowledged.



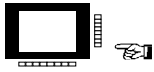
Alarm log

Display alarm log:

Display complete log of alarms and messages which have occurred.

Display new

Update alarm log.



Acknowledge MMC alarm

MMC 103:

Select and delete MMC alarm.

© Siemens AG 1998 All rights reserved.

© Siemens AG 1998 All rights reserved.

SIEMENS AG
A&D MC IS
P.O. Box 3180

D-91050 Erlangen
Federal Republic of Germany

(Fax ++49 9131/98-1145)

Suggestions

Corrections

for Publication/Manual

SINUMERIK 840D/810D/FM-NC

User Documentation

From

Name

Company/Department

Address

_____

_____

Telephone: /

_____

Telefax: /

Short Guide
Operation

Order No.: 6FC5298-5AA10-0BP0
Edition: 12.98

Should you come across any printing errors when reading this publication, please notify us on this sheet. Suggestions for improvement are also welcome.

Suggestions and/or corrections