SIEMENS

SINUMERIK 840D sl

CNC: ShopTurn

Commissioning Manual

Valid for

Control
SINUMERIK 840D sl/840DE sl

Software	*Version*
NCU system software for SINUMERIK 840D sl/840DE sl	1.5
with ShopTurn	7.5

01/2008 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.

Edition	Order No.	Remarks
03/2001	6FC5 297–6AD70–0BP0	**A**
01/2002	6FC5 297–6AD70–0BP1	**C**
03/2004	6FC5 297–6AD70–0BP2	**C**
08/2005	6FC5 397–5AP10–0BA0	**C**
11/2006	6FC5 397–5AP10–1BA0	**C**
01/2008	6FC5 397–5AP10–2BA0	**C**

Further information is available in the Internet under:
http://www.siemens.com/motioncontrol

This publication was produced with Interleaf V 7

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.

We have checked that the contents of this document correspond to the hardware and software described. Nevertheless, differences might exist and therefore we cannot guarantee that they are completely identical. The data in this document is regularly checked and the necessary corrections are included in subsequent editions. Suggestions for improvement are also welcome.

Subject to change without prior notice.

Order No. 6FC5 397–5AP10–2BA0
Printed in the Federal Republic of Germany

Siemens–Aktiengesellschaft

Preface

SINUMERIK documentation

The SINUMERIK documentation is organized in three parts:

- General documentation
- User documentation
- Manufacturer/Service documentation

A documentation overview with the available languages, updated monthly, can be found on the Internet under:
http://www.siemens.com/motioncontrol
Follow the menu items —> ”Support” —> ”Technical Documentation” —> ”Documentation Overview” .

The Internet edition of the DOConCD, the DOCon Web, can be found under:
http://www.automation.siemens.com/doconweb

You can find information about training courses on offer and about the FAQs (frequently asked questions) on the Internet under:
http://www.siemens.com/motioncontrol under ”Support”.

Target group

This documentation is intended for manufacturers of single–carriage turning machines using SINUMERIK 840D sl. It provides information needed for configuring and commissioning ShopTurn.

Standard scope

This document provides information about the control system design and the interfaces of the individual components. It also describes the commissioning procedure for ShopTurn with SINUMERIK 840D sl.

For detailed information about individual functions, function assignment and performance data of individual components, please refer to the appropriate document for the subject concerned (e.g. manuals, description of functions etc.).

User–oriented activities such as the creation of parts programs and control operating procedures are described in details in separate documents.

Further descriptions of tasks to be performed by the machine tool manufacturer are also available for the standard SINUMERIK 840D sl. We may refer to them in this documentation if appropriate.

This documentation only describes 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.

Further, for the sake of simplicity, this documentation does not contain all detailed information about all types of the product and cannot cover every conceivable case of installation, operation or maintenance.

Technical Support

If you have any technical questions, please contact our hotline:

Europe/Africa

Tel.: +49 (0) 180 5050–222
Fax: +49 (0) 180 5050–223
Internet: http://www.siemens.com/automation/support–request

America

Tel.: +1 (0) 423 262 2522
Fax: +1 (0) 423 262 2200
E–mail: mailto:techsupport.sea@siemens.com

Asia/Pacific region

Tel.: +86 (0) 1064 719 990
Fax: +86 (0) 1064 747 474
E–mail: mailto:techsupport.asia@siemens.com

Note

Country–specific telephone numbers for technical support are provided on the Internet at:
http://www.siemens.com/automation/service&support

Questions about this documentation

For questions on the documentation (suggestions, corrections), please send a fax or e–mail to the following address:

Fax: +49 (0) 9131– 98 63315
E–mail: mailto:docu.motioncontrol@siemens.com

You can find a fax template in the appendix to this document.

SINUMERIK Internet address

http://www.siemens.com/sinumerik

Search guides

The alarms of SINUMERIK 840D sl can be found in

References: /DAsl/ Diagnostics Manual

For further useful information on startup and troubleshooting, please refer to

References: /FB/, D1, ”Diagnostics Tools”

Notes

The following symbols with special significance are used in the documentation:

Note

This symbol always appears in the document where further information is provided.

Safety information

This manual contains information which you must observe in order to ensure your own personal safety, as well as to avoid material damage. Notes relating to your safety are highlighted in the manual by means of a warning triangle; no warning triangle appears in conjunction with notes relating to material damage. Depending on the hazard level, warnings are indicated in a descending order as follows:

Danger

indicates that death or serious injury **will** result if proper precautions are not taken.

Warning

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

Caution

with a safety alert symbol, indicates that minor personal injury **may** result if proper precautions are not taken.

Caution

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

Notice

indicates that an undesirable outcome or state **may** arise if the relevant notes are not observed.

Qualified personnel

The device/system may only be set up and used in conjunction with this documentation. Only **qualified personnel** should be allowed to commission and operate the device/system. 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.

Intended use

Please note the following:

Warning

This equipment is only allowed to be used for the applications described in the catalog and in the technical description, and only in conjunction with non–Siemens equipment and components recommended by Siemens. Correct, reliable operation of the product required proper transport, storage, positioning and assembly, as well as careful operation and maintenance.

Unit of measurement

In this manual, the units of the parameters are always indicated as metric values. The equivalent imperial units are shown in the table below.

Metric	Inch
mm	in
mm/tooth	in/tooth
mm/min	in/min
mm/rev	in/rev
m/min	ft/min

Contents

Space for your notes

Hardware

1

System structure

The hardware configuration for ShopTurn is as standard for SINUMERIK 840D sl.

References: /IDsl/, Commissioning CNC: NCK, PLC, drive, SINUMERIK 840D sl;
/GDsl/, Device Manual NCU, SINUMERIK 840D sl

Table 1-1 Basic components

Basic components	Order No.	Comment
TCU	6FC5312–0DA00–0AA0	
NCU 710.1	6FC5371–0AA10–0AA0	CNC: 3 MB; PLC: 128 KB
NCU 710.2	6FC5371–0AA10–0AA1	CNC: 3 MB; PLC: 512 KB
NCU 720.1	6FC5372–0AA00–0AA0	CNC: 3 MB; PLC: 128 KB
NCU 720.2	6FC5372–0AA00–0AA1	CNC: 3 MB; PLC: 512 KB
NCU 720.2PN	6FC5372–0AA01–0AA1	CNC: 3 MB; PLC: 512 KB
NCU 730.1	6FC5373–0AA00–0AA0	CNC: 3 MB; PLC: 128 KB
NCU 730.2	6FC5374–0AA01–0AA0	CNC: 3 MB; PLC: 128 KB
NCU 730.2 PN	6FC5373–0AA01–0AA1	CNC: 3 MB; PLC: 512 KB

Table 1-2 Operator components

Operator Components	Order No.	Comment
OP010 operator panel	6FC5203–0AF00–0AA0	
OP010C operator panel	6FC5203–0AF01–0AA0	
OP010S operator panel	6FC5203–0AF04–0AA0	
OP012 operator panel	6FC5203–0AF02–0AA0	
OP015 operator panel	6FC5203–0AF03–0AA0	
OP015A operator panel	6FC5203–0AF05–0AB0	
OP015AT operator panel	6FC5203–0AF05–1AB0	
TP015A operator panel	6FC5203–0AF08–0AB0	
TP015AT operator panel	6FC5203–0AF08–1AB0	
PROFIBUS DP direct key module	6FC5247–0AF11–0AA0	
Direct key module mounting kit	6FC5247–0AF30–0AA0	

Table 1-2 Operator components

PCU 50.3 – C	6FC5210–0DF31–2AA0	1.5 GHz, 512 MB, Windows XP
PCU 50.3 – P	6FC5210–0DF33–2AA0	2.0 GHz, 1024 MB, Windows XP

Table 1-2 Operator components

Operator Components	Order No.	Comment
MCP 310	6FC5203–0AF23–1AA0	
MCP 310 IE	6FC5303–0AF23–1AA0	
MCP 483	6FC5203–0AF22–1AA2	
MCP 483C IE	6FC5303–0AF22–0AA0	
MCP 483 IE	6FC5303–0AF22–1AA0	
CNC full keyboard KB 310C	6FC5203–0AF21–0AA0	
CNC Full Keyboard KB 483C	6FC5203–0AF20–0AA1	
PC keyboard, standard	6FC5203–0AC01–3AA0	

■

Supplementary Conditions

2

When using ShopTurn, please observe the following supplementary conditions:

- ShopTurn only runs in channel 1, mode group 1.
- ShopTurn has been designed for use with universal turning machines with four axes (X, Z, Y, and auxiliary axis for counterspindle) and 3 spindles (main, tool, and counterspindle).
- ShopTurn only runs with tool management.
- If you wish to use rotating tools, you need the software option ”cylinder surface transformation” (Tracyl) and ”end face machining” (Transmit). Order No. 6FC5 800–0AM27–0YB0
- If you wish to use a counterspindle on your machine, you need the software options ”travel to fixed stop” (Order No. 6FC5 800–0AM01–0YB0) and ”synchronous spindle” (Order No. 6FC5 800–0AM14–0YB0).
- With ShopTurn Open, you must not change the position of the following softkeys in the basic menu bar; i.e. in the REGIE.INI file these functions must always have been assigned a specific task.
  Task 0 (horizontal softkey 1): Operating area – Machine
  Task 1 (horizontal softkey 2): Operating area – Program manager
  Task 2 (horizontal softkey 3): Operating area – Program
  Task 4 (horizontal softkey 5): Operating area – Tools/Zero offsets
- With ShopTurn it is possible via TCU to use several operator panels.
- The HMI and Windows screen savers must not be used together.
  References: /IHA/, Commissioning Base Software and HMI Advanced, SINUMERIK 840D sl/840Di/840D;
  /IHE/ Commissioning Base Software and HMI Embedded sl, SINUMERIK 840D sl

■

Space for your notes

Reserved Functions

3

The following functions are utilized by ShopTurn and must not be assigned for other purposes.

PROG_EVENT

The system cycle PROG_EVENT.SPF is used by the standard cycles and by ShopTurn.
If you wish to use the cycle PROG_EVENT.SPF for user functions as well, it is necessary to implement these user functions in the cycles CYCPE_US.SPF and CYCPE1US.SPF. Save these cycles in the directory for user cycles or manufacturer cycles.

■

Space for your notes

Commissioning

4

4.1 Prerequisites

Data transfer

For data transfer you require:

- Hardware
  - Programming device with Windows XP or a PC with Ethernet
  - Memory stick, if required (Order No.: 6ES7 648–0DC20–0AA0)
- Software
  - SIMATIC Step7, version 5.3 SP2 or higher (for Order No., see SIMATIC catalog)

ShopTurn software

- ShopTurn on NCU (HMI Embedded sl)

With the ShopTurn variant on NCU (HMI Embedded sl), the software is already pre–installed on the CompactFlash Card:

  - Drive software (Sinamics)
  - PLC software
  - NCK software
  - ShopTurn
  - Cycles

The software is provided in 6 languages (German, English, French, Italian, Spanish, and Chinese).

- ShopTurn for PCU 50.3

The necessary software is located on the ”NCU–SysSW and ShopTurn–HMI” CD–ROM. The software is provided in 6 languages (German, English, French, Italian, Spanish, and Chinese).
The procedure for installing the software on PCU and NCK/PLC is described in the following sections.

Note

The NCU software components are located in file SIEMENSD.RTF (German)/SIEMENSE.RTF (English).
A compatibility list can be found in file 840D_sl_compatibility_list.xls.

4.2 First commissioning

4.2.1 Sequence

Before you begin commissioning, please read the general conditions and reserved functions.

Proceed as follows when commissioning:

- For installation of ShopTurn on PCU from CD (only for ShopTurn on PCU 50.3). For ShopTurn on NCU (HMI Embedded sl) the ShopTurn software is already pre–installed.
- PLC commissioning
- NCK commissioning
- Installing additional functions (optional)
- Adapting display machine data
- Customizing the operator interface (optional)
- Testing using the acceptance certificate

You can commission tool management either together with NCK and PLC commissioning or afterwards. If tool management is already set up on the machine, you only need to adapt the display machine data for tool management; (see Section 8.2 ”Commissioning sequence”).

A detailed description of commissioning is provided in the following documents:

References: /IDsl/, Commissioning CNC: NCK, PLC, drive, SINUMERIK 840D sl;
/IHA/, Commissioning Base Software and HMI Advanced, SINUMERIK 840D sl/840Di/840D;
/IHE/, Commissioning Base Software and HMI Embedded sl, SINUMERIK 840D sl

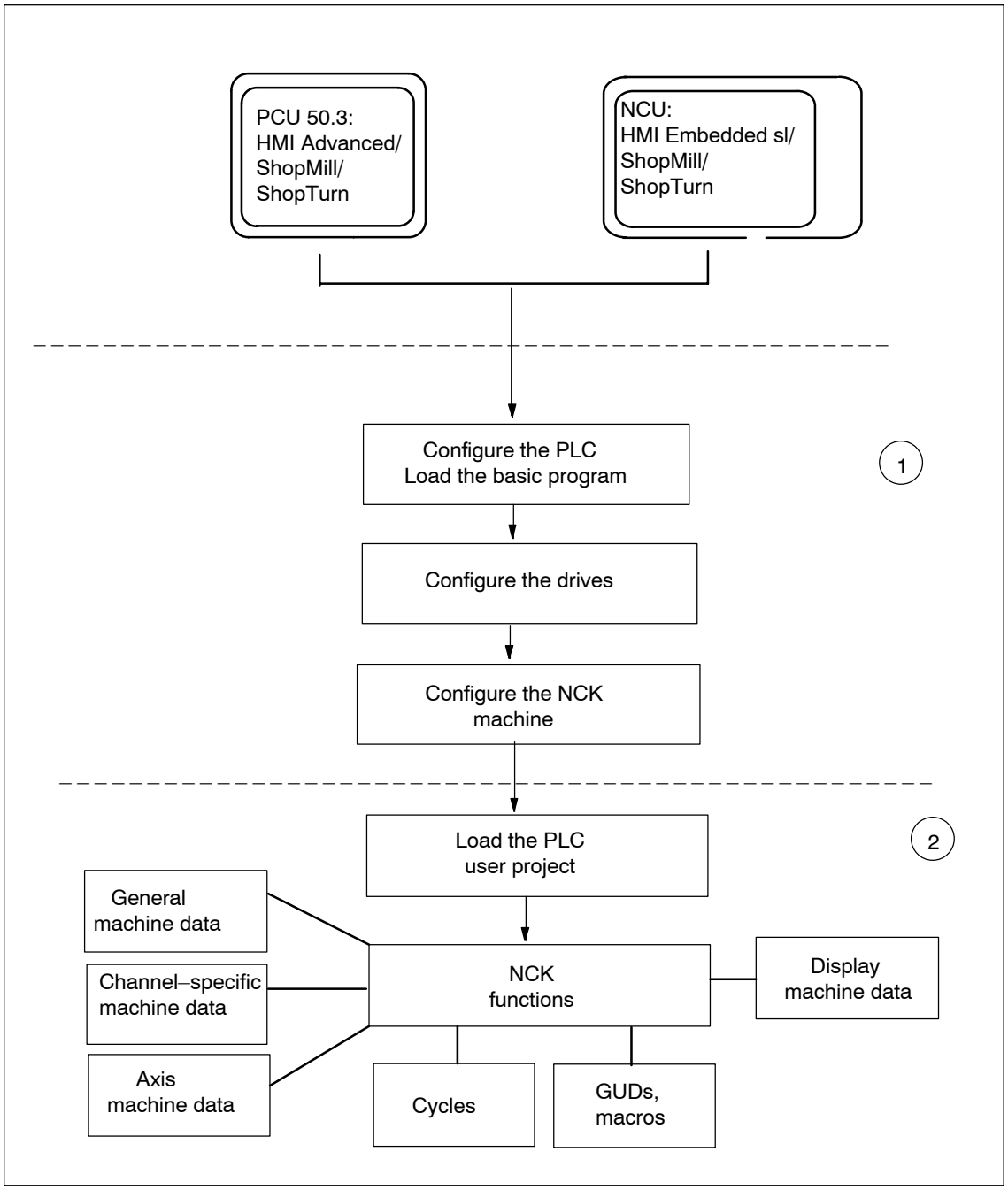


Fig. 4-1 General sequence of first commissioning

General sequence of first commissioning

(1) The first step when commissioning is to configure the control system, i.e. to set up communication between all involved components. PLC, drive and NCK are commissioned

References: /IDsl/, Commissioning CNC: NCK, PLC, drive, SINUMERIK 840D sl

(2) In a second step, the individual functions are commissioned in several runs. The commissioning sequence for these functions is described in the following sections.

4.2.2 Installation of ShopTurn on NCU (HMI Embedded sl)

For the purposes of operating ShopTurn on NCU (HMI Embedded sl) the whole software is pre–installed on the CompactFlash Card. Just a few adaptations of the NCK and PLC are needed.

Note

ShopTurn uses the alarm texts and PLC messages of the CNC ISO user interface. You can find detailed information on this subject as well as the the software installation in the following document:
References: /IHE/, Commissioning Base Software and HMI Embedded sl, SINUMERIK 840D sl

---

4.2.3 Installing ShopTurn on the PCU 50.3

The HMI Advanced software must be installed on the PCU 50.3 before you install ShopTurn.
Following installation, the internal HMI Embedded on the NCU must be deactivated. Deactivation and activation can be carried out with the aid of the WinSCP service program. Start WinSCP and select ”Open Terminal” under Commands.
The ”sc disable hmi” action deactivates the internal HMI Embedded.
The ”sc enable hmi” action activates the internal HMI Embedded.

References: /IDsl/, Commissioning Manual CNC: NCK, PLC, drive, SINUMERIK 840D sl;

/IHA/, Commissioning Base Software and HMI Advanced, SINUMERIK 840D sl/840Di/840D

You can install ShopTurn on the PCU 50.3 in one of three different ways:

- Installation via a network link
- Installation from a data carrier (e.g. DVD)
- Installation via USB interface

Note

ShopTurn uses the alarm texts and PLC messages of the CNC ISO user interface. You can find detailed information on this subject as well as the the software installation in the following document:
References: /IHE/, Commissioning Base Software and HMI Advanced, SINUMERIK 840D sl/840Di/840D

4.2.4 PLC commissioning

To commission the PLC, you must first create and then load a PLC user project.

To commission the PLC, proceed as follows:

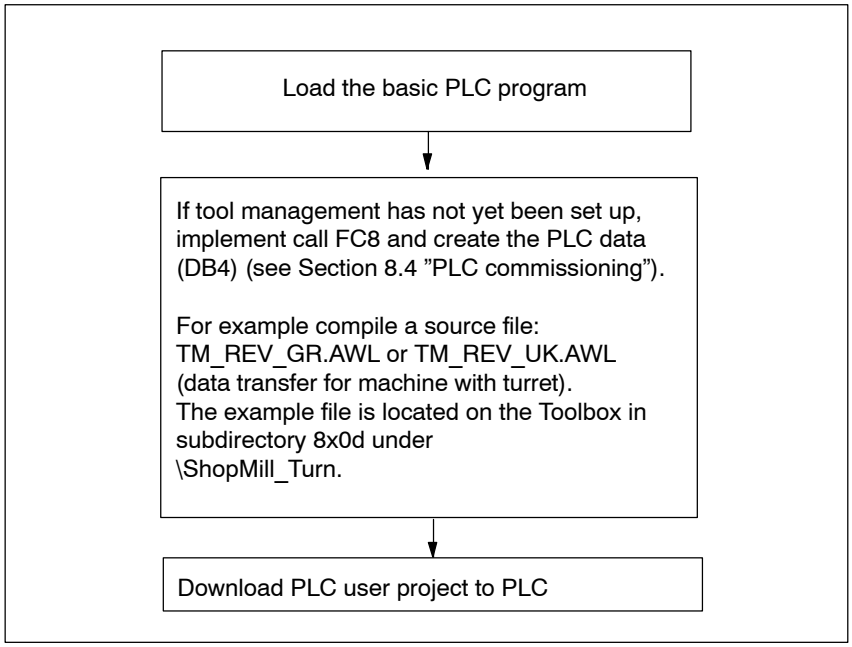


Fig. 4-2 PLC commissioning

4.2.5 NCK commissioning

NCK commissioning comprises the following points:

- Set up axes and spindle(s)
- Load ShopTurn machine data, definitions, and cycles
- Set up tool management

You only need to set up the axes and spindles if you have not yet done so on the machine. When setting up the axes and spindles, please observe the general conditions; (see Section 2 ”General conditions”).

Similarly, you only need to set up tool management in the NCK if this does not yet exist.

Note:
Example files are located in the directories CYCLES\SC\PROG\TEMPLATES_DEU and CYCLES\SC\PROG\TEMPLATES_ENG.

SINUMERIK 840D sl

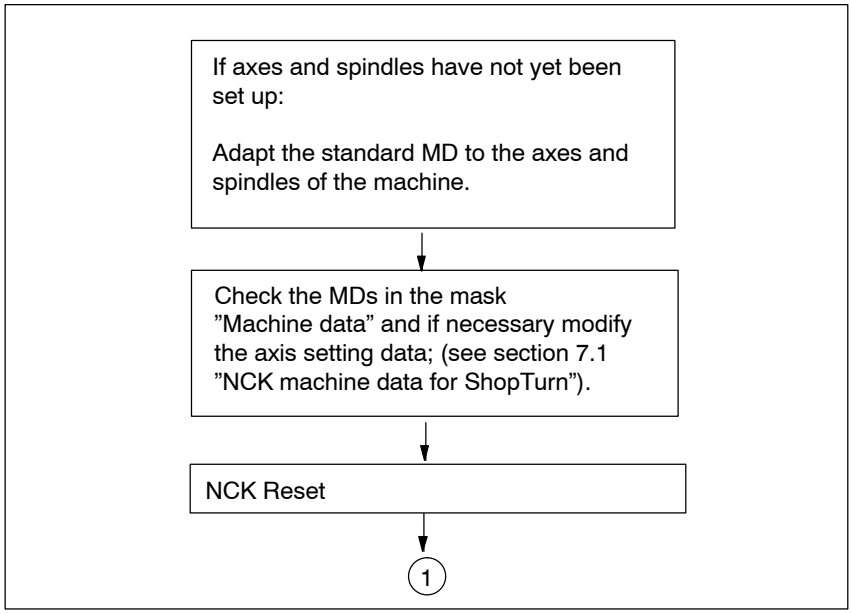


Fig. 4-3 NCK SINUMERIK 840D sl commissioning

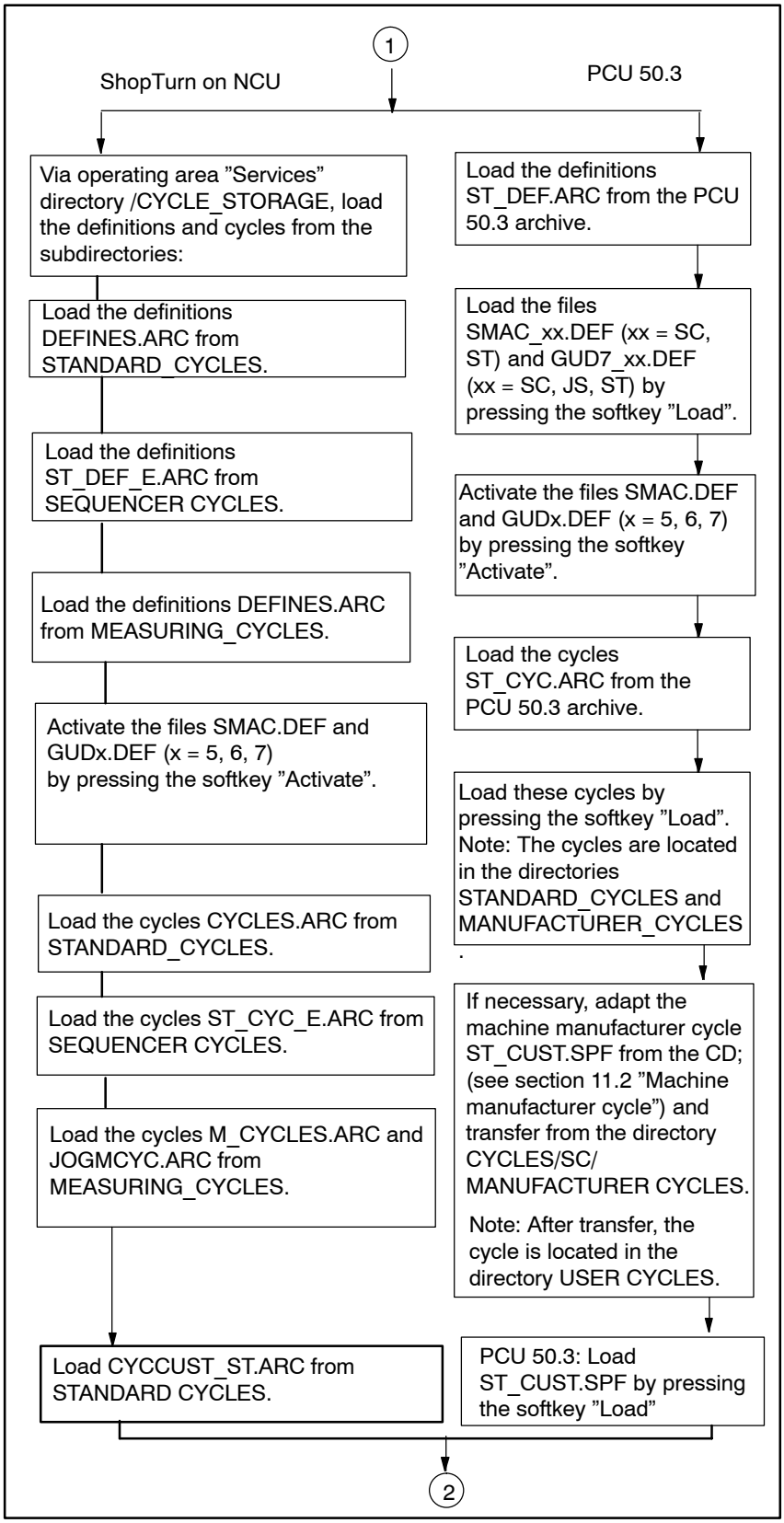


Fig. 4-4 NCK SINUMERIK 840D sl commissioning

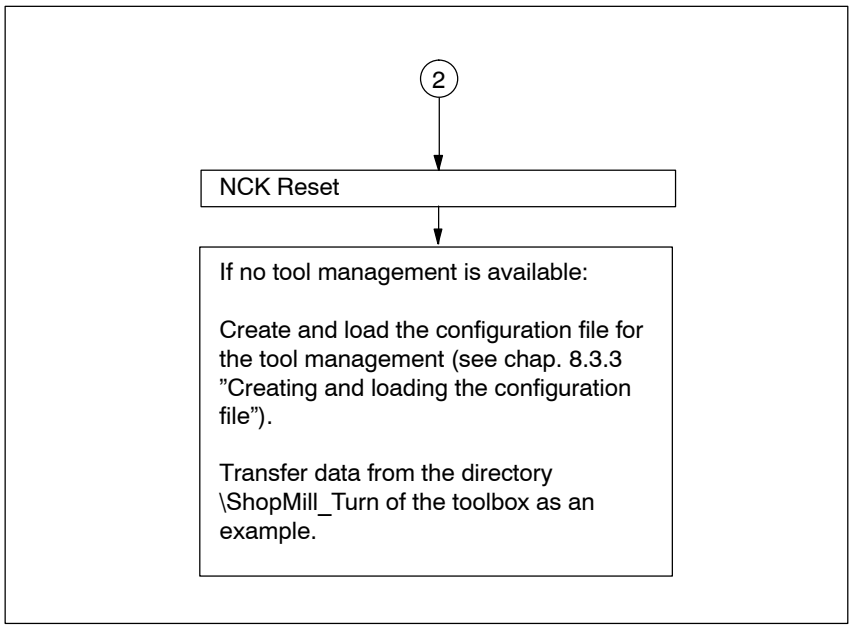


Fig. 4-5 NCK SINUMERIK 840D sl commissioning

4.2.6 Display machine data

Once you have completed installation of ShopTurn on the PCU and commissioning of NCK and PLC, you must adapt the display machine data.
The display machine data is listed in Section 7.2 ”Display machine data for ShopTurn”.

4.2.7 Acceptance report

The acceptance report can be used to test the installed ShopTurn functions once ShopTurn commissioning has been completed.
The acceptance report is included on the ShopTurn CD–ROM.

■

PLC Program

5

5.1 Structure of the PLC program

In the OBs 1, 40, and 100, the tool management and basic PLC program (FB 1, FC 2, ...) must be activated.

For a description of the function blocks and the basic PLC program please refer to:

References: /FB1/, Description of Functions, basic machine, P3 sl, "Basic PLC program"

5.2 Sample source files

ShopTurn includes a variety of source files for sample blocks. You can adapt and compile these source files or you can use your own blocks.

Table 5-1 Sample source files

Source	Mnemonics	Note	Block	Comment
TM_REV_GR. AWL	German	The indicated block numbers are pro-grammed in absolute terms.	FC 100	Sample block for config-uring tool management The block is called in OB100.
			FB 110	Sample block for data transfer of tool manage-ment. The block is called in OB1.
			DB 110	Instance data block for FB 110
TM_REV_UK. AWL	English	like TM_REV_GR.AWL		

Table 5-1 Sample source files

Source	Mnemonics	Note	Block	Comment
TM_REV_M_GR.AWL	German	The block designation is symbolic. For example, the following block numbers can be assigned.		
		TM_REV_M	FC55, Data type FC55	Sample block for tracking the turret position.
		DB_TM_REV_M	DB 55 Data type DB55	Data block
		FB4_INST_DB	DB56 Data type FB4	Instance data block for FB4
TM_REV_M_UK.AWL	English	like TM_REV_M_GR.AWL		

The sample blocks are located in the toolbox under \ShopMill_Turn.

5.3 Standard interface signals for/from ShopTurn

The following table lists the standard interface signals that are affected by the ShopTurn interface (DB19).

Table 5-2 Standard interface signals for/from ShopTurn

Byte	Designation
DB19	**Signals from operator panel (HMI——>PLC)**
DBB21	Active MMC operating area see section 6.1 ”HMI interface DB19”
DBX20 Bit6	Simulation active see section 6.1 ”HMI interface DB19”
DBW24	Current screen number in ShopTurn see section 6.1 ”HMI interface DB19”
DB21	**Signals to NCK channel (PLC——>NCK)**
DBX7.5	Deactivate global start lock see section 6.2 ”HMI interface DB21”

Note

- Feed disable must not be activated in the PLC user program while the spindle is stationary, because with ”Hole thread cutting with positioning on circle” the axes are positioned on the next hole as per the federate even with a stationary spindle.

■

Space for your notes

Signal Description

6

6.1 HMI interface DB19

DB19 **DBX20.6** Data block	E_SimActiv **Simulation active** Signal(s) from ShopTurn		
Edge evaluation: No		Signal(s) updated: Cyclically	Signal(s) valid from software version: ShopTurn 6.1
Significance of signal	0: Exit simulation 1: Start simulation		

DB19 **DBB21** Data block	E_AcitivWA **Active MMC operating area** Signal(s) from ShopTurn		
Edge evaluation: No		Signal(s) updated: Cyclically	Signal(s) valid from software version: ShopTurn 7.1
Significance of signal	OPEN (HMI Advanced) SK number (as predefined in the control (Task+1)) Classic (HMI Embedded) 201: ST machine 202: ST directory 203: ST program 204: ST alarms/messages 205: ST tool		

DB19 **DBW24** Data block	Mask number **Current screen number in ShopTurn** Signal(s) from ShopTurn		
Edge evaluation: No		Signal(s) updated: Cyclically	Signal(s) valid from software version: ShopTurn 7.5
Significance of signal	The signal outputs the screen number of the current ShopTurn screen. The following screen numbers can be output: **No. ShopTurn screen** Machine Manual mode (without ”Manual machine” option) 19 Basic screen 2 T, S, M... 21 Set ZO* 30 Workpiece zero 31 Workpiece zero – User screenform* 34 Workpiece zero – User screenform* 35 Workpiece zero – User screenform* 36 Workpiece zero – User screenform* 37 Workpiece zero – User screenform* 38 Workpiece zero – User screenform* 40 Workpiece zero – User screenform* 5 Workpiece zero – Measure edge Z 50 Measure tool 51 Measure tool – Manual– X/User screenform* 52 Measure tool – Manual– Z/User screenform* 53 Measure tool – Magnifying glass*/User screenform* 54 Measure tool – User screenform* 55 Measure tool – User screenform* 56 Measure tool – Calibration probe*/User screenform* 57 Measure tool – User screenform* 58 Measure tool – Autom. – Z* 59 Measure tool – Autom. – X* 4 Position 18 Face milling* 80 Stock removal* 81 Cycle start screen (stock removal/facemilling) – Accept with OK* 90 Tailstock user screenform* 91 Tailstock user screenform – Accept with OK* 1 ShopTurn settings Machine Manual mode (with ”Manual machine” option): 19 Basic screen 50 Measure tool 51 Measure tool – Manual– X/User screenform* 52 Measure tool – Manual – Z/User screenform* 53 Measure tool – Magnifying glass*/User screenform* 54 Measure tool – User screenform* 55 Measure tool – User screenform* 56 Measure tool – Calibration probe*/User screenform* 57 Measure tool – User screenform* 58 Measure tool – Autom. – Z* 59 Measure tool – Autom. – X* 21 Set ZO* 1300 Straight line		

DB19 **DBW24** Data block	Mask number **Current screen number in ShopTurn** Signal(s) from ShopTurn
Significance of signal	1400 Drilling 1410 Drilling – Center 1420 Drilling – Thread center 1433 Drilling – Centering* 1434 Drilling – Drilling* 1435 Drilling – Reaming* 1440 Drilling – Deep–hole drilling* 1453 Drilling – Tapping* 1454 Drilling– Thread milling* 1500 Turning 1513 Turning – Stock removal 1 1514 Turning – Stock removal 2 1515 Turning – Stock removal 3 1523 Turning – Groove 1 1524 Turning – Groove 2 1525 Turning – Groove 3 1533 Turning – Undercut form E 1534 Turning – Undercut form F 1535 Turning – Undercut thread DIN 1536 Turning – Undercut thread 1543 Turning – Thread longitudinal 1544 Turning – Thread taper 1545 Turning – Thread face 1550 Turning – Cut–off 1600 Milling* 1613 Milling – Rectangular pocket* 1614 Milling – Circular pocket* 1623 Milling – Rectangular spigot* 1624 Milling – Circular spigot* 1633 Milling – Longitudinal groove* 1634 Milling – Circumferential groove* 1640 Milling – Multi–edge* 1670 Milling – Engraving* 1730 Simulation – 3–window view* 1740 Simulation – Side view* 1750 Simulation – Front view* 1760 Simulation – Volume model* 1731 Simultaneous recording – 3–window view* 1741 Simultaneous recording – Side view* 1751 Simultaneous recording – Front view* 1761 Simultaneous recording – Volume model* 1777 Simulation settings 81 Cycle start screen (drilling/turning/milling) – Accept with OK* 90 Tailstock user screenform* 91 Tailstock user screenform – Accept with OK* 1 ShopTurn settings MDA mode: 20 MDA Machine Auto mode: 200 Basic screen 210 Program control 220 Block search 230 User screenform* 241 Simultaneous recording – Settings* 242 Simultaneous recording – 3–window view* 243 Simultaneous recording – Side view* 244 Simultaneous recording – Front view* 245 Simultaneous recording – Volume model* 250 Extended softkey menu – Setting

DB19 **DBW24** Data block	Mask number **Current screen number in ShopTurn** Signal(s) from ShopTurn		
Edge evaluation: No		Signal(s) updated: Cyclically	Signal(s) valid from software version: ShopTurn 7.5
Significance of signal	Operating area program manager: First softkey menu 300 Directory NC 310 Part program* 320 Subprogram* 330 User directory 1* 340 User directory 2* 350 User directory 3* 360 User directory 4* Second softkey menu 380 Standard cycles* 381 Manufacturer cycles* 382 User cycles* 383 User directory 5* 384 User directory 6* 385 User directory 7* 386 User directory 8* Operating area program: 400 Machining plan/G code editor 411 Simulation – Settings* 412 Simulation – 3–window view* 413 Simulation – Side view* 414 Simulation – Front view* 415 Simulation – Volume model* Operating area messages/alarms: 500 Messages 510 User screenform* 520 User screenform* Operating area tools/zero offsets: 600 Tool list 610 Tool wear 620 User tool list* 630 Magazine 640 Zero offset 650 R parameters 660 User screenform* 670 Spindles 680 User data 690 Machine data Run screen 910 Run screen in operating area Machine Manual* 920 Run screen in operating area Machine MDA* 930 Run screen in operating area Machine Auto* * = If screen is available		

6.2 HMI interface DB21

In ShopTurn a program can only be started by default in the machine area. Starting a program in any of the other areas (e.g. tools) is prevented by a global start lock.

Note

You can specify, via MD 9719, bit 9, that it should be possible to start a program from all masks.

In automated sequences, e.g. a start from the PLC as used on a machine with a pallet change system, this global start lock can be deactivated by means of interface signal DB21.DBX7.5.
This prevents a program start of this nature being locked by the operator interface.

DB21 **DBX7.5** Data block	suppressStartLock **Deactivate global start lock** PLC —> NCK		
Edge evaluation: No		Signal(s) updated: Cyclically	Signal(s) valid from software version: ShopTurn 7.1
Significance of signal	0: Do not cancel global start lock 1: Deactivate global start lock		

So that an alarm is output in the event of a start being attempted while the global start lock is activated, bit 6 in the machine data 16956 $MN_ENABLE_ALARM_MASK must be set.

6.3 Overview of former ShopTurn interface

With the new software version of ShopTurn the ShopTurn PLC program and the associated ShopTurn interface DB 82 are no longer used. The following tables show where you can find the old DB82 interface signals.

6.3.1 Signals to ShopTurn (input signals)

Table 6-1 Signals to ShopTurn (input signals)

Address DB82 DBX	Name Comments	Spares
0.0 – 0.7	CMM_IN.transfer_base_sig Transfer mode for MTTS signal	Omitted because ShopTurn PLC no longer exists
2.0	CMM_IN.base_sig.main_mode_mill.manual ShopTurn operating mode – manual	Transfer is via FC19/FC24 to the standard interface DB11.DBX0.2 JOG.
2.1	CMM_IN.base_sig.main_mode_mill.automatic ShopTurn operating mode – automatic	Transfer is via FC19/FC24 to the standard interface DB11.DBX0.0 AUTO.
4.0	CMM_IN.base_sig.reset Reset for ShopTurn	Transfer is via FC19/FC24 to the standard interface DB21.DBX7.7 Reset.
4.1	CMM_IN.base_sig.nc_cycle_start Cycle start	Transfer is via FC19/FC24 to the standard interface DB21.DBX7.1 NC–Start.
4.2	CMM_IN.base_sig.nc_cycle_stop Cycle stop	Transfer is via FC19/FC24 to the standard interface DB21.DBX7.3 NC–Stop.
6.0	CMM_IN.sub_mode_mill.tool Operating area – tool	Omitted. Use the appropriate key on the operator panel; (see section 10.6 OP hotkeys, PLC keys).
6.1	CMM_IN.sub_mode_mill.directory Operating area – Directory	Omitted. Use the appropriate key on the operator panel; (see section 10.6 OP hotkeys, PLC keys).
6.2	CMM_IN.sub_mode_mill.messages Operating area – Alarms/messages	Omitted. Use the appropriate key on the operator panel; (see section 10.6 OP hotkeys, PLC keys).
6.3	CMM_IN.sub_mode_mill.program Operating area – program	Omitted. Use the appropriate key on the operator panel; (see section 10.6 OP hotkeys, PLC keys).
6.4	CMM_IN.sub_mode_mill.oem1 Operating area – OEM1	n.a.
6.5	CMM_IN.sub_mode_mill.oem2 Operating area – OEM2	n.a.
6.6	CMM_IN.sub_mode_mill.customer Operating area – Customer	n.a.
6.7	CMM_IN.sub_mode_mill.mda Operating area – MDA	Transfer is via FC19/FC24 to the standard interface DB11.DBX0.1 MDA.
9.5	CMM_IN.program_extern_selected Program is selected in the PLC.	This is no longer required because the logic in the ShopTurn interface has been modified.

Table 6-1 Signals to ShopTurn (input signals)

Address DB82 DBX	Name Comments	Spares
9.6	CMM_IN.disable_cnc_standard Lock the switchover to CNC–ISO operator interface	ShopTurn Open (PCU 50.3): not available, ShopTurn on NCU (HMI Embedded) Assign protective levels to lock the remaining operator areas
9.7	CMM_IN.cmm_activ_in_cnc_mode ShopTurn PLC active during CNC–ISO operation	Omitted because ShopTurn PLC no longer exists.
10.0	CMM_IN.program_test_request Select function for program test	MMC –> PLC DB21.DBX25.7 PLC –> NCK DB21.DBX1.7 Connect as in the PLC user program.
10.1	CMM_IN.dry_run_request Select the function DryRun	MMC –> PLC DB21.DBX24.6 PLC –> NCK DB21.DBX0.6 Connect as in the PLC user program.
10.2	CMM_IN.m01_request Select function M01	MMC –> PLC DB21.DBX24.5 PLC –> NCK DB21.DBX0.5 Connect as in the PLC user program.
10.3	CMM_IN.skip_block_request Select function for skipping record	MMC –> PLC DB21.DBX26.0 ff PLC –> NCK DB21.DBX2.0 ff Connect as in the PLC user program
10.4	CMM_IN.boot_standard System boot in CNC–ISO operator interface	ShopTurn Open (PCU 50.3): Modify Power on task in the schedule, or remove the ShopTurn operating area from the schedule, or assign a protective level to the ShopTurn operating area in the schedule. ShopTurn on NCU (HMI Embedded): Assign a protective level to the ShopTurn operating area.
10.5	CMM_IN.nck_auto_req Prepare record search PLC	Omitted because operating areas Jog, Automatic, and MDA are now identical with the NCK operating modes
10.7	CMM_IN.ignore_nck_alarm Ignore NCK alarm in the event of cycle start	Omitted because the NCK start is no longer manipulated by ShopTurn
11.1	CMM_IN.get_tool_data Update tools data	Omitted because the data is automatically updated with the NCK function ”Extended tool counter”
11.2	CMM_IN.c_axis_feed_drive Separate feed drive as C axis drive	DB31ff.DBX56.0
11.3	CMM_IN.select_spindle_readout_0 Select spindle speed display, bit 0	DB31ff.DBX56.1 All bits 0: The main spindle rule applies More than 1 bit active: The following sequence applies: 1. Main spindle 2. Rotating tool spindle 3. Counterspindle

Table 6-1 Signals to ShopTurn (input signals)

Address DB82 DBX	Name Comments	Spares
11.4	CMM_IN.select_spindle_readout_1 Select spindle speed display, bit 1	DB31ff.DBX56.1 All bits 0: The main spindle rule applies More than 1 bit active: The following sequence applies: 1. Main spindle 2. Rotating tool spindle 3. Counterspindle
11.5	CMM_IN.drf_request Select the function DRF	MMC –> PLC DB21.DBX24.3 PLC –> NCK DB21.DBX0.3 Connect as in the PLC user program.

6.3.2 Signals from ShopTurn (output signals)

Table 6-2 Signals from ShopTurn (output signals)

Address DB82 DBX	Name Comments	Spares
30.0	CMM_OUT.base_sig.main_mode_mill.manual ShopTurn operating mode – manual	Transfer is via FC19/FC24 to the standard interface DB11.DBX6.2 JOG
30.1	CMM_OUT.base_sig.main_mode_mill.automatic ShopTurn operating mode – automatic	Transfer is via FC19/FC24 to the standard interface DB11.DBX6.0 AUTO
32.0	CMM_OUT.base_sig.reset Reset performed	Can be simulated with the interface signal DB21.DBX35.7 channel status reset
32.1	CMM_OUT.base_sig.nc_cycle_activ Cycle active	Transfer is via FC19/FC24 to the standard interface DB21.DBX
32.2	CMM_OUT.base_sig.nc_cycle_activ Cycle interrupted	Transfer is via FC19/FC24 to the standard interface DB21.DBX
34.0	CMM_OUT.sub_mode_mill.tool Operating area – Tool is selected	Standard interface signal DB19.DBB21 = 205
34.1	CMM_OUT.sub_mode_mill.directory Operating area – Directory is selected	Standard interface signal DB19.DBB21 = 202
34.2	CMM_OUT.sub_mode_mill.messages Operating area – Alarms/Messages is selected	Standard interface signal DB19.DBB21 = 204
34.3	CMM_OUT.sub_mode_mill.program Operating area – Program is selected	Standard interface signal DB19.DBB21 = 203
34.7	CMM_OUT.sub_mode_mill.mda Operating area – MDA is selected	Transfer is via FC19/FC24 to the standard interface DB11.DBX6.1 MDA
36.0	CMM_OUT.cmm_plc_activ ShopTurn PLC active	Omitted because ShopTurn PLC no longer exists
36.1	CMM_OUT.cmm_mmc_activ ShopTurn operator interface active	ShopTurn Open (PCU 50.3): is not evaluated ShopTurn on NCU (HMI Embedded): DB19.DBB21
36.7	CMM_OUT.ext_prog_sel External processing program is selected	n.a.

Table 6-2 Signals from ShopTurn (output signals)

Address DB82 DBX	Name Comments	Spares
37.0	CMM_OUT.program_selection_done Acknowledgment from HMI that a program has been selected	Omitted because of the new start lock logic of the NCK. The program can be selected and started directly.
37.1	CMM_OUT.program_test_activ Function for program test is active	Standard interface signal DB21.DBX33.7
37.2	CMM_OUT.dry_run_activ Function for DryRun is active	Standard interface signal DB21.DBX318.6
37.3	CMM_OUT.m01_activ Function M01 is active	Standard interface signal DB21. DBX32.5
37.4	CMM_OUT.skip_block_activ Function for skipping record is active	Standard interface signal DB21. DBX26.0ff
37.7	CMM_OUT.start_up_activ ShopTurn boot active	n.a.
38.1	CMM_OUT.tool_un_load_internal Load/unload tool without moving magazine	Standard interface signal DB71.DBX32.0 for the1st loading point
38.2	CMM_OUT.drf_activ Function DRF is active	Standard interface signal DB21.DBX24.3
38.3	CMM_OUT.nc_start_ineffective NC start has no effect	Global start lock can be deactivated. Standard interface signal DB21.DBX7.5
44	CMM_OUT.mask_number Current screen number in ShopTurn	Standard interface signal DB19.DBW24

Space for your notes

Machine Data

7

7.1 NCK machine data for ShopTurn

When commissioning the NCK, all relevant NCK machine data (including tool management data) must be set with the ShopTurn–specific values.

In the operating area ”Tools – zero” open the mask ”Machine data” by pressing the softkey ”Machine data” in the extended softkey bar. Here all the machine data needed for ShopTurn is displayed with information regarding setpoint and actual values.
Correctly set machine data is marked with a tick. Machine data marked with an exclamation point must be corrected accordingly. A red exclamation point in the header (No., MD, Set, Actual) will indicate when it is necessary to correct machine data.

The following symbols indicate the rule for setpoint to actual value:

- =: must be exact
- ≤: must be at least
- &: certain bits must be exact

If no rule is indicated for setpoint to actual value, it is only a suggestion.

NCK machine data with exact values must be set as specified. NCK machine data with minimum values can be adapted to the
specific features of your machine.

The activation mode for each item of machine data is indicated in the column after the actual value display.

- po: Power On (softkey ”NCK reset”)
- cf: Configuration (softkey ”Set MD effective”)
- so: Immediately (no action required)
- re: Reset (”Reset” key on machine control panel)

Note

Files SIEMENSD.RTF and SIEMENSE.RTF list the necessary settings for the ShopTurn machine data. You are advised to print these out. You can then check and if necessary correct the specific values more conveniently.

"Use the "Actual = Set" softkey to adjust incorrect values so that they meet the minimum ShopTurn requirements. These are confirmed when you click "OK" to acknowledge the query that follows.
If, for instance, the actual value is too low, the value of the machine data will be set to the lowest setpoint. For bit masks only the missing bits are set.

Caution

The tool length corrections are set in ShopTurn (setting data 42940 $SC_TOOL_LENGTH_CONST and 42950 $SC_TOOL_LENGTH_TYPE); length 1 always refers to the X direction and length 2 always refers to the Z direction irrespective of machining layer and cutting position.

Note

Please note that the machine data record also contains machine data for configuring the memory.

For an exact description of all NCK machine data please refer to:
References: /LIS1sl/, lists (book 1);
/LIS2sl/, lists (book 2);
/IDsl/, Commissioning CNC: NCK, PLC, drive, SINUMERIK 840D sl;
/FB/, Description of Functions

Setup feed rate

Use the SD 42600 JOG_FEED_PER_REV_SOURCE and SD 43300 $SA_ASSIGN_FEED_PER_REV_SOURCE setting data of the relevant axes to specify which setup feedrate will be evaluated in manual mode.

- SD 42600 JOG_FEED_PER_REV_SOURCE = –3 and
  SD $SA_ASSIGN_FEED_PER_REV_SOURCE = –3:
  When the main spindle is rotating, the JOG function uses the revolutional feedrate. When the spindle is stationary, the feedrate for each minute is used.

- SD 42600 JOG_FEED_PER_REV_SOURCE = 0 and
  SD $SA_ASSIGN_FEED_PER_REV_SOURCE = 0:
  The feedrate used depends on the setting data
  SD 41100 JOG_REV_IS_ACTIVE:
  Bit 0 = 0 setup feedrate in mm/min
  (with rotating and stationary spindle)
  Bit 0 = 1 setup feedrate in mm/rev

The setup feed rates can be entered in the mask "Machine – manual" → ">" → "ShopTurn settings".

Use the option "Manual machine", enter the feed in the basic mask "Manual".
References: /BATsl/, Operation/Programming ShopTurn

7.2 Display machine data for ShopTurn

Once ShopTurn has been installed on the PCU and commissioning of the NCK and PLC is complete, you still need to adapt the display machine data.
In any case, you must check the settings
of the display machine data that are identified in table 7-1 by means of "*".

7.2.1 Overview of display machine data

Table 7-1 Display machine data for ShopTurn

MD number	MD identifier	Comment	Preset default
9014	$MM_USE_CHANNEL_DISPLAY_DATA	Use channel–specific display machine data	0
9020	$MM_TECHNOLOGY	Basic configuration turning/milling	1
9422	$MM_MA_PRESET_MODE	Preset/basic offset in JOG	1
9428	$MM_MA_SPIND_MAX_POWER	Factor for display of spindle utilization	100
9429	$MM_MA_SPIND_POWER_RANGE	Display area for spindle utilization	200
9450	$MM_WRITE_TOA_FINE_LIMIT	Limit value for wear fine	0.999
9451	$MM_WRITE_ZOA_FINE_LIMIT	Limit value for fine adjustment	0.999
9460	$MM_PROGRAM_SETTINGS	Settings in the Program area	HD
9478*	$MM_TO_OPTION_MASK	Settings for ShopTurn	1
9550*	$MM_CTM_CYC_ROUGH_RELEASE_DIST	Retraction distance for stock removal at a contour	1
9551*	$MM_CTM_CYC_ROUGH_RELEASE_ANGLE	Retraction angle for stock removal at a contour	45
9552*	$MM_CTM_CYC_ROUGH_BLANC_OFFS	Blank offset for stock removal at a contour	1
9553*	$MM_CTM_CYC_ROUGH_TRACE_ANGLE	Starting angle for tracing a contour	5
9554	$MM_CTM_CYC_ROUGH_MIN_REST_MAT_1	Starting thickness, with reference to the final machining allowance, for machining residual material (axis 1)	50
9555	$MM_CTM_CYC_ROUGH_MIN_REST_MAT_2	Starting thickness, with reference to the final machining allowance, for machining residual material (axis 2)	50
9556	$MM_CTM_CYC_ROUGH_VAR_DEPTH	Percentage for variable cutting depth when turning a contour	20
9557	$MM_CTM_CYC_ROUGH_FEED_INT_TIME	Feed interruption time when turning a contour	–1
9558	$MM_CTM_CYC_ROUGH_INT_REL_DIST	Retraction path feed interruption Contour turning	0
9560	$MM_CTM_TURN_GROOV_TOOL_BEND	Retraction due to tool bending when plunge–turning a groove	0.1
9561	$MM_CTM_TURN_GROOV_FREE_CUT_VAL	Retraction depth before turning operation when plunge–turning a groove	0.1
9606	$MM_CTM_SIMULATION_TIME_NEW_POS	Simulation updating rate of actual value	350
9611	$MM_CTM_CROSS_AX_DIAMETER_ON	Diameter display for active transverse axes	1
9619	$MM_CTM_G91_DIAMETER_ON	Incremental infeed	0
9621	$MM_CTM_CYCLE_DWELL_TIME	Dwell time for cycles	–1
9626	$MM_CTM_TRACE	Settings for ShopTurn	1
9630	$MM_CTM_FIN_FEED_PERCENT	Finishing feed in percent	100
9639	$MM_CTM_MAX_TOOL_WEAR	Upper input limit for tool wear	1
9640	$MM_CTM_ENABLE_CALC_THREAD_PITCH	Calculation of thread depth if pitch entered	0

MD number	MD identifier	Comment	Preset default
9646	$MM_CTM_FACTOR_O_CALC_THR_PITCH	Factor for calculating the external thread depth if pitch entered	0.6134
9647	$MM_CTM_FACTOR_I_CALC_THR_PITCH	Factor for calculating the internal thread depth if pitch entered	0.5413
9648	$MM_CTM_ROUGH_O_RELEASE_DIST	Retraction distance for stock removal/cutting by external machining	1.0
9649	$MM_CTM_ROUGH_I_RELEASE_DIST	Retraction distance for stock removal/cutting by internal machining	0.5
9650*	$MM_CMM_POS_COORDINATE_SYSTEM	Position of coordinate system	34
9651*	$MM_CMM_TOOL_MANAGEMENT	Tool management variant	4
9652*	$MM_CMM_TOOL_LIFE_CONTROL	Tool monitoring	1
9654	$MM_CMM_SPEED_FIELD_DISPLAY_RES	Number of decimal places in the speed input field	0
9657	$MM_CMM_CYC_MIN_CONT_PO_TO_RAD	Variation of smallest possible cutter radius in percent	5
9658	$MM_CMM_CYC_MAX_CONT_PO_TO_RAD	Variation of largest possible cutter radius	0.01
9663	$MM_CMM_TOOL_DISPLAY_IN_DIAM	Display radius/diameter for tool	1
9664	$MM_CMM_MAX_INP_FEED_P_MIN	Maximum feed in mm/min	10000.0
9665	$MM_CMM_MAX_INP_FEED_P_ROT	Maximum feed in mm/rev	1.0
9666	$MM_CMM_MAX_INP_FEED_P_TOOTH	Maximum feed in mm/tooth	1.0
9667*	$MM_CMM_FOLLOW_ON_TOOL_ACTIVE	Tool preselection active	0
9668*	$MM_CMM_M_CODE_COOLANT_I_AND_II	M function coolant I and II	–1
9669	$MM_CMM_FACE_MILL_EFF_TOOL_DIAM	Effective cutter diameter for face milling	85.0
9670	$MM_CMM_START_RAD_CONTOUR_POCKE	Radius of approach circle for finishing contour pockets plus half the final machining allowance (–1 = safety clearance)	–1.0
9671	$MM_CMM_TOOL_LOAD_DEFAULT_MAG	Load tool in default magazine	0
9672*	$MM_CMM_FIXED_TOOL_PLACE	Fixed location coding	1
9673*	$MM_CMM_TOOL_LOAD_STATION	Number of loading point	1
9674	$MM_CMM_ENABLE_TOOL_MAGAZINE	Display of magazine list	1
9675	$MM_CMM_CUSTOMER_START_PICTURE	Customized boot screen	0
9676*	$MM_CMM_DIRECTORY_SOFTKEY_PATH1	Path for drive names in directory management	–
9677*	$MM_CMM_DIRECTORY_SOFTKEY_PATH2	Path for drive names in directory management	–
9678*	$MM_CMM_DIRECTORY_SOFTKEY_PATH3	Path for drive names in directory management	–
9679*	$MM_CMM_DIRECTORY_SOFTKEY_PATH4	Path for drive names in directory management	–
9680*	$MM_CMM_M_CODE_COOLANT_I	M function coolant I	8
9681*	$MM_CMM_M_CODE_COOLANT_II	M function coolant II	7
9686*	$MM_CMM_M_CODE_COOLANT_OFF	M function for coolant OFF	9
9687	$MM_CMM_TOOL_MOVE_DEFAULT_MAG	Reload tool in default magazine	0
9718*	$MM_CMM_OPTION_MASK_2	Settings for ShopTurn	2
9719*	$MM_CMM_OPTION_MASK	Settings for ShopTurn	H5
9724	$MM_CMM_CIRCLE_RAPID_FEED	Rapid feed for positioning on circular path	5000
9725	$MM_CMM_ENABLE_QUICK_M_CODES	Enable fast M functions	0
9729	$MM_CMM_G_CODE_TOOL_CHANGE_PROG	Program name for tool change in the G code	–
9749*	$MM_CMM_ENABLE_MEAS_T_AUTO	Enable automatic tool measurement	1
9751*	$MM_CMM_MEAS_T_PROBE_INPUT	Measuring input for tool probe	0
9754	$MM_CMM_MEAS_DIST_TOOL_LENGTH	Maximum measurement distance for tool length for rotating spindle	10
9759	$MM_CMM_MAX_CIRC_SPEED_ROT_SP	Maximum circumferential speed for tool measurement for rotating spindle	100

MD number	MD identifier	Comment	Preset default
9760	$MM_CMM_MAX_SPIND_SPEED_ROT_SP	Maximum speed for tool measurement for rotating spindle	1000
9771	$MM_CMM_MAX_FEED_ROT_SP	Maximum feed for tool measurement for rotating spindle	20
9772	$MM_CMM_T_PROBE_MEASURING_DIST	Measurement distance for tool measurement with stationary spindle	10
9773	$MM_CMM_T_PROBE_MEASURING_FEED	Feed rate for tool measurement with stationary spindle	300
9777	$MM_CMM_ENABLE_TIME_DISPLAY	Control for time display	0x7F
9802	$MM_ST_INDEX_AXIS_C_SUB	Axis index for separate C axis of counterspindle	0
9803*	$MM_ST_INDEX_AXIS_4	Axis index for 4th axis	5
9804*	$MM_ST_INDEX_SPINDLE_MAIN	Axis index for main spindle	3
9805*	$MM_ST_INDEX_SPINDLE_TOOL	Axis index for tool spindle	4
9806*	$MM_ST_INDEX_SPINDLE_SUB	Axis index for counter–spindle	6
9807*	$MM_ST_INDEX_AXIS_C	Axis index for C axis	3
9808*	$MM_ST_INDEX_AXIS_B	Axis index for B axis	0
9810*	$MM_ST_GEAR_STEPS_SPINDLE_MAIN	Number of gears on main spindle	0
9811*	$MM_ST_GEAR_STEPS_SPINDLE_TOOL	Number of gears on tool spindle	0
9812*	$MM_ST_GEAR_STEPS_SPINDLE_SUB	Number of gears on counter–spindle	0
9820	$MM_ST_MAGN_GLASS_POS_1	Position of magnifying glass for measuring tool, 1st axis	0
9821	$MM_ST_MAGN_GLASS_POS_2	Position of magnifying glass for measuring tool, 2nd axis	0
9822*	$MM_ST_DISPL_DIR_MAIN_SPIND_M3	Displayed direction of rotation for main spindle with M3	0
9823*	$MM_ST_DISPL_DIR_SUB_SPIND_M3	Displayed direction of rotation for counter–spindle with M3	0
9824*	$MM_ST_DISPL_DIR_MAIN_C_AX_INV	Displayed direction of rotation for C axis main spindle with M3	0
9825*	$MM_ST_DISPL_DIR_SUB_C_AX_INV	Displayed direction of rotation for C axis counter–spindle with M3	0
9826*	$MM_ST_DEFAULT_DIR_TURN_TOOLS	Main direction of rotation for all turning tools	3
9827*	$MM_ST_DEFAULT_MACHINING_SENSE	Basic settings for machining direction – milling	0
9828*	$MM_ST_MEAS_T_PROBE_INPUT_SUB	Input number for tool probe for counterspindle	1
9829	$MM_ST_SPINDLE_CHUCK_TYPES	Type of spindle chuck	0
9830	$MM_ST_SPINDLE_PARA_ZL0	Chuck dimension for main spindle	0
9831	$MM_ST_SPINDLE_PARA_ZL1	Chuck dimension for counter–spindle	0
9832	$MM_ST_SPINDLE_PARA_ZL2	Stop dimension for counter–spindle	0
9833	$MM_ST_SPINDLE_PARA_ZL3	Jaw dimension for counter–spindle	0
9836	$MM_ST_TAILSTOCK_DIAM	Tailstock diameter	0
9837	$MM_ST_TAILSTOCK_LENGTH	Tailstock length	0
9838	$MM_ST_BORDER_TOOL_LEN_X_REV_2	Limit value of tool length X for the 2nd turret	0
9840*	$MM_ST_ENABLE_MAGN_GLASS	Magnifying glass function in manual: Measuring a tool	0
9841*	$MM_ST_ENABLE_PART_OFF_RECEPT	Enable receptacle function for cut–off	0
9842*	$MM_ST_ENABLE_TAILSTOCK	Enable tail stock	0
9843*	$MM_ST_ENABLE_SPINDLE_CLAMPING	Enable spindle clamping (C axis)	0
9849	$MM_ST_CYCLE_SUB_SP_PARK_POS_Y	Parking position of Y axis with counterspindle	0

MD number	MD identifier	Comment	Preset default
9850	$MM_ST_CYCLE_THREAD_RETURN_DIST	Return distance for thread turning	2
9851*	$MM_ST_CYCLE_SUB_SP_WORK_POS	Retraction position Z for counter–spindle	0
9852	$MM_ST_CYCLE_SUB_SP_DIST	Distance as of which feed rate is used when approaching fixed stop with counter–spindle	10
9853	$MM_ST_CYCLE_SUB_SP_FEED	Feed rate for traveling to fixed stop with counter–spindle	0
9854	$MM_ST_CYCLE_SUB_SP_FORCE	Force in percent for traveling to fixed stop with counter–spindle	10
9855	$MM_ST_CYCLE_TAP_SETTINGS	Settings for thread tapping	0
9856	$MM_ST_CYCLE_TAP_MID_SETTINGS	Settings for center tapping	0
9857	$MM_ST_CYCLE_RET_DIST_FIXEDSTOP	Retraction distance before clamping after traveling to fixed stop	0
9858	$MM_ST_CYCLE_RET_DIST_PART_OFF	Retraction distance before cut–off with counter–spindle	0
9859	$MM_ST_CYCLE_PART_OFF_CTRL_DIST	Distance for cut–off check	0.1
9860	$MM_ST_CYCLE_PART_OFF_CTRL_FEED	Feed for cut–off check	0
9861	$MM_ST_CYCLE_PART_OFF_CTRL_FORC	Force in percent for cut–off check	10
9862	$MM_ST_CYC_DRILL_MID_MAX_ECCENT	Maximum center offset for center drilling	0.5
9863	$MM_ST_MAX_INP_AREA_GAMMA	Maximum input area for gamma alignment angle	5
9897	$MM_ST_OPTION_MASK_MAN_FUNC	Settings for ShopTurn manual functions	0
9898*	$MM_ST_OPTION_MASK	Settings for ShopTurn	H7000
9899*	$MM_ST_TRACE	Settings for ShopTurn	0

7.2.2 Description of display machine data

9014 MD number	$MM_USE_CHANNEL_DISPLAY_DATA Use channel–specific display machine data		
Default setting: 0	Min. input limit: 0	Max. input limit: 1	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: LONG		Valid as of software version: ShopTurn 6.4	
Significance:	In this MD you stipulate whether you want to use channel–specific display machine data. 0 = No channel–specific display machine data 1 = Channel–specific display machine data Note: With ShopTurn this must be set to MD = 0.		

9020 MD number	$MM_TECHNOLOGY Basic configuration turning/milling		
Default setting: 1	Min. input limit: 0	Max. input limit: 2	
Change becomes effective after: POWER ON	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version: ShopTurn 6.1	
Significance:	In this MD you stipulate the basic configuration for simulation and free contour programming. 0 = No specific configuration 1 = Turning machine configuration 2 = Milling machine configuration		

9422 MD number	$MM_MA_PRESET_MODE Preset/basic offset in JOG		
Default setting: 1	Min. input limit: 0	Max. input limit: 3	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.4	
Significance:	In this MD you stipulate the behavior of the function ”Set zero offset” in ”Machine – manual” mode. ≠ 2: Zero point is saved in the currently active zero offset, in other cases it is saved in the basic offset = 2: Zero point is saved in basic offset		

9428 MD number	$MM_MA_SPIND_MAX_POWER Maximum value of the spindle performance display		
Default setting: 100	Min. input limit: 100	Max. input limit: ***	
Change becomes effective after: POWER ON	Protection level: 3/4	Units: %	
Data type: WORD		Valid as of software version ShopTurn 7.1	
Significance:	In this MD enter the factor by which the supplied spindle utilization will be multiplied.		

9429 MD number	$MM_MA_SPIND_POWER_RANGE Display area for spindle utilization		
Default setting: 200	Min. input limit: 100	Max. input limit: ***	
Change becomes effective after: POWER ON	Protection level: 3/4	Units: %	
Data type: WORD		Valid as of software version ShopTurn 7.1	
Significance:	In this MD you stipulate the display range of the bar displaying spindle utilization. Depending upon the value entered, the displayed percentage values and the extension of the color areas change. Value entered = 100: Percentage values 0, 80, and 100% are displayed. The color display changes from green to red starting at 80%. Value entered = > 100, e.g. 200: Percentage values 0, 100, and 200% are displayed. The colored display changes from green to red as of 100%.		

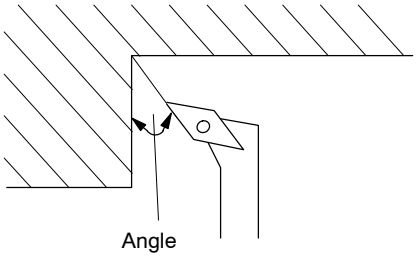
9450 MD number	$MM_WRITE_TOA_FINE_LIMIT Limit value for wear fine		
Default setting: 0.999	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version ShopTurn 6.4	
Significance:	With this MD you define the upper incremental limit (limit value for wear fine) for tool wear (length, radius). The incremental upper limit is only effective if the active protection level is greater than the protection level set in MD 9203 USER_CLASS_WRITE_FINE. The absolute upper limit is set in MD 9639 $MM_CTM_MAX_TOOL_WEAR.		

9451 MD number	$MM_WRITE_ZOA_FINE_LIMIT Limit value for fine adjustment		
Default setting: 0.999	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version ShopTurn 6.4	
Significance:	When entering the fine adjustment the difference between the previous value and the new value must not exceed the value specified in this MD.		

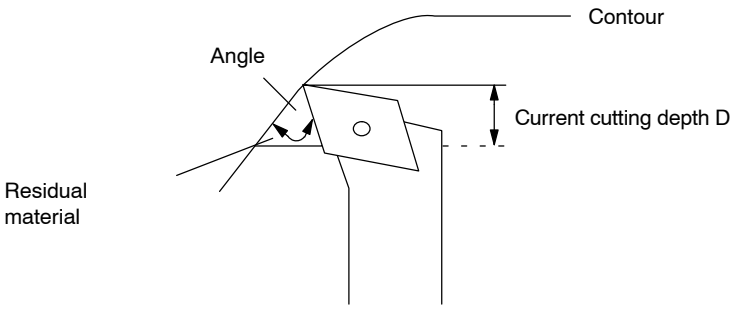
9460 MD number	$MM_PROGRAM_SETTINGS Settings in the Program area		
Default setting: HD	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: Hex	
Data type: LONG		Valid as of software version ShopTurn 6.4	
Significance:	Bit 0 to 4: Reserved Bit 5: Display hidden lines (;*HD*) in the G code editor Bit 6: Reserved Bit7: Switch off G program checking during mold making		

9478 MD number	**$MM_TO_OPTION_MASK** **Settings for ShopTurn**		
Default setting: 1	Min. input limit: 0000	Max. input limit: FFFF	
Change becomes effective after: POWER ON	Protection level: 1	Units: Hex	
Data type: LONG		Valid as of software version ShopTurn 7.5	
Significance:	Bit 0: Display tool parameters "Number of teeth", "Spindle" and "Coolant" in the tool management. Bit 1: Reserved Bit 2: Display additional list in the tool management. Bit 3: Disable creation of new tools directly on a magazine location. Bit 4 to bit 6: Reserved Bit7: Disable editing of tool parameters (tool type, tool name) if the tools are in the magazine. Bit 8: Evaluate file TO_TURN.INI for configuration of the tool management user interface. Bit 9: Disable loading/unloading of tools if a program is being executed on the machine. Bit 10: Calculate by adding tool wear entries. Bit 11 and bit 12: Reserved Bit 13: Display intermediate memory (spindle and dual gripper). Bit 14: Reserved Bit 15: Disable loading/unloading of tools in/out of spindles. Bit 16: Do not delete tool wear when entering a geometric value. Bit 17: Hide tool relocation Bit 18: Hide magazine positioning. Bit 19: With Multifix steel holder, tool can be selected in tool list. Bit 20: Spindle is loaded directly. Bit 21: Basic length is displayed in "Tool orientation" detail display. Bit 22: 3D and edge probes not unloaded with "Unload all" function. Bit 23: Reserved Bit 24: Magazine positioning in the tool wear list. Bit 25: Magazine number is always displayed in the loading and relocation dialog.		

9550 MD number	**$MM_CTM_CYC_ROUGH_RELEASE_DIST** **Retraction distance for stock removal at a contour**		
Default setting: 1	Min. input limit: 0	Max. input limit: 10	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version: ShopTurn 6.3	
Significance:	This MD is used to stipulate the distance by which both axes are lifted off the contour during rough cutting operations. This value also applies to stock removal, plunge–cutting, and groove turning.		

9551 MD number	$MM_CTM_CYC_ROUGH_RELEASE_ANGLE **Retraction angle for stock removal at a contour**		
Default setting: 45	Min. input limit: 0	Max. input limit: 90	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: Degrees	
Data type: DOUBLE		Valid as of software version: ShopTurn 6.3	
Significance:	This MD is used to stipulate the angle at which axes are lifted off the contour during rough cutting operations. This value also applies to stock removal, plunge–cutting, and groove turning.  		

9552 MD number	$MM_CTM_CYC_ROUGH_BLANC_OFFS **Blank offset for stock removal at a contour**		
Default setting: 1	Min. input limit: 0	Max. input limit: 100	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version: ShopTurn 6.3	
Significance:	This MD is used to stipulate the distance from the blank at which G0 is switched over to G1 during stock removal at a contour to compensate for any blank allowances. This value also applies to stock removal, plunge–cutting, and groove turning.		

9553 MD number	$MM_CTM_CYC_ROUGH_TRACE_ANGLE **Starting angle for tracing a contour**		
Default setting: 5	Min. input limit: 0	Max. input limit: 90	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: Degrees	
Data type: DOUBLE		Valid as of software version: ShopTurn 6.3	
Significance:	This MD is used to stipulate the angle between the cutting edge and the contour as of which, when removing stock from an edge or a contour (rough cutting), rounding will automatically take place to remove any residual material. If the angle of the residual material is greater than that specified in the MD, the tool will round the contour.  		

9554 MD number	$MM_CTM_CYC_ROUGH_MIN_REST_MAT1 **Starting thickness, with reference to the final machining allowance, for machining residual material (axis 1)**		
Default setting: 50	Min. input limit: 0	Max. input limit: 1000	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: %	
Data type: DOUBLE		Valid as of software version: ShopTurn 6.3	
Significance:	This MD is used to stipulate the limit value for machining residual material in the direction of axis 1 (Z axis). This value also applies to stock removal, plunge–cutting, and groove turning. Example: If the MD is set to 50% and the final machining allowance is 0.5mm, any residual material thinner than 0.25 mm is not machined in a separate machining step but is removed during finishing.		
Corresponding to	MD 9555: $MM_CTM_CYC_ROUGH_MIN_RESET_MAT2		

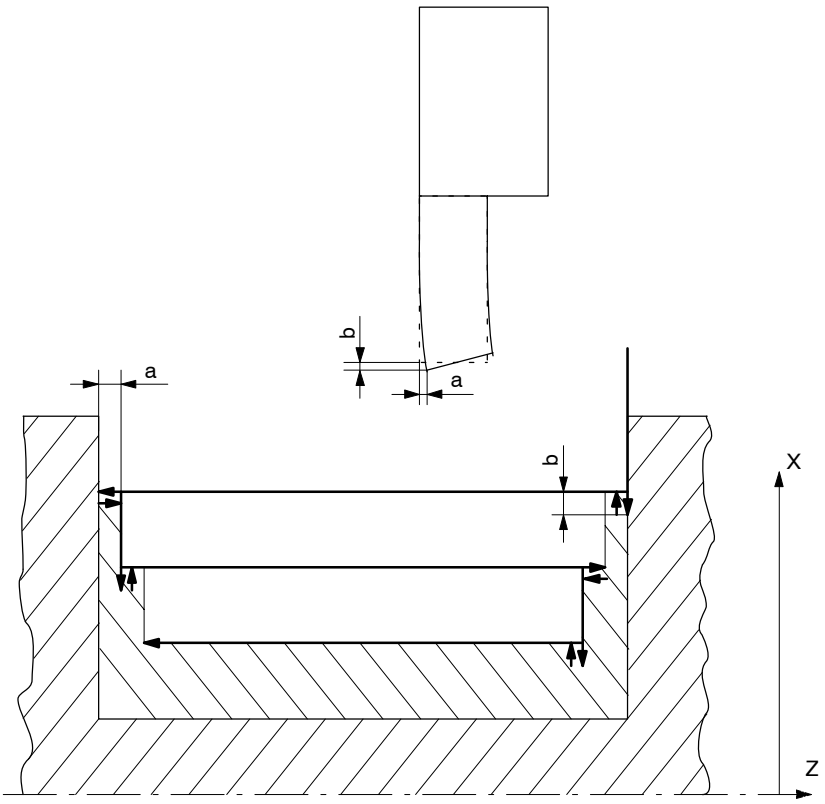
9555 MD number	$MM_CTM_CYC_ROUGH_MIN_REST_MAT2 **Starting thickness, with reference to the final machining allowance, for machining residual material (axis 2)**		
Default setting: 50	Min. input limit: 0	Max. input limit: 1000	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: %	
Data type: DOUBLE		Valid as of software version: ShopTurn 6.3	
Significance:	This MD is used to stipulate the limit value for machining residual material in the direction of axis 2 (X axis). This value also applies to stock removal, plunge–cutting, and groove turning. Example: If the MD is set to 50% and the final machining allowance is 0.5mm, any residual material thinner than 0.25 mm is not machined in a separate machining step but is removed during finishing.		
Corresponding to	MD 9554: $MM_CTM_CYC_ROUGH_MIN_RESET_MAT1		

9556 MD number	$MM_CTM_CYC_ROUGH_VAR_DEPTH **Percentage for variable cutting depth when turning a contour**		
Default setting: 20	Min. input limit: 0	Max. input limit: 50	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: %	
Data type: BYTE		Valid as of software version: ShopTurn 6.3	
Significance:	Specify the percentage for changing the cutting depth when turning a contour. You can select changing the cutting depth for stock removal and for removing residual material.		

9557 MD number	$MM_CTM_CYC_ROUGH_FEED_INT_TIME **Feed interruption time when turning a contour**		
Default setting: –1	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: DOUBLE		Valid as of software version: ShopTurn 6.4	
Significance:	This MD is used to stipulate the feed interruption time for contour turning (removing stock, plunge–cutting, plunge–turning). The MD is only evaluated if MD 9558 MM_CTM_CYC_ROUGH_INT_REL_DIST = 0. > 0: Interruption time in seconds < 0: Interruption time in revolutions 0: No interruption		

9558 MD number	$MM_CTM_CYC_ROUGH_INT_REL_DIST **Retraction distance at feed interruption Contour turning**		
Default setting: 0	Min. input limit: 0	Max. input limit: 10	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version: ShopTurn 6.4	
Significance:	This MD is used to stipulate the retraction distance for feed interruption when contour turning (removing stock, plunge–cutting, plunge–turning). > 0: Retraction distance for feed interruption MD 9557 $MM_CTM_CYC_ROUGH_FEED_INT_TIME is not effective. = 0: No retraction distance		

9560 MD number	$MM_CTM_TURN_GROOV_TOOL_BEND **Retraction due to tool bending when plunge–turning a groove**		
Default setting: 0.1	Min. input limit: 0.0	Max. input limit: 1.0	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version: ShopTurn 6.3	
Significance:	In the stock removal cycle, as the tool bends during plunge turning, the tool cannot travel right up to the contour. This MD is used to stipulate the lateral distance to the last cut by which each subsequent cut is shortened.		



a: Distance to last cut, MD 9560
b: Retraction between plunge–cutting and stock removal, MD 9651.

9561 MD number	**$MM_CTM_TURN_GROOV_FREE_CUT_VAL** **Retraction depth before turning operation when plunge–turning a groove**		
Default setting: 0.1	Min. input limit: 0.0	Max. input limit: 1.0	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version: ShopTurn 6.3	
Significance:	In the stock removal cycle, as the tool bends during plunge turning, the tool would make an excessively deep cut. In this MD you define the tool retraction distance between grooving and stock removal. Diagram see MD 9650 $MM_CTM_TURN_GROOV_TOOL_BEND		

9606 MD number	**$MM_CTM_SIMULATION_TIME_NEW_POS** **Simulation updating rate of actual value**		
Default setting: 350	Min. input limit: 0	Max. input limit: 4000	
Change becomes effective after: POWER ON	Protection level: 3/4	Units: ms	
Data type: WORD		Valid as of software version: ShopTurn 6.1	
Significance:	This MD is used to stipulate the time intervals at which the simulation graphic must be updated to show the current machining process on the machine tool. Value = 0 means no update.		

9611 MD number	**$MM_CTM_CROSS_AX_DIAMETER_ON** **Diameter display for active transverse axes**		
Default setting: 1	Min. input limit: 0	Max. input limit: 1	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version: ShopTurn 6.1	
Significance:	= 0: • Absolute values input as radius value • Zero offsets always specified as radius value = 1: • Position display as diameter • Distance to go as diameter • Absolute paths as diameter		

9619 MD number	**$MM_CTM_G91_DIAMETER_ON** **Incremental infeed**		
Default setting: 0	Min. input limit: 0	Max. input limit: 1	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version: ShopTurn 6.1	
Significance:	If the coordinates are entered as incremental dimensions, you can choose between radius and diameter programming. 0 = Entry as radius 1 = Entry as diameter		

9621 MD number	$MM_CTM_CYCLE_DWELL_TIME **Dwell time for cycles**		
Default setting: –1	Min. input limit: –100	Max. input limit: +100	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4		Units: See meaning
Data type: DOUBLE		Valid as of software version: ShopTurn 6.4	
Significance:	This MD is used to stipulate the dwell time between plunge and retraction for the plunge–cutting cycles (”Turning” → ”Plunge–cutting”). > 0 = Dwell time in seconds < 0 = Dwell time in spindle revolutions		

9626 MD number	$MM_CTM_TRACE **Settings for ShopTurn**		
Default setting: 1	Min. input limit: 0000	Max. input limit: FFFF	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4		Units: Hex
Data type: WORD		Valid as of software version: ShopTurn 7.1	
Significance:	Bit 0: Free Bit 1: Display system messages from ShopTurn in the dialog line (for diagnostics purposes only). Bit 2 to bit 12: Reserved Bit 13: Display cyclical lead time of ShopTurn between 1st and 2nd vertical softkey (for diagnostics purposes only). Bit 14 to 16: Reserved		

9630 MD number	$MM_CTM_FIN_SPEED_PERCENT **Finishing feed in percent**		
Default setting: 100	Min. input limit: 1	Max. input limit: 100	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4		Units: %
Data type: WORD		Valid as of software version: ShopTurn 6.1	
Significance:	This MD is used to stipulate a feed rate for the finishing process when complete machining ”Roughing and finishing” is selected; this feed rate is a percentage of the value entered for parameter F (feed rate).		

9639 MD number	$MM_CTM_MAX_TOOL_WEAR **Upper input limit for tool wear**		
Default setting: 1	Min. input limit: 0	Max. input limit: 10	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4		Units: mm
Data type: DOUBLE		Valid as of software version: ShopTurn 6.4	
Significance:	With this MD you define the upper absolute limit for tool wear (length, radius). This means that the total value must not exceed the upper absolute limit when entering the wear value in the tool wear list. The incremental upper limit is defined in MD 9450 $MM_WRITE_TOA_FINE_LIMIT.		

9640 MD number	$MM_CTM_ENABLE_CALC_THREAD_PITCH **Calculation of thread depth if pitch entered**		
Default setting: 0	Min. input limit: 0	Max. input limit: 1	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version: ShopTurn 6.3	
Significance:	This MD is used to calculate thread depth K for a metric thread according to pitch P (mm/rev) and thread type (internal/external thread). 0 = Thread depth K is not calculated 1 = Thread depth K is calculated		

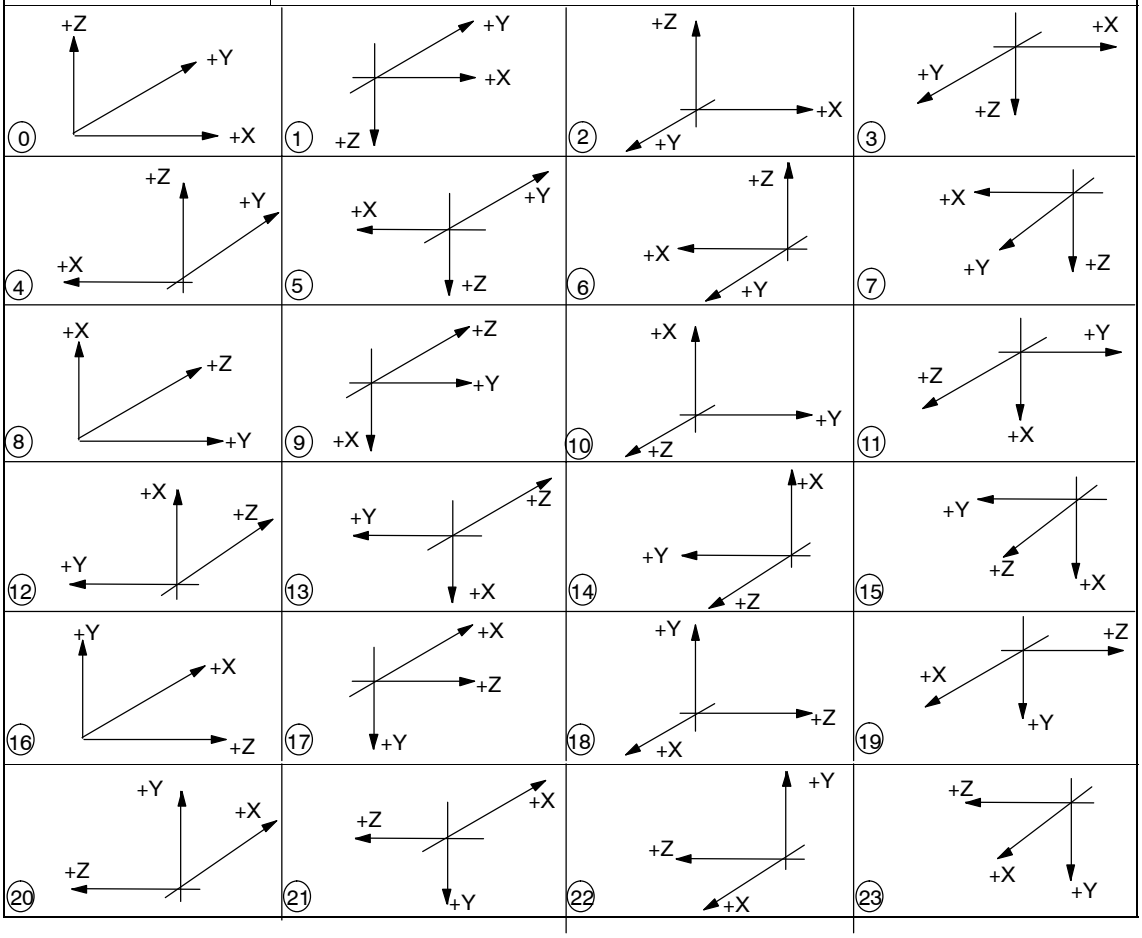
9646 MD number	$MM_CTM_FACTOR_O_CALC_THR_PITCH **Factor for calculating the external thread depth if pitch entered**		
Default setting: 0,6134	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: DOUBLE		Valid as of software version: ShopTurn 6.3	
Significance:	This MD is used to stipulate the factor for converting thread pitch to thread depth for metric external threads.		

9647 MD number	$MM_CTM_FACTOR_I_CALC_THR_PITCH **Factor for calculating the internal thread depth if pitch entered**		
Default setting: 0,5413	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: DOUBLE		Valid as of software version: ShopTurn 6.3	
Significance:	This MD is used to stipulate the factor for converting thread pitch to thread depth for metric internal threads.		

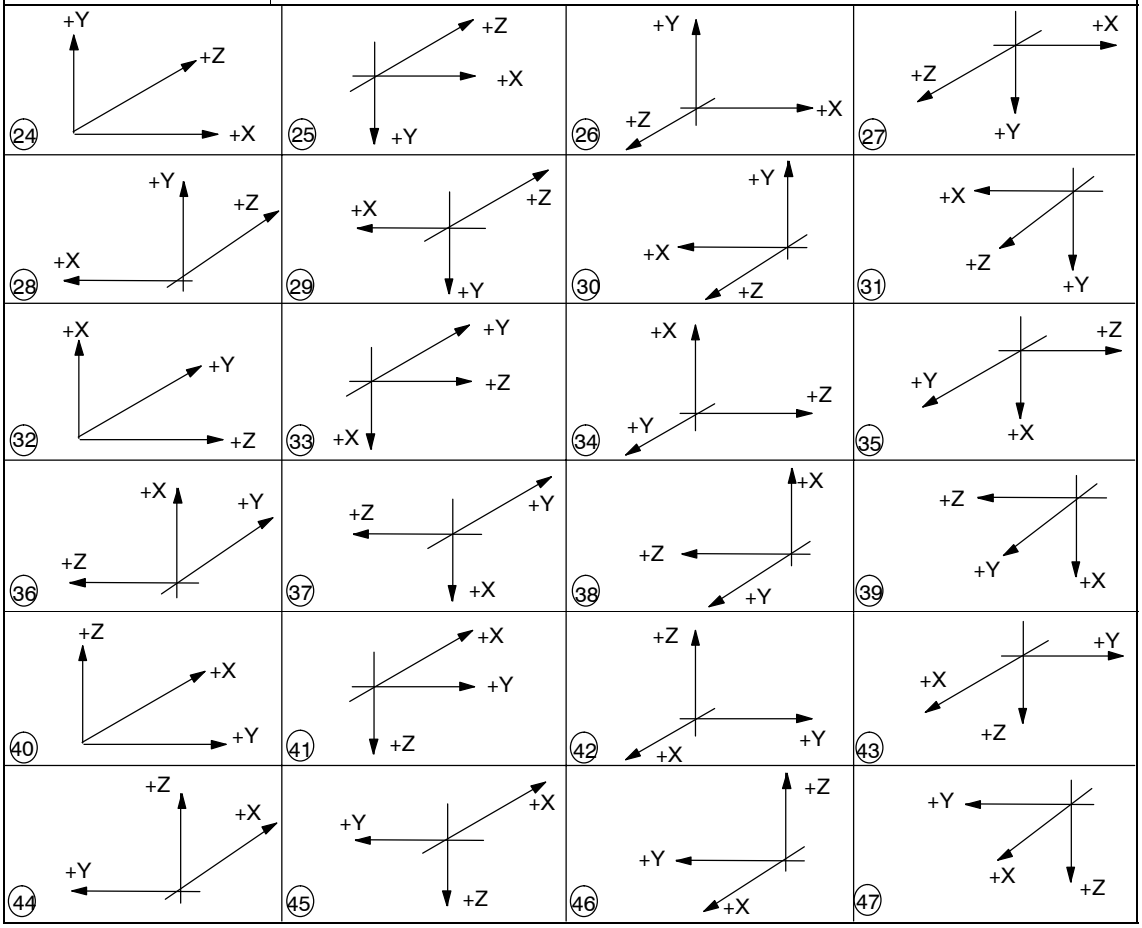
9648 MD number	$MM_CTM_ROUGH_O_RELEASE_DIST **Retraction distance for stock removal/cutting by external machining**		
Default setting: 1,0	Min. input limit: –1	Max. input limit: 100	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version: ShopTurn 6.1	
Significance:	This MD is used to stipulate the distance by which the tool is retracted from the outside contour for cutting and stock removal by external machining. This does not apply to stock removal of a contour. –1 = Retraction by twice the tool nose radius		

9649 MD number	$MM_CTM_ROUGH_I_RELEASE_DIST **Retraction distance for stock removal/cutting by internal machining**		
Default setting: 0,5	Min. input limit: –1	Max. input limit: 100	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version: ShopTurn 6.1	
Significance:	This MD is used to stipulate the distance by which the tool is retracted from the inside contour for cutting and stock removal by internal machining. This does not apply to stock removal of a contour. –1 = Retraction by twice the tool nose radius		

9650 MD number	**$MM_CMM_POS_COORDINATE_SYSTEM** **Position of coordinate system**		
Default setting: 34	Min. input limit: 0	Max. input limit: 47	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to adapt the coordinates system of the operator interface to that of the machine. Depending upon the selected position, all help displays, process graphics, simulation and entry fields with circle direction data automatically change on the ShopTurn user interface. The coordinate system can take on the positions listed below. Also note MD 9719 $MM_CMM_OPTION_MASK, bit 31 (training coordinates system). Examples: **0: Vertical turning machine (vertical boring and turning mill)** **19: Horizontal turning machine, processing in front of turning center** **34: Horizontal turning machine, processing behind turning center (inclined bed turning machine)**		



9650 MD number	$MM_CMM_POS_COORDINATE_SYSTEM **Position of coordinate system**



9651 MD number	$MM_CMM_TOOL_MANAGEMENT **Tool management variant**		
Default setting: 4	Min. input limit: 1	Max. input limit: 4	
Change becomes effective after: POWER ON	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.1	
Significance:	Choice of two tool management variants: 2: Tool management without loading/unloading 4: Tool management with loading/unloading		

9652 MD number	$MM_CMM_TOOL_LIFE_CONTROL **Tool monitoring**		
Default setting: 1	Min. input limit: 0	Max. input limit: 1	
Change becomes effective after: POWER ON	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to activate tool monitoring. 0 = Tool monitoring is not displayed 1 = Tool monitoring is displayed		

9654 MD number	$MM_CMM_SPEED_FIELD_DISPLAY_RES **Number of decimal places in the speed input field**		
Default setting: 0	Min. input limit: 0	Max. input limit: 4	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to stipulate the number of decimal places in parameter field S (speed).		

9657 MD number	$MM_CMM_CYC_MIN_CONT_PO_TO_RAD **Variation of smallest possible cutter radius in %**		
Default setting: 5	Min. input limit: 0	Max. input limit: 50	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: %	
Data type: WORD		Valid as of software version ShopTurn 6.1	
Significance:	This MD is required for milling contour pockets. This parameter is used to stipulate the percentage by which the radius of the cutter being used may be smaller than the one used for generation.		

9658 MD number	$MM_CMM_CYC_MAX_CONT_PO_TO_RAD **Variation of largest possible cutter radius**		
Default setting: 0.01	Min. input limit: 0.0	Max. input limit: 10.0	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is required for milling contour pockets. This parameter is used to stipulate the amount by which the radius of the cutter being used may be larger than the one used for generation.		

9663 MD number	$MM_CMM_TOOL_DISPLAY_IN_DIAM **Display radius/diameter for tool**		
Default setting: 1	Min. input limit: 0	Max. input limit: 1	
Change becomes effective after: POWER ON	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to stipulate the display/input format for the tool (only for milling and drilling tools): 0 = Radius 1 = Diameter		

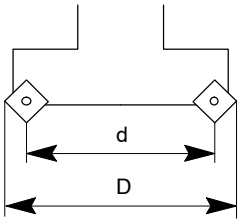
9664 MD number	$MM_CMM_MAX_INP_FEED_P_MIN **Maximum feed in mm/min**		
Default setting: 10000.0	Min. input limit: 0.0	Max. input limit: 100000.0	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm/min	
Data type: DOUBLE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to enter the upper limit for the feed rate input in mm/min.		

9665 MD number	$MM_CMM_MAX_INP_FEED_P_ROT **Maximum feed in mm/rev**		
Default setting: 1.0	Min. input limit: 0.0	Max. input limit: 10.0	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm/rev	
Data type: DOUBLE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to enter the upper limit for the feed rate input in mm/rev.		

9666 MD number	$MM_CMM_MAX_INP_FEED_P_TOOTH **Maximum feed in mm/tooth**		
Default setting: 1.0	Min. input limit: 0.0	Max. input limit: 5.0	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm/tooth	
Data type: DOUBLE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to enter the upper limit for the feed rate input in mm/tooth.		

9667 MD number	$MM_CMM_FOLLOW_ON_TOOL_ACTIVE **Tool preselection active**		
Default setting: 0	Min. input limit: 0	Max. input limit: 1	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to activate tool preselection in a magazine (e.g. chain magazine); i.e. the following tool is already brought to the loading point for a pending tool change. 0 = Tool preselection is not active 1 = Tool preselection is active Note: Tool preselection is not necessary for a turret revolver; i.e. the MD must be set to ”0”.		

9668 MD number	$MM_CMM_M_CODE_COOLANT_I_AND_II **M function coolant I and II**		
Default setting: –1	Min. input limit: –1	Max. input limit: 32767	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: WORD		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to stipulate the M function if in the tools list coolants I and II are both active at the same time. Value: –1 = No M function xy = M function xy for coolant I and II		

9669 MD number	$MM_CMM_FACE_MILL_EFF_TOOL_DIAM **Effective cutter diameter for face milling**		
Default setting: 85.0	Min. input limit: 50.0	Max. input limit: 100.0	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: %	
Data type: DOUBLE		Valid as of software version ShopTurn 6.3	
Significance:	This MD is used to stipulate the effective cutter diameter for face milling. The effective cutter diameter is derived from the ratio d/D, where d = cut diameter and D = largest cutter diameter.  d = 85 D = 100 Effective milling diameter: d/D = 85/100 = 0.85 –> 85%		

9670 MD number	$MM_CMM_START_RAD_CONTOUR_POCKE **Radius of approach circle for finishing contour pockets plus half the final machining allowance (–1 = safety clearance)**		
Default setting: –1.0	Min. input limit: –1	Max. input limit: 100.0	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version ShopTurn 6.1	
Significance:	This MD affects the radius of the approach circle when finishing contour pockets. –1 = The radius is selected such that the safety clearance for final machining is observed at the start point. >0 = The radius is selected such that the value of this MD for final machining allowance is observed at the start point.		

9671 MD number	$MM_CMM_TOOL_LOAD_DEFAULT_MAG **Load tool in default magazine**		
Default setting: 0	Min. input limit: 0	Max. input limit: 30	
Change becomes effective after: POWER ON	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.4	
Significance:	This MD is used to stipulate the magazine in which ShopTurn first searches for an empty location when loading a tool.		

9672 MD number	$MM_CMM_FIXED_TOOL_PLACE **Fixed location coding**		
Default setting: 1	Min. input limit: 0	Max. input limit: 1	
Change becomes effective after: POWER ON	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to stipulate the state of the tools: 0 = Tool with variable location coding in the magazine 1 = Tools with fixed location coding in the magazine Note: With a turret revolver the tools are always allocated to a fixed location; i.e. this MD must be set to ”1”.		

9673 MD number	$MM_CMM_TOOL_LOAD_STATION Number of loading point		
Default setting: 1	Min. input limit: 1	Max. input limit: 2	
Change becomes effective after: POWER ON	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to stipulate the loading point at which the magazine and the spindle are loaded and unloaded. (The magazine and the spindle are always loaded/unloaded at the same loading point.) 1 = Loading point 1 2 = Loading point 2		

9674 MD number	$MM_CMM_ENABLE_TOOL_MAGAZINE Display of magazine list		
Default setting: 1	Min. input limit: 0	Max. input limit: 1	
Change becomes effective after: POWER ON	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.1	
Significance:	0 = Magazine list is not displayed 1 = Magazine list is displayed		

9675 MD number	$MM_CMM_CUSTOMER_START_PICTURE Customized boot screen		
Default setting: 0	Min. input limit: 0	Max. input limit: 1	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.1	
Significance:	The customized boot screen is activated as follows: 0 = Siemens boot screen 1 = Customized boot screen		

9676 MD number	$MM_CMM_DIRECTORY_SOFTKEY_PATH1 Path for drive names in directory management		
Default setting: –	Min. input limit: –	Max. input limit: –	
Change becomes effective after: POWER ON	Protection level: 3/4	Units: –	
Data type: STRING (80 characters)		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to stipulate the path for the drive name of the 2nd softkey (horizontal bar) in directory management with hard disk network link. If a blank string is entered in the display machine data this softkey is not displayed.		

9677 MD number	$MM_CMM_DIRECTORY_SOFTKEY_PATH2 Path for drive names in directory management		
Default setting: –	Min. input limit: –	Max. input limit: –	
Change becomes effective after: POWER ON	Protection level: 3/4	Units: –	
Data type: STRING (80 characters)		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to stipulate the path for the drive name of the 3rd softkey (horizontal bar) in directory management with hard disk network link. If a blank string is entered in the display machine data this softkey is not displayed.		

9678 MD number	$MM_CMM_DIRECTORY_SOFTKEY_PATH3 **Path for drive names in directory management**		
Default setting: –	Min. input limit: –	Max. input limit: –	
Change becomes effective after: POWER ON	Protection level: 3/4	Units: –	
Data type: STRING (80 characters)		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to stipulate the path for the drive name of the 4th softkey (horizontal bar) in directory management with hard disk network link. If a blank string is entered in the display machine data this softkey is not displayed.		

9679 MD number	$MM_CMM_DIRECTORY_SOFTKEY_PATH4 **Path for drive names in directory management**		
Default setting: –	Min. input limit: –	Max. input limit: –	
Change becomes effective after: POWER ON	Protection level: 3/4	Units: –	
Data type: STRING (80 characters)		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to stipulate the path for the drive name of the 5th softkey (horizontal bar) in directory management with hard disk network link. If a blank string is entered in the display machine data this softkey is not displayed.		

9680 MD number	$MM_CMM_M_CODE_COOLANT_I **M function coolant I**		
Default setting: 8	Min. input limit: 0	Max. input limit: 32767	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: WORD		Valid as of software version ShopTurn 6.2	
Significance:	This MD is used to stipulate the M function for coolant I; this is output when the tool is changed.		

9681 MD number	$MM_CMM_M_CODE_COOLANT_II **M function coolant II**		
Default setting: 7	Min. input limit: 0	Max. input limit: 32767	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: WORD		Valid as of software version ShopTurn 6.2	
Significance:	This MD is used to stipulate the M function for coolant II; this is output when the tool is changed.		

9686 MD number	$MM_CMM_M_CODE_COOLANT_OFF **M function for coolant OFF**		
Default setting: 9	Min. input limit: 0	Max. input limit: 32767	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: LONG		Valid as of software version ShopTurn 6.3	
Significance:	This MD is used to stipulate the M function for switching the coolant OFF; this is output when the tool is changed.		

9687 MD number	$MM_CMM_TOOL_MOVE_DEFAULT_MAG **Reload tool in default magazine**		
Default setting: 0	Min. input limit: 0	Max. input limit: 30	
Change becomes effective after: POWER ON	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.4	
Significance:	This MD is used to stipulate the magazine in which ShopTurn first searches for an empty location when reloading a tool.		

9718 MD number	$MM_CMM_OPTION_MASK_2 **Settings for ShopTurn**		
Default setting: 2	Min. input limit: 0000	Max. input limit: FFFF	
Change becomes effective after: IMMEDIATELY	Protection level: 1	Units: Hex	
Data type: LONG		Valid as of software version ShopTurn 7.1	
Significance:	Bit 0: Do not jump to Automatic mode when an external program (from the PLC) is selected for execution. Bit 1: Influence simulation speed by feed override Bit 2: Do not display the basic offset in the measuring and program mask and cannot be entered. Bit 3 to bit 6: Reserved Bit7: Deactivate the function approaching and departing along the software limit switches for the swivel heads. Bits 8 to 12: Reserved Bit 13: The correction angle is inverted during boring (E_DE_BOR).		

9719 MD number	$MM_CMM_OPTION_MASK **Settings for ShopTurn**		
Default setting: H5	Min. input limit: 0000	Max. input limit: FFFF	
Change becomes effective after: IMMEDIATELY	Protection level: 1	Units: Hex	
Data type: LONG		Valid as of software version ShopTurn 7.1	
Significance:	Bit 0 to bit 8: Reserved Bit 9: Start program execution in all screens. Bit 10 to bit 17: Reserved Bit 18: When switching from manual/MDI/Auto, change the operating mode but not the ShopTurn user interface. Bit 19: Instead of ”MCS” and ”WCS” display texts ”Machine” and ”Workpiece”. Bit 20: Display work offsets not as ”ZO1”, but as ”G54” (except in the work offset list). Bit 21: Enable basic record display. Bit 22 to bit 25: Reserved Bit 26: Enable directory ”Part programs” in program manager. Bit 27: Enable directory ”Subroutines” in program manager. Bit 28 to bit 30: Reserved Bit 31: Always display coordinates system for G17 as follows: X to right, Y up (right–handed coordinates system) or down (left–handed coordinates system).		

9724 MD number	$MM_CMM_CIRCLE_RAPID_FEED **Rapid feed for positioning on circular path**		
Default setting: 5000	Min. input limit: 0	Max. input limit: 100000	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm/min	
Data type: DOUBLE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to stipulate the rapid feed rate in mm/min for positioning on a circular path. This applies for the functions ”full circle”/”pitch circle” (”Drilling” menu → ”Positions”) and ”circumferential groove” (”Milling” menu → ”Groove”).		

9725 MD number	$MM_CMM_ENABLE_QUICK_M_CODES **Enable fast M functions**		
Default setting: 0	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.4	
Significance:	This MD is used to enable fast M functions. This means that the M functions are executed by the PLC without confirmation. Bit 0: Coolant 1 ON Bit 1: Coolant 2 ON Bit 2: Coolant 1 and 2 ON Bit 3: Coolant OFF		

9729 MD number	$MM_CMM_G_CODE_TOOL_CHANGE_PROG **Program name for tool change in the G code**		
Default setting: –	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: STRING (24)		Valid as of software version ShopTurn 7.1	
Significance:	This MD is used to stipulate the program name for the tool change program. The tool change program is automatically called as soon as you have programmed a tool in the G code via softkey.		

9777 MD number	$MM_CMM_ENABLE_TIME_DISPLAY **Enable the time display**		
Default setting: 0x7F	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 7.1	
Significance:	This MD is used to define the display of the runtimes. Bit 0: Program progress display Bit 1: Display time Bit 2: Display date Bit 3: Display machine runtime Bit 4: Display processing time Bit 5: Display utilization Bit 6: Display program repeats		

Note

For a description of the display machine data for the measuring cycle, please refer to section 9.1.2 ”Display machine data for measuring cycle”.

9802 MD number	$MM_ST_INDEX_AXIS_C_SUB **Axis index for separate C axis of counterspindle**		
Default setting: 0	Min. input limit: 0	Max. input limit: 127	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.4	
Significance:	This MD is used to enter the axis number of the additional counterspindle.		

9803 MD number	$MM_ST_INDEX_AXIS_4 Axis index for 4th axis		
Default setting: 5	Min. input limit: 0	Max. input limit: 127	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: UBYTE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to enter the axis number of the 4th axis (special axis) you want to display. This can be, for example, the axis you are using to drive the counter–spindle. The axis index of the Y axis is not entered here.		

9804 MD number	$MM_ST_INDEX_SPINDLE_MAIN Axis index for main spindle		
Default setting: 3	Min. input limit: 0	Max. input limit: 127	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: UBYTE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to enter the axis number of the main spindle.		

9805 MD number	$MM_ST_INDEX_SPINDLE_TOOL Axis index for tool spindle		
Default setting: 4	Min. input limit: 0	Max. input limit: 127	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: UBYTE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to enter the axis number of the tool spindle.		

9806 MD number	$MM_ST_INDEX_SPINDLE_SUB Axis index for counter–spindle		
Default setting: 6	Min. input limit: 0	Max. input limit: 127	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: UBYTE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to enter the axis number of the counter–spindle.		

9807 MD number	$MM_ST_INDEX_AXIS_C Axis index for C axis		
Default setting: 3	Min. input limit: 0	Max. input limit: 127	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: UBYTE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to enter the axis number of the C axis.		

9808 MD number	$MM_ST_INDEX_AXIS_B Axis index for B axis		
Default setting: 0	Min. input limit: 0	Max. input limit: 127	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 7.2	
Significance:	This MD is used to enter the axis number of the B axis. This is necessary for aligning tools or machining inclined surfaces.		

9810 MD number	**$MM_ST_GEAR_STEPS_SPINDLE_MAIN** **Number of gears on main spindle**		
Default setting: 0	Min. input limit: 0	Max. input limit: 5	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: UBYTE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to stipulate the number of gear stages for the main spindle.		

9811 MD number	**$MM_ST_GEAR_STEPS_SPINDLE_TOOL** **Number of gears on tool spindle**		
Default setting: 0	Min. input limit: 0	Max. input limit: 5	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: UBYTE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to stipulate the number of gear stages for the tool spindle.		

9812 MD number	**$MM_ST_GEAR_STEPS_SPINDLE_SUB** **Number of gears on counter–spindle**		
Default setting: 0	Min. input limit: 0	Max. input limit: 5	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: UBYTE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to stipulate the number of gear stages for the counter–spindle.		

9820 MD number	**$MM_ST_MAGN_GLASS_POS_1** **Position of magnifying glass for measuring tool, 1st axis**		
Default setting: 0	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to enter the X coordinate of the zoom–in function.		

9821 MD number	**$MM_ST_MAGN_GLASS_POS2** **Position of magnifying glass for measuring tool, 2nd axis**		
Default setting: 0	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to enter the Z coordinate of the zoom–in function.		

9822 MD number	**$MM_ST_DISPL_DIR_MAIN_SPIND_M3** **Displayed direction of rotation for main spindle with M3**		
Default setting: 0	Min. input limit: 0	Max. input limit: 1	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to stipulate which direction of rotation of the main spindle is displayed for M function M3 in the operator interface. The settings for the direction of rotation are as seen looking from the inside out. 0 = Clockwise 1 = Counterclockwise For more information please refer to the section on ”Spindle control”.		

9823 MD number	**$MM_ST_DISPL_DIR_SUB_SPIND_M3** **Displayed direction of rotation for counter–spindle with M3**		
Default setting: 0	Min. input limit: 0	Max. input limit: 1	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to stipulate which direction of rotation of the counter–spindle is displayed for M function M3 in the operator interface. The settings for the direction of rotation are as seen looking from the inside out. 0 = Clockwise 1 = Counterclockwise For more information please refer to the section on ”Spindle control”.		

9824 MD number	**$MM_ST_DISPL_DIR_MAIN_C_AX_INV** **Displayed direction of rotation for C axis main spindle with M3**		
Default setting: 0	Min. input limit: 0	Max. input limit: 1	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to stipulate which direction of rotation of the C axis (main spindle) is displayed for M function M3 in the operator interface. The settings must be made according to the actual direction of rotation of the C axis on the machine. The settings for the direction of rotation are as looking from the inside out. 0 = Clockwise 1 = Counterclockwise For more information please refer to the section on ”Spindle control”.		

9825 MD number	**$MM_ST_DISPL_DIR_SUB_C_AX_INV** **Displayed direction of rotation for C axis counter–spindle with M3**		
Default setting: 0	Min. input limit: 0	Max. input limit: 1	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to stipulate which direction of rotation of the C axis (counter–spindle) is displayed for M function M3 in the operator interface. The settings must be made according to the actual direction of rotation of the C axis on the machine. The settings for the direction of rotation are as looking from the inside out. 0 = Counterclockwise 1 = Clockwise For more information please refer to the section on ”Spindle control”.		

9826 MD number	**$MM_ST_DEFAULT_DIR_TURN_TOOLS** **Main direction of rotation for all turning tools**		
Default setting: 3	Min. input limit: 3	Max. input limit: 4	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to stipulate the main direction of rotation for all turning tools. 3 = M3 4 = M4		

9827 MD number	$MM_ST_DEFAULT_MACHINING_SENSE **Basic settings for machining direction – milling**		
Default setting: 0	Min. input limit: 0	Max. input limit: 1	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to stipulate the basic setting for the direction of machining rotation when milling, except for track milling. 0 = Reverse 1 = Synchronous The basic setting only has an effect in the case of newly created programs.		

9829 MD number	$MM_ST_SPINDLE_CHUCK_TYPES **Type of spindle chuck**		
Default setting: 0	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.4	
Significance:	This MD is used to stipulate which edge is measured for the counter–spindle. This edge then acts automatically as reference point when moving the counter–spindle. Bit 0 = 0: Counter–spindle without jaw dimensions (dimensions of front edge) Bit 0 = 1: Counter–spindle with jaw dimensions (dimensions of stop edge) Or, alternatively, you can make these settings in the ”Tools zero offset” menu → ”>” → ”Spindles” in the ”Jaw type” parameter. Changes to the MD are automatically transferred to the parameter and vice versa.		

9830 MD number	$MM_ST_SPINDLE_PARA_ZL0 **Chuck dimension for main spindle**		
Default setting: 0	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version ShopTurn 6.4	
Significance:	This MD is used to stipulate the chuck dimension ZL0 of the main spindle. The chuck dimensions are required when you use the chuck of the main spindle as reference point for manual tool measurement. Or, alternatively, you can make these settings in the ”Tools zero offset” menu → ”>” → ”Spindles” in parameter ”ZL0”. Changes to the MD are automatically transferred to the parameter and vice versa.		

9831 MD number	$MM_ST_SPINDLE_PARA_ZL1 **Chuck dimension for counter–spindle**		
Default setting: 0	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version ShopTurn 6.4	
Significance:	This MD is used to stipulate the chuck dimension ZL1 of the counter–spindle. The chuck dimensions are required when you use the chuck of the counter–spindle as reference point for manual tool measurement. You also need the chuck dimensions for determining the reference point when moving the counter–spindle (see MD 9829 $MM_ST_SPINDLE_CHUCK_TYPES). Or, alternatively, you can make these settings in the ”Tools zero offset” menu → ”>” → ”Spindles” in parameter ”ZL1”. Changes to the MD are automatically transferred to the parameter and vice versa.		

9832 MD number	$MM_ST_SPINDLE_PARA_ZL2 **Stop dimension for counter–spindle**		
Default setting: 0	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version ShopTurn 6.4	
Significance:	This MD is used to stipulate the stop dimension ZL2 of the counter–spindle. You need the stop dimensions for determining the reference point when moving the counter–spindle (see MD 9829 $MM_ST_SPINDLE_CHUCK_TYPES). Or, alternatively, you can make these settings in the ”Tools zero offset” menu → ”>” → ”Spindles” in parameter ”ZL2”. Changes to the MD are automatically transferred to the parameter and vice versa.		

9833 MD number	$MM_ST_SPINDLE_PARA_ZL3 **Jaw dimension for counter–spindle**		
Default setting: 0	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version ShopTurn 6.4	
Significance:	This MD is used to stipulate the jaw dimension ZL3 of the counter–spindle. You need the jaw dimensions for determining the reference point when moving the counter–spindle (see MD 9829 $MM_ST_SPINDLE_CHUCK_TYPES). Or, alternatively, you can make these settings in the ”Tools zero offset” menu → ”>” → ”Spindles” in parameter ”ZL3”. Changes to the MD are automatically transferred to the parameter and vice versa.		

9836 MD number	$MM_ST_TAILSTOCK_DIAM **Tailstock diameter**		
Default setting: 0	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version ShopTurn 7.1	
Significance:	This MD is used to stipulate the diameter of the tail stock. The tail stock diameter is needed for the purposes of displaying the tail stock in simulation.		

9837 MD number	$MM_ST_TAILSTOCK_LENGTH **Tailstock length**		
Default setting: 0	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version ShopTurn 7.1	
Significance:	This MD is used to stipulate the length of the tail stock. The tail stock length is needed for the purposes of displaying the tail stock in simulation.		

9838 MD number	$MM_ST_BORDER_TOOL_LEN_X_REV_2 **Limit value of tool length X for the 2nd tool carrier**		
Default setting: 0	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version ShopTurn 7.1	
Significance:	This MD is used to specify the limit value of tool length X for the 2nd tool carrier. MD 9838 = 0; Only one tool carrier is available X < MD 9838; Tool belongs to tool carrier 1 X ≥ MD 9838; Tool belongs to tool carrier 2		

9840 MD number	$MM_ST_ENABLE_MAGN_GLASS **Magnifying glass function in manual: Measuring a tool**		
Default setting: 0	Min. input limit: 0	Max. input limit: 1	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.1	
Significance:	This MD can be used to activate the ”Measure tool with zoom–in” function. 0 = Function not available 1 = Function available		

9841 MD number	$MM_ST_ENABLE_PART_OFF_RECEPT **Enable receptacle function for cut–off**		
Default setting: 0	Min. input limit: 0	Max. input limit: 1	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.1	
Significance:	This MD can be used, if you have implemented the ”Receptacle for cut–off” function via machine manufacturer cycle ST_CUST.SPF, to activate the ”Part gripper” parameter in the ”Part cut–off” mask. 0 = ”Part gripper” parameter is not displayed. 1 = ”Part gripper” parameter is displayed.		

9842 MD number	$MM_ST_ENABLE_TAILSTOCK **Enable tail stock**		
Default setting: 0	Min. input limit: 0	Max. input limit: 1	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: BYTE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to activate the ”tail stock” parameter in the program header mask. 0 = ”Tail stock” parameter is not displayed 1 = ”Tail stock” parameter is displayed		

9843 MD number	$MM_ST_ENABLE_SPINDLE_CLAMPING **Enable spindle clamping (C axis)**		
Default setting: 0	Min. input limit: 0	Max. input limit: 1	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: WORD		Valid as of software version ShopTurn 6.4	
Significance:	This MD can be used, if you have implemented the ”Clamp/unclamp spindle” function via machine manufacturer cycle ST_CUST.SPF, to activate the ”Clamp/unclamp spindle” parameter in the drilling and milling masks. 0 = ”Clamp/unclamp spindle” parameter is not displayed in the drilling and milling masks. ShopTurn automatically clamps the spindle when machining if clamping is useful for the current application. 1 = ”Clamp/unclamp spindle” parameter is displayed in the drilling and milling masks. The operator decides for which machining the spindle should be clamped.		

9849 MD number	$MM_ST_CYCLE_SUB_SP_PARK_POS_Y **Parking position of Y axis with counterspindle**		
Default setting: 0	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version ShopTurn 7.2	
Significance:	This MD is used to specify the Y position in the MCS when approaching the parking position for the counterspindle cycle.		

9850 MD number	**$MM_ST_CYCLE_THREAD_RETURN_DIST** **Return distance for thread turning**		
Default setting: 2	Min. input limit: 0	Max. input limit: 1000	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to stipulate the distance from the workpiece by which the tool is retracted between infeeds for thread cutting.		

9851 MD number	**$MM_ST_CYCLE_SUB_SP_WORK_POS** **Retraction position Z for counter–spindle**		
Default setting: 0	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to stipulate the position in the Z direction to which the counter–spindle travels at program start. –1: Suppress retraction of counter–spindle		

9852 MD number	**$MM_ST_CYCLE_SUB_SP_DIST** **Distance as of which feed rate is used when approaching fixed stop with counter–spindle**		
Default setting: 10	Min. input limit: 0.001	Max. input limit: 1000	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to stipulate the distance from the programmed target position as of which the counter–spindle travels with a special feed rate when approaching the fixed stop. You define the feed rate in MD 9853 $MM_ST_CYCLE_SUB_SP_FEED.		

9853 MD number	**$MM_ST_CYCLE_SUB_SP_FEED** **Feed rate for traveling to fixed stop with counter–spindle**		
Default setting: 0	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm/min	
Data type: DOUBLE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to stipulate the feed rate with which the counter–spindle travels to the fixed stop. You define the distance as of which the axis travels at this feed rate in MD 9852 $MM_ST_CYCLE_SUB_SP_DIST		

9854 MD number	**$MM_ST_CYCLE_SUB_SP_FORCE** **Force in percent for traveling to fixed stop, counter–spindle**		
Default setting: 10	Min. input limit: 1	Max. input limit: 100	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: %	
Data type: DOUBLE		Valid as of software version ShopTurn 6.1	
Significance:	This MD is used to stipulate the percentage of the driving force with which the counter–spindle should stop when traveling to the fixed stop.		

9855 MD number	$MM_ST_CYCLE_TAP_SETTINGS Settings for rigid tapping		
Default setting: 0	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: WORD		Valid as of software version ShopTurn 6.3	
Significance:	Units digit: Exact response 0: Exact stop response active as before cycle call 1: Exact stop G601 2: Exact stop G602 3: Exact stop G603 Tens digit: Feed–forward control 0: With/without feed–forward control active as before cycle call 1: With feed–forward control FFWON 2: Without feed–forward control FFWOF Hundreds digit: Acceleration 0: SOFT/BRISK/DRIVE active as before cycle call 1: With jerk limiting SOFT 2: Without jerk limiting BRISK 3: Reduced acceleration DRIVE Thousands digit: MCALL 0: For MCALL Reactivate spindle operation 1: For MCALL remain in position control		

9856 MD number	$MM_ST_CYCLE_TAP_MID_SETTINGS Settings for center tapping		
Default setting: 0	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: WORD		Valid as of software version ShopTurn 6.3	
Significance:	Units digit: Exact response 0: Exact stop response active as before cycle call 1: Exact stop G601 2: Exact stop G602 3: Exact stop G603 Tens digit: Feed–forward control 0: With/without feed–forward control active as before cycle call 1: With feed–forward control FFWON 2: Without feed–forward control FFWOF Hundreds digit: Acceleration 0: SOFT/BRISK/DRIVE active as before cycle call 1: With jerk limiting SOFT 2: Without jerk limiting BRISK 3: Reduced acceleration DRIVE Thousands digit: MCALL 0: For MCALL Reactivate spindle operation 1: For MCALL remain in position control		

9857 MD number	$MM_ST_CYCLE_RET_DIST_FIXEDSTOP Retraction distance before clamping after traveling to fixed stop		
Default setting: 0	Min. input limit: 0	Max. input limit: 10	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version ShopTurn 6.3	
Significance:	This MD is used to stipulate the retraction distance that the counter–spindle is to travel between traveling to fixed stop and gripping in order to eliminate compressive stress in the workpiece.		

9858 MD number	$MM_ST_CYCLE_RET_DIST_PART_OFF Retraction distance before cut–off with counter–spindle		
Default setting: 0	Min. input limit: 0	Max. input limit: 1	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version ShopTurn 6.3	
Significance:	This MD is used to stipulate the retraction distance that the counter–spindle is to travel before cut–off in order to exert tensile stress on the workpiece. This relieves pressure on the tool when cutting off.		

9859 MD number	$MM_ST_CYCLE_PART_OFF_CTRL_DIST Distance for cut–off check		
Default setting: 0.1	Min. input limit: 0	Max. input limit: 10	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version ShopTurn 6.3	
Significance:	This MD is used to stipulate the distance that the counter–spindle is to travel after cut–off in order to perform a cut–off check. The cut–off check employs the ”travel to fixed stop” function. The cut–off is successful if travel to fixed stop fails. 0 = Do not carry out cut–off check		
Corresponding to	MD 9860 $MM_ST_CYCLE_PART_OFF_CTRL_FEED		

9860 MD number	$MM_ST_CYCLE_PART_OFF_CTRL_FEED Feed for cut–off check		
Default setting: 0	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm/min	
Data type: DOUBLE		Valid as of software version ShopTurn 6.3	
Significance:	This MD is used to stipulate the feed with which the counter–spindle is to travel after cut–off in order to perform a cut–off check. The cut–off check employs the ”travel to fixed stop” function. The cut–off is successful if travel to fixed stop fails. 0 = Do not carry out cut–off check		
Corresponding to	MD 9859 $MM_ST_CYCLE_PART_OFF_CTRL_DIST		

9861 MD number	$MM_ST_CYCLE_PART_OFF_CTRL_FORC Force in percent for cut–off check		
Default setting: 10	Min. input limit: 1	Max. input limit: 100	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: %	
Data type: BYTE		Valid as of software version ShopTurn 6.3	
Significance:	This MD is used to stipulate the percentage of the driving force with which the cut–off check is to be performed after cut–off. The cut–off check employs the ”travel to fixed stop” function. The cut–off is successful if travel to fixed stop fails.		

9862 MD number	$MM_ST_CYC_DRILL_MID_MAX_ECCENT Maximum center offset for center drilling		
Default setting: 0.5	Min. input limit: 0.0	Max. input limit: 10.0	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: mm	
Data type: DOUBLE		Valid as of software version ShopTurn 6.4	
Significance:	This MD is used to stipulate the maximum center offset for ”center drilling”.		

9863 MD number	$MM_ST_MAX_INP_AREA_GAMMA **Maximum input area for gamma alignment angle**		
Default setting: 5	Min. input limit: 0	Max. input limit: 90	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: DOUBLE		Valid as of software version ShopTurn 7.2	
Significance:	This MD is used to specify the maximum input area for the gamma alignment angle. The value acts as a +/– area for the angle values 0° and 180°.		

9897 MD number	$MM_ST_OPTION_MASK_MAN_FUNC **Settings for ShopTurn manual functions**		
Default setting: 8	Min. input limit: –	Max. input limit: –	
Change becomes effective after: IMMEDIATELY	Protection level: 3/4	Units: –	
Data type: LONG		Valid as of software version ShopTurn 7.1	
Significance:	Bit 0 = 0: Control of main spindle via machine control panel Bit 0 = 1: Control of main spindle via operator interface Bit 1 = 0: Control of tool spindle via machine control panel Bit 1 = 1: Control of tool spindle via operator interface Bit 2: Reserved Bit 3 = 0: Manual taper turning hidden Bit 3 = 1: Manual taper turning displayed		

9898 MD number	$MM_ST_OPTION_MASK **Settings for ShopTurn**		
Default setting: H7000	Min. input limit: 0000	Max. input limit: FFFF_FFFF	
Change becomes effective after: IMMEDIATELY	Protection level: 1	Units: Hex	
Data type: LONG		Valid as of software version ShopTurn 7.5	
Significance:	Bit 0 = 1: Enable machining "internal/rear" in screens that define the machining level themselves. Bit 1 to bit 5: Reserved Bit 6 = 1: Allow simultaneous recording despite program start. Bit 7 and bit 8: Reserved Bit 9 = 1: Disable entering an offset in X in the zero offset list. Bit 10 = 1: Display program view under simultaneous recording. Bit 11 = 0: After reset, retain currently active zero offset. Bit 11 = 1 and MD 20152[7] = 0: After reset, the zero offset which is entered in menu "T, S, M" remains effective. Bit 12 = 1: Enable "Plunge–turning" and "Plunge–turning residual material" functions. Bit 13 = 1: Enable "Plunge–cutting" and "Plunge–cutting residual material" functions. Bit 14 = 1: Allow negative final machining allowance for contour turning. Bit 15 = 1: Counterspindle: Allow internal/external clamping. Bit 16 = 1: Measure tools with probe: Additional probe on counterspindle. Bit 17 = 1: Tool measurement for rotating tools with rotating spindle Bit 18 = 1: Display column "Rotation" in the zero offset list. Bit 19 to bit 24: Reserved Bit 25: A full cross–section of the side view is always displayed in the simulation. Bit 26: Beta and gamma angles are not positioned during tool measurements. Bit 27: Reserved		

9899 MD number	**$MM_ST_TRACE** **Settings for ShopTurn**		
Default setting: 0	Min. input limit: 0000	Max. input limit: FFFF	
Change becomes effective after: IMMEDIATELY	Protection level: 1		Units: Hex
Data type: LONG		Valid as of software version ShopTurn 7.1	
Significance:	Bit 0: Reserved		

■

Space for your notes

Tool Management

8

8.1 Overview of functions

Option

ShopTurn only runs when the tool management option is set. Tool management is shipped together with ShopTurn as standard. The option is already set in the standard set of machine data for ShopTurn.

References: /FBW/, Description of Tool Management Functions

Data

Data storage and management is carried out in the NCK. All data can be read and written by manual entry, via the initialization program, or by data transfer.

Operation

Operation is performed via system screens.

Programming

When using tool management a tool can be called using another name, e.g. ”Roughing tool_80”.
The tool can still also be called using a T no. (tool number). The T no. is then the name of the tool.

PLC

Separate PLC blocks are available for tool management to handle communication between NCK and PLC.

Tools

With ShopTurn on NCU (HMI Embedded sl), a maximum of 250 tools can be set up; on the PCU 50.3 the maximum number of tools is limited by MD 18082 $MN_MM_NUM_TOOL.
Up to 9 cutting edges can be defined per tool.

Replacement tools

Maximum 98 replacement tools can be set up for each tool.

Magazine

It is possible to manage turret, chain or disk–type magazines.
If you want to use chain or disk–type magazines, you need to set bit 13 in display machine data 9478 $MM_TO_OPTION_MASK.
The maximum number of magazines is set in the NC.
The magazine list can be hidden via display MD 9674 $MM_CMM_ ENABLE_TOOL_MAGAZINE.

Magazine location blocking

Magazine locations can be blocked, e.g. for oversized tools occupying adjacent magazine locations.

Monitoring functions

Tool monitoring in the tool management system can be performed according to tool life, the number of completed workpieces, or tool wear. Replacement tools (sister tools) are distinguished by their Duplo number (DP).
The tool monitoring can be deactivated via the display MD 9652 $MM_CMM_TOOL_LIFE_CONTROL.
The wear monitoring is activated via the MD 18080, bit 5.

Tool management without loading/ unloading

Tool management without the load/unload softkey can be set by means of value 2 in display MD 9651 $MM_CMM_TOOL_MANAGEMENT.

Tool management with loading/ unloading

Tool management with the load/unload softkey can be set by means of value 4 in display MD 9651 $MM_CMM_TOOL_MANAGEMENT.

Load

Loading brings the tool to its magazine location.

Unload

Unloading removes the tool from the magazine.

Sorting

Tools can be sorted in the tool list and the tool wear list according to magazine location, name, type, and tool number.

Other functions

- Loading/unloading point for tools by means of display MD 9673 $MM_CMM_TOOL_LOAD_STATION
- Display tools (mill/drill) in diameter or radius via display MD 9663 $MM_CMM_TOOL_DISPLAY_IN_DIAM.

Modifying data

Note

Changes to the tool and cutting edge data made via system variables in the part program are only displayed in the tool list in the ShopTurn user interface if they refer to the active tool.

8.2 Commissioning sequence

You can start up tool management either together with ShopTurn or afterwards 4.2 (see section ”First commissioning sequence”).

To commission tool management, proceed as follows:

1. NCK commissioning
2. PLC commissioning
3. Adapt display machine data

If tool management has already been installed on your machine, you do not need to install tool management specifically for ShopTurn; i.e. it is not necessary to start up the NCK/PLC; you only need to adapt the display machine data for tool management.

For a precise description of tool management commissioning, please refer to:
References: /FBIW/, Description of Functions Tool Management

Note

With PCU 50.3 you can also commission tool management (generating configuration file and PLC data) via the CNC–ISO operator interface.

References: /FBW/, Description of Tool Management Functions

8.3 Commissioning in NCK

Requirements

- PCU commissioning is performed and the connection to the NCK established.
- NCK commissioning is carried out with the default machine data.

Implementation

- Enter the NCK machine data for tool management.
- Create a new configuration file for tool management or use and adapt the examples from the toolbox.
- Load the configuration file into the controller.

8.3.1 Enter the NCK machine data

In order to use tool management the machine data for memory setting and for activating tool management must be set.

Machine data for memory setting

Memory space must be made available for tool management in the battery–backed RAM.

The following machine data must be set:

MD 18080	Activate the memory for the tool management
MD 18082	Number of tools to be managed by the NCK
MD 18084	Number of turrets that the NCK can manage (min. 3); include intermediate memory and loading magazine in the count!
MD 18086	Number of turret locations that the NCK can manage; include 1 intermediate memory location (tool holder) and 2 load locations in the count!
MD 18100	Number of cutting edges in the NCK

Note

ShopTurn can manage 9 cutting edges per tool.
Define the total number of all available cutting edges in MD 18105 MM_MAX_CUTTING_EDGE_NO.
It is not necessary to enter the number of tools multiplied by 9 cutting edges.
You can simply specify approximately how many cutting edges you will need in total.

Example

Assignment of machine data:

18082=40;	40 tools (12 turret locations + 28 additional locations)
18084=3;	1 turret + 1 buffer magazine + 1 load magazine
18086=15;	12 turret locations + 1 buffer location + 2 load locations
18100=80;	80 cutting edges

Note

Via the machine data you simply reserve memory; the locations are only actually assigned to the turret, etc. when the configuration file is created and loaded; (see section 8.3.3 ”Creating and loading the configuration file”).

Modifying the ”memory–influencing” machine data also reformats the battery–backed RAM. Data must therefore be backed up in advance.

Machine data for activating tool management

In addition, the following machine data must be set for activating tool management:

MD 20310	Channel–specific activation of tool management
MD 20320	Activation of tool life monitoring for the spindle specified here
MD 20124	Activate tool holder

Note

Bits 0–3 of MD 20310 $MC_TOOL_MANAGEMENT_MASK and MD 18080 $MN_MM_TOOL_MANAGEMENT_MASK must always be set identically.

Default

For a detailed description of the NCK machine data for tool management please refer to the next section 8.3.2 ”Description of the NC Machine Data for Tool Management”.

8.3.2 Description of NCK machine data

18080	**MM_TOOL_MANAGEMENT_MASK**		
MD number	Activate the memory for tool management		
Default setting: HB	Min. input limit: 0	Max. input limit: 0xFFFF	
Change becomes effective after: POWER ON	Protection level: 1/4	Units: HEX	
Data type: DWORD	Valid as of software version: 4.3		
Significance:	Activation of the tool management memory with ”0” means: The tool management data set does not occupy any memory; tool management is not available. Bit 0=1: Memory for data specific to tool management is available; the MDs for reserving memory must be set accordingly (18086 MM_NUM_MAGAZINE_LOCATION, 18084 MM_NUM_MAGAZINE) Bit 1=1: Memory is available for monitoring data Bit 2=1: Memory is available for user data (CC data) Bit 3=1: Memory is available for considering the adjacent location Bit 4=1: Memory and function release for PI service _N_TSEARC = ”Complex search for tools in magazine” is available. Bit 5=1: Wear monitoring active (SW5 and later, 840D) Bit 6=1: Wear grouping available (SW5 and later, 840D) Bit 7=1: Reserve memory for magazine location adapters Bit 8=1: Memory for operation and/or setup corrections Bit 9=1: Tools in a turret no longer vacate their turret location on tool change (in the display) The coded type of memory reservation enables economic use of memory management for the functionality provided. Example: Standard memory reservation for tool management: MD = 3 (bit 0 + 1=1) means tool management and tool monitoring data are available MD = 1 means tool management without tool monitoring data		

18082	**MM_NUM_TOOL**		
MD number	Number of tools the NCK can manage		
Default setting: 24	Min. input limit: 0	Max. input limit: 600	
Change becomes effective after: POWER ON	Protection level: 2/4	Units: —	
Data type: DWORD	Valid as of software version: 2.		
Significance:	The number of tools which the NCK can manage is entered here. The maximum number of tools equals the number of cutting edges in the NCK. Battery–backed memory is reserved for the number of tools.		
Additional references	Description of functions: Memory Configuration (S7), Tool Offset (W1)		

18084	MM_NUM_TOOL_MAGAZINE		
MD number	Number of magazines the NCK can manage		
Default setting: 3	Min. input limit: 0	Max. input limit: 32	
Change becomes effective after: POWER ON	Protection level: 2/4	Units: —	
Data type: DWORD	Valid as of software version: 2.		
Significance:	Number of magazines that the NCK can manage (active and background magazines). This MD is used to reserve battery–backed memory for the magazines. Important: One load magazine and a buffer magazine is set up in the tool management for each TOA unit. These magazines must be taken into account. Value = 0: Tool management cannot be active because no data can be created.		
Additional references	Description of functions: Memory configuration (S7)		

18086	MM_NUM_MAGAZINE_LOC		
MD number	Number of magazine locations the NCK can manage		
Default setting: 15	Min. input limit: 0	Max. input limit: 600	
Change becomes effective after: POWER ON	Protection level: 2/4	Units: —	
Data type: DWORD	Valid as of software version: 2.		
Significance:	Number of magazines the NCK can manage. This MD reserves the battery–backed memory for the magazine locations. Important: The locations in the intermediate memory and a loading magazine must be taken into account. Value = 0: Tool management cannot be active because no data can be created.		
Additional references	Description of functions: Memory configuration (S7)		

18100	MM_NUM_CUTTING_EDGES_IN_TOA		
MD number	Number of tool cutting edges per TOA block		
Default setting: 48	Min. input limit: 0	Max. input limit: 1500	
Change becomes effective after: POWER ON	Protection level: 2/4	Units: —	
Data type: DWORD	Valid as of software version: 2.		
Significance:	This MD is used to stipulate the number of cutting edges in the NCK. For each cutting edge – irrespective of tool type – this machine data reserves approximately 250 bytes per TOA block in the battery–backed memory. Tools with cutting edges of type 400–499 (= grinding tools) occupy space for a cutting edge in addition. Example: Define 10 grinding tools each with one cutting edge. The following minimum requirements apply: MM_NUM_TOOL = 10 MM_NUM_CUTTING_EDGES_IN_TOA = 20 See also MM_NUM_TOOL Special cases: If this machine data is altered the buffered data is lost. References: /FBW/, "Description of Tool Management Functions"		
Additional references	Description of functions: Memory configuration (S7)		

20124 MD number	**TOOL_MANAGEMENT_TOOLHOLDER** Toolholder number		
Default setting: 0,0,0,...	Min. input limit: 0	Max. input limit: 16	
Change becomes effective after: POWER ON	Protection level: 2/7	Units: –	
Data type: DWORD	Valid as of software version: 3.2.		
Significance:	This MD only takes effect if tool management is active. It is used to stipulate whether a tool holder number or spindle number have to be indicated in order to define the location of the tool to be loaded. Tool management must know on which tool holder the tool is to be loaded. If the MD is larger than 0, the spindle numbers $TC_MPP5 are interpreted as tool holder numbers. The automatic address extension of T and of M06 is then the value of this MD and no longer the value of MD 20090 SPIND_DEF_MASTER_SPIND. For machines with several tool holders without a designated master spindle the MD acts as a default value defining the tool holder to which the tool is to be loaded on tool change. SETMTH(n) designates tool holder n as the master tool holder. Tools which are to be loaded in a buffer location of the spindle type and which have the value $TC_MPP5=n have the effect of correcting the tool path. Tools with a value not equal to n have no effect on the offset. Command SETMTH restores the tool holder stipulated in the MD as master tool holder. When defining the magazine locations of internal magazines, spindle locations –$TC_MPP1=2=spindle location– can be assigned a location type index ($TC_MPP5). This assignes a concrete tool holder to the location.		
Corresponding to...	MD 20090 SPIND_DEF_MASTER_SPIND MD 20110: RESET_MODE_MASK MD 20112: START_MODE_MASK MD 20122: TOOL_RESET_NAME MD 20130: CUTTING_EDGE_RESET_VALUE		
Additional references			

20310 MD number	**TOOL_MANAGEMENT_MASK** Channel–specific activation of tool management		
Default setting: 1400B	Min. input limit: 0	Max. input limit: 0xFFFFF	
Changes effective after POWER ON	Protection level: 2/4	Units: HEX	
Data type: DWORD	Valid as of software version: 2		

Significance:

MD = 0: Tool management inactive
Bit 0=1: Tool management active
The tool management functions are enabled for the current channel.
Bit 1=1: Tool monitoring function active
Functions that are used for tool monitoring (tool life and workpiece count) are enabled.
Bit 2=1: OEM functions active
The memory can be used for user data (see also MD 18090 to 18098).
Bit 3=1: Considering the adjacent location active
Bit 0 to Bit 3 must be set as for MD 18080: MM_TOOL_MANAGEMENT_MASK.
Bit 4=1: The PLC is able to request another tool change preparation process with modified parameters.

With T selection or M06 the part program pauses until acknowledged by the PLC program

Bit 5=1: When the tool command is output the main run of the main spindle can be stopped within one OB1 cycle (e.g. by read–in halt).
Bit 5=0: When the command is output to the PLC the main run of the main spindle is resumed.
Bit 6=1: When the tool command is output the main run of the auxiliary spindle can be stopped within one OB1 cycle (e.g. by read–in halt).
Bit 6=0: When the command is output to the PLC the main run of the main spindle is resumed.
Bit 7=1: The main run of the main spindle is halted until acknowledgment with status 1 is received via FC7, FC8.
Bit 7=0: When the command is output to the PLC the main run of the main spindle is resumed.
Bit 8=1: The main run of the auxiliary spindle is halted until acknowledgment with status 1 is received via FC7, FC8.
Bit 8=0: When the command is output to the PLC the main run of the auxiliary spindle is resumed.
Bit 9: Reserved 10=1: M06 is delayed until ”prepare change” is received via FC8 (status 1) from the PLC.
The change signal (e.g. M06) is only output when tool selection (DBX [n+0].2) is acknowledged. The part program is halted with M06 until tool selection is acknowledged.
Bit 10=0: The tool change ON command NCK –> PLC is not output until the PLC preparation acknowledgment has been received. This is relevant for PLC command 3 (i.e. programming of M06 in a block containing no T).
Bit 11=1: The preparation command is output even if it has already been issued once for the same tool.
This system is used to position the chain with the first '' Tx ''
and to check with the second call whether the tool is in the correct tool–change location (e.g. in front of the changing station)
Bit 11=0: The preparation command can be output only once for a tool.
Bit 12=1: The preparation command is executed even if the tool is already in the spindle.
This means that the T selection signal (DB72.DBXn.2) is set even if it has already been set once for the same tool. (Tx...Tx)
Bit 12=0: The preparation command is not executed if the tool is already inserted in the spindle.

20310 MD number	**TOOL_MANAGEMENT_MASK** Channel–specific activation of tool management
Significance:	Bit 13=1: Only for systems with sufficient memory capacity. Recording of tool sequences in a diagnostics buffer. On reset the commands are transferred from the diag nostics buffer to the passive file system (NCATR xx.MPF under part program). This file is required by the Hotline. Tool sequences are recorded in the diagnostics buffer only on systems with sufficient memory (NCU572, NCU573). Bit 14=1: An automatic tool change takes place on reset and start as per MD20120 TOOL_RESET_NAME MD20110 RESET_MODE_MASK MD20124 TOOL_MANAGEMENT_TOOLHOLDER. If the tool specified in TOOL_RESET_NAME is to be loaded (this is set in RESET_MODE_MASK), a tool selection and change command is output to the user interface on RESET or START (DB72). If RESET_MODE_MASK is set, thus retaining the active tool, and the active tool is disabled in the spindle (by the user), a tool change command for a replacement tool is output to the user interface. If no replacement tool is available, an error message is output. Bit 14=0: No automatic tool change takes place on RESET and START. Bit 15=1: The tool is not returned in response to several preparation commands (Tx–>Tx). This method of function activation permits various combinations. Example for default activation of tool management: MD 20310 TOOL_MANAGEMENT_MASK = 3 (Bit0 + 1 = 1) Bit16=1: T location number is active Bit 15=0: The tool is not returned. Bit 16=1: T=location number is active. Bit 17=1: Tool life decrementation can be started/stopped via the PLC in channel DB 2.1...DBx 1.3. Bit 18=1: Activation of monitoring ”last tool in tool group”. Bit 18=0: No monitoring for ”last tool in tool group” Bit 19=1: Activation for bits 5 to 8 Bit 19=0: Functions described under bits 5 to 8 are not available. Bit 20=0: On PLC signal ”program testing active” the commands generated on are not output to the PLC. The NCK acknowledges the commands itself. Magazine and tool data are not changed. Exception: The status of the tool activated in the test area can be changed to ”active”. Bit 20=1: On PLC signal ”program testing active” the commands generated are output to the PLC. In so doing. depending on the type of PLC acknowledgment, tool/magazine data in the NCK may be altered. If the acknowledgment parameters for the target magazine are set with the same values as the source magazine, the tool is not transported and thus no data is modified in the NCK. Exception: The status of the tool activated in the test area can be changed to ”active”. Bit 21=0: Ignore tool status ”W” on tool selection. Bit 21=1: Tools with status ”W” cannot be selected by another tool change or tool preparation command. Bit 22=0 Default setting Bit 22=1 If the function T=”Location” (see Bit 16) is active, the tool groups are automatically divided into subgroups. $TC_TP11 is the grouping and selection parameter. On transition to replacement tool only those tools in the tool group are recognized as replacement tools that in the $TC_TP11–value have at least one bit of the tool set to the programmed location. Bit 23=0 Default setting Tool management selects the tool in the main run with optimum certainty; i.e. the interpreter must, if correction is required, wait for the end of tool selection. Bit 23=1 For single applications the interpreter selects the tool itself; i.e. if correction is required, no synchronization with the main run is needed. (If after selection but before loading the tool becomes no longer usable, a non–correctable alarm may result.)

20320 MD number	**TOOL_TIME_MONITOR_MASK** Activation of tool life monitoring for the spindle here specified		
Default setting: 1	Min. input limit: 1	Max. input limit: 4	
Change becomes effective after: POWER ON	Protection level: 2/4	Units: —	
Data type: DWORD	Valid as of software version: 2		
Significance:	Value = 1: Monitoring is carried out for spindle 1. Value = 2: Monitoring is carried out for spindle 1 and spindle 2.		
Additional references	Description of functions: Memory configuration (S7)		

8.3.3 Creating and loading the configuration file

For tool management commissioning, a configuration file for the turret must be created and loaded in the NCK.

Create a new configuration file or adapt the example from the toolbox.

Note

Make sure that you adapt the data in the configuration file to match the machine data for tool management.

The configuration file must include the following settings:

- Define the type of search strategy
- Define the real magazine
- Define buffer magazine
- Define load magazine
- Define locations for the real magazine
- Define locations for the buffer magazine
- Define the spindle assignment
- Define the locations for the load magazine
- Define the distances for the real magazine

Transfer the configuration file to the NCK.

Example

The configuration file TM_REV12.8x0 is located in the PLC toolbox under \ShopMill_Turn and has been generated for a turret magazine with 12 locations,

1 tool holder, and 2 loading points.

Enter the value 15 in MD 18086 $MN_MM_NUM_MAGAZINE_LOCATION.

Adapt configuration file TM_REV.12.8X0 at the positions marked **bold**.

```
%_N_TO_TMA_INI
CHANDATA(1)
;————————————————————————————————————————————
;TM_REV.12.8x0
;
;tool management configuration for ShopTurn machine with
turret
;————————————————————————————————————————————

;delete old data
;————————————————
```

```
$TC_MAP1[0]=0
$TC_DP1[0,0]=0

;type of search strategy
;————————————————————

$TC_MAMP2=257        ;search forward from 1st location for
                     ;active tools

;definition of magazines
;————————————————————

;real magazine
$TC_MAP1[1]=3        ;magazine type (3: turret, 1: chain)
$TC_MAP3[1]=17       ;magazine status (17: active magazine,
                     ;enabled for loading)
$TC_MAP6[1]=1        ;number of lines in the magazine
$TC_MAP7[1]=12       ;number of magazine locations

;buffer magazine
$TC_MAP1[9998]=7     ;magazine type (7: buffer)
$TC_MAP3[9998]=17
$TC_MAP6[9998]=1
$TC_MAP7[9998]=1     ;number of buffer locations
                     ;(1: spindle)

;load magazine
$TC_MAP1[9999]=9     ;magazine type (9: load magazine)
$TC_MAP3[9999]=17
$TC_MAP6[9999]=1
$TC_MAP7[9999]=2     ;number of load points

;locations of real magazine
;————————————————————————

;location no 1
$TC_MPP1[1,1]=1      ;location kind (1: magazine location)
$TC_MPP2[1,1]=1      ;location type
$TC_MPP3[1,1]=1      ;consider adjacent location (1: on)
$TC_MPP4[1,1]=2      ;location state (2: location free)
$TC_MPP5[1,1]=1      ;location kind index (1: location no 1)

;location no 2
$TC_MPP1[1,2]=1      ;location kind (1: magazine location)
$TC_MPP2[1,2]=1
$TC_MPP3[1,2]=1
$TC_MPP4[1,2]=2
$TC_MPP5[1,2]=2      ;location kind index (2: location no 2)
;
$TC_MPP1[1,3]=1
$TC_MPP2[1,3]=1
$TC_MPP3[1,3]=1
$TC_MPP4[1,3]=2
$TC_MPP5[1,3]=3
;

.
.
.
```

```
$TC_MPP1[1,12]=1
$TC_MPP2[1,12]=1
$TC_MPP3[1,12]=1
$TC_MPP4[1,12]=2
$TC_MPP5[1,12]=12

;

;locations of buffer magazine
;————————————————————————

;spindle
$TC_MPP1[9998,1]=2   ;location kind (2: spindle)
$TC_MPP2[9998,1]=0   ;location type
$TC_MPP3[9998,1]=0   ;consider adjacent
$TC_MPP4[9998,1]=2   ;location state (2: location free)
$TC_MPP5[9998,1]=1   ;location kind index (1: spindle)


;locations of load magazine
;————————————————————————

;1st load point
$TC_MPP1[9999,1]=7   ;location kind (7: load point)
$TC_MPP2[9999,1]=0   ;location type
$TC_MPP3[9999,1]=0   ;consider adjacent
$TC_MPP4[9999,1]=2   ;location status (2: location free)
$TC_MPP5[9999,1]=1   ;location kind index (1: load point 1)


;distance to change position of real magazine
;————————————————————————————————————————————

$TC_MDP2[1,1]=0      ;spindle
$TC_MDP1[1,1]=0      ;1st load point
$TC_MDP1[1,2]=0      ;2nd load point


M17
```

Variable description

The main variables for the configuration file are described here. For a detailed description of the system variables, please refer to:

References: /FBW/, Description of Tool Management Functions

$TC_MAP1[MagazineNo]= Magazine type

- 1: Chain
- 3: Revolver
- 5: Flat magazine
- 7: Internal magazine tool buffer
- 9: Internal magazine loading station

$TC_MAP3[MagazineNo]= Magazine status

- Bit 0 = 1: Active magazine
- Bit 1 = 1: Blocked
- Bit 2 = 1: Magazine is at load position
- Bit 3 = 1: Tool motion is active
- Bit 4 = 1: Enabled for loading

Default: Bit 0 and bit 4 set

$TC_MAP6= Number of magazines
for ShopTurn: 1

$TC_MAP7= Number of locations
e.g. number of buffer locations 1= tool holder

$TC_MAMP2= Type of search strategy

This mask is divided into a right and left byte.

- The right byte describes the tool search (bits 0 and 1).
- The left byte describes the empty location search for the active tool.

A value must be specified for both strategies.

- Bit 0 = 1: Search for active tool with tool designation (1)
- Bit 1 = 1: Search for next tool with tool designation (2)
- Bit 8 = 1: Search from 1st location onwards (256)
- Bit 9 = 1: Search from current location onwards (512)
- Bit 10 = 1: Search from last location backwards (1024)
- Bit 11 = 1: Search from current location backwards (2048)
- Bit 12 = 1: Search from current location symmetrically (4096)

Example: $TC_MAMP2=257 (bit 0=1 and bit 8=1)

Bit 0 = 1: Search for active tool with tool designation,
Bit 8 = 1 Search from 1st location onwards

$TC_MPP1[MagazineNo, LocNo]= Type of location:

- 1 = magazine location
- 2 = Tool holder
- 3 = gripper
- 4 = loader
- 5 = transfer location
- 6 = loading station
- 7 = loading point

Default: Value corresponding to location type

$TC_MPP2[MagazineNo, LocNo]= Type of location:

Any values can be entered here. The values must match the tools to be loaded at the location.

Example:

- Value = 0: Every tool fits in this location
- Value = 1: Location for heavy tools
- Value = 2: Location for long tools

$TC_MPP3[MagazineNo, LocNo]= Considering adjacent location ON = 1

- Value = 1: An adjacent location is considered for the location specified
- Value = 0: No adjacent location is considered for the location specified
- Value = 0: This must be entered for buffer and load locations.

$TC_MPP4[MagazineNo, LocNo]= Location status:

- Bit 0 = 1: Blocked
- Bit 1 = 1: Free/allocated

Default: Bit 1 set

$TC_MPP5[MagazineNo, LocNo]= Location type index

For $TC_MPP1[Magazine no., Location no.]=1 (location type is the magazine location), the location number is entered here. For other location types, the type index is incremented.

$TC_MDP2[MagazineNo, BufferNo.]= Distances between buffer and magazine

Turret revolver: Value = 0

$TC_MDP1[MagazineNo, BufferNo.]= Distances between load points and magazine

Turret revolver: Value = 0

8.4 Commissioning in the PLC

Requirements

- PCU commissioning is performed and the connection to the NCK established.
- NCK commissioning is carried out with the NCK machine data for ShopTurn.
- The basic PLC program is loaded.

General

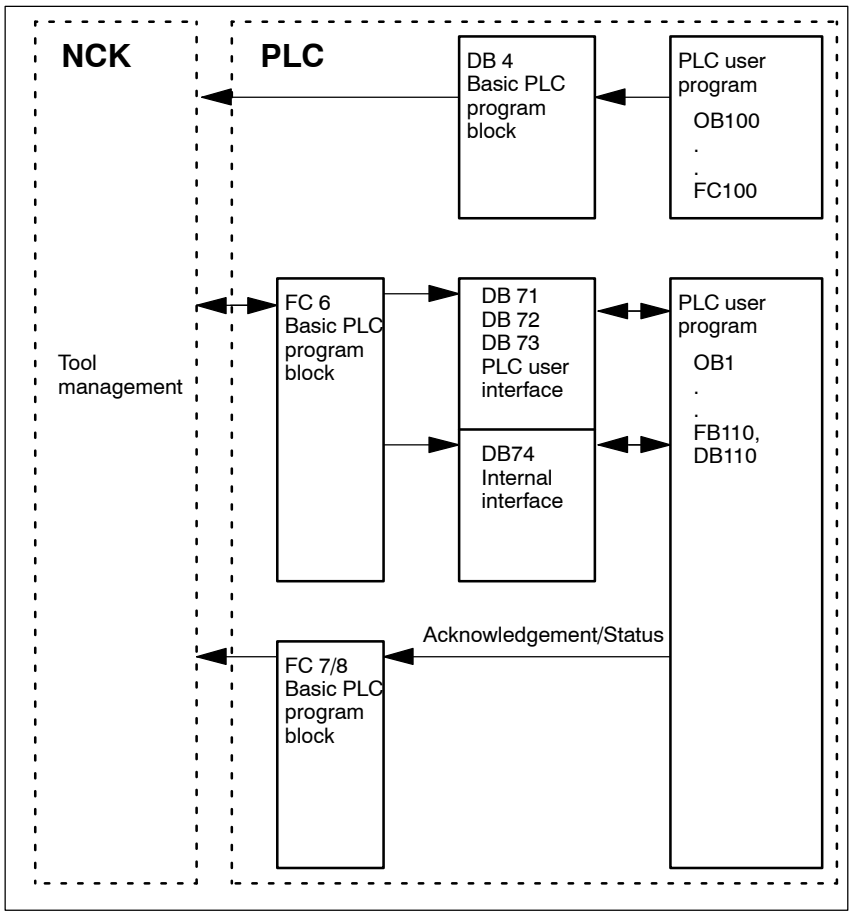


Fig. 8-1 Overview of tool management

FC 6 supplies data blocks DB71, 72, and 73 with the information for the new and old tools. Block FC 6 is called by the basic PLC program and must not be called again in the PLC user program.

To ensure that tool management always knows where the current tool is located, each tool location change must be reported to tool management via FC 7/8 (transfer block). The FC 7/8 (transfer block) is called by the PLC user program (FB110).
FB110 is provided as example.

Data blocks DB71, 72, 73, and 74 are set up automatically. The lengths of the data blocks are determined by the parameters for tool management in DB4. DB4 is written by the PLC user program. FB100 is available as an example for this.

Implementation

Implement call FC 8 and create PLC data (DB4).
For this purpose, either adjust the blocks FC 100 and FB110 in the directory \ShopMill_Turn or use your own blocks.

8.4.1 Example for FC 100 and FB 110

The AWL source TM_REV.AWL is provided as an example for tool management in directory \ShopMill_Turn.

Procedure

- Modify source file TM_REV.AWL (data transfer for turret) and then compile the file.

  Source file TM_REV.AWL contains the following blocks:

  – FC 100 (block for tool management configuration)

  – FB 110, DB 110 (blocks for data transfer in tool management)

  The blocks for transferring tool management data (FB110, DB110) must be adapted to suit the individual machine.

- Load the blocks to the PLC.
- Call the blocks in OB 1 and OB 100:

  – Call FC 100 in OB 100

  – Call FB 110 in OB 1

FC 100

Block FC 100 transfers the PLC data for tool management to DB4.

The PLC data is set up for 2 loading stations (DB71), one tool holder (DB72), and the turret magazine (DB73).
The signals for DB71, DB72, and DB73 are described in section 8.4.2 ”Signal description”.

The ”Real MagLoc” parameter of FC 100 (number of locations on the turret magazine) must be assigned values when FC 100 is called.

FB 110

Block FB 110 controls the data transfer for tool management.

The block includes the following functions:

- Acknowledge load/unload/relocate for 1st loading point
- Acknowledge change for tool holder
- Abort, i.e. negative acknowledgment for the above–mentioned functions

The acknowledgment for these functions can be enabled by the PLC via the input parameter of FB 110, e.g. confirm load/unload via user key.
The default settings for the input parameters are for automatic acknowledgement, so that block FB 110 can be used on test stations involving no further link to the machine; (see Table 8-1). Automatic acknowledgment is performed by means of interface signals from the tool management data blocks; each of these can be skipped.

Data transfer

For data transfer with the turret FB 110 from STL source file TM_REV.AWL can be used.

Table 8-1 Input parameters of FB 110 from TM_REV.AWL

Signal	Type	Default	Comment
Change_Rev_IF1	BOOL	TRUE	Acknowledge change for turret
Load_IF1	BOOL	TRUE	Acknowledge loading for loading point 1
Unload_IF1	BOOL	TRUE	Acknowledge unloading for loading point 1
Relocate_IF1	BOOL	TRUE	Acknowledge relocation for loading point 1
Position_IF1	BOOL	TRUE	Acknowledge positioning to loading point 1
Reset_IF	BOOL	FALSE	Abort for one of the above–mentioned functions

Note

MD 9673 CMM_TOOL_LOAD_STATION defines the interface via which the magazine will be loaded or unloaded.

8.4.2 Signal description

Overview of data blocks

The following data blocks are used by tool management; i.e. they must not be assigned by the PLC user program:

DB 71	for loading/unloading points
DB 72	for tool holder
DB 73	for turret
DB 74	internal data block for tool management

If you need to change the data for magazines, buffers, or loading positions, delete data blocks DB 71 to DB 74 and restart the PLC.

Overview DB71

DB71 Data block	**Signals of load/unload points** NCK –>PLC interface							
Byte	Bit7	Bit6	Bit5	Bit4	Bit3	Bit2	Bit1	Bit0
	Interfaces							
DBB 0	INT 8	INT 7	INT 6	INT 5	INT 4	INT 3	INT 2	INT 1
DBB 1	INT 16	INT 15	INT 14	INT 13	INT 12	INT 11	INT 10	INT 9
DBB 2, 3								
DBB n + 0				NC program positions magazine	Position at loading point	Relocate	Unload	Load
DBB n + 1	Unassigned							
DBB n + 2	Assigned channel (8bit–Int)							
DBB n + 3	Tool management number (8bit–Int)							
DBD n + 4	$P_VDITCP[0] User parameter 0 (DWord)							
DBD n + 8	$P_VDITCP[1] User parameter 1 (DWord)							
DBD n + 12	$P_VDITCP[2] User parameter 2 (DWord)							
DBW n + 16	Identifier for load/unload point (Int), (fixed value 9999)							
DBW n + 18	Location no. of load/unload point (Int)							
DBW n + 20	Magazine no. (source) for loading/relocation/positioning (Int)							
DBW n + 22	Location no. (source) for loading/relocation/positioning (Int)							
DBW n + 24	Magazine number (target) for loading/relocating (Int)							
DBW n + 26	Location number (target) for loading/relocating (Int)							
DBW n + 28 HMI on PLC								Load/unload without moving magazine

Initial addresses of load/unload locations:

Loading/unloading location	1:	n = 4
	2:	n = 34
	3:	n = 64
	4:	n = 94

Example calculation of address DBW n+24 (magazine no. target)

n = (m–1) * len + 4 m = location no. of loading station/point
len = 30 (length of a loading station)

m =2 ; len = 30 n = (2–1) * 30 + 4 ==> n = 34
DBW (34 + 24) = DBW 58

Address for magazine no. target of 2nd load point is DBW 58.

Loading point 1 is intended for loading/unloading in (all) spindles/tool holders. This must be considered when assigning the loading interfaces (applies with ShopTurn on NCU (HMI Embedded sl); with PCU 50.3 this is taken into account automatically). Loading point 1 is also used to relocate/ position tools in any location (e.g. buffer location).

Overview DB72

DB72 Data block	**Spindle as change position** Interface NCK–>PLC							
Byte	Bit7	Bit6	Bit5	Bit4	Bit3	Bit2	Bit1	Bit0
DBB 0	INT 8	INT 7	INT 6	INT 5	INT 4	INT 3	INT 2	INT 1
DBB 1	INT 16	INT 15	INT 14	INT 13	INT 12	INT 11	INT 10	INT 9
DBB 2, 3								
DBB n + 0	Reserved	Detach manual tool	Attach manual tool	OldTool in BL No. (n+42)	TO	Prepare change	Change tool (initiated by: M06)	Obligatory change
DBB n + 1	Unassigned							
DBB n + 2	Assigned channel (8–bit Int)							
DBB n + 3	Tool management number (8bit–Int)							
DBD n + 4	$P_VDITCP[0], user–assignable parameter 0 (DWord)							
DBD n + 8	$P_VDITCP[1], user–assignable parameter 1 (DWord)							
DBD n + 12	$P_VDITCP[2], user–assignable parameter 2 (DWord)							
DBW n + 16	Buffer identifier (Int), fixed value 9998) equals "Target position for new tool"							
DBW n + 18	Relative location (target) in buffer magazine (Int)							
DBW n + 20	Magazine no. (source) for new tool (Int)							
DBW n + 22	Location no. (source) for new tool (Int)							
DBW n + 24	Magazine no. (target) for old tool (Int)							
DBW n + 26	Location no. (target) for old tool (Int)							
DBW n + 28	New tool: Location type (Int)							
DBW n + 30	New tool: Size on left (Int)							
DBW n + 32	New tool: Size on right (Int)							
DBW n + 34	New tool: Size at top (Int)							
DBW n + 36	New tool: Size at bottom (Int)							
DBW n + 38	Tool status for new tool							
	Tool has been used	Tool with fixed loc. code		Prewarning limit reached	Measure tool		Enable tool	Active tool
DBW n + 40	New tool: Internal T number of NCK (Int)							
DBW n + 42	If DBX (n+0.4) = 1, then the buffer location of the old tool must be entered here							
DBW n + 44	Spare							
DBW n + 46	Spare							

Initial addresses of spindles:	Spindle 1: n = 4 Spindle 2: n = 52 Spindle 3: n = 100
n = (m–1)* len + 4	m = Location number of change position len = 48

Note

DBB (n+1) to DBW (n+46) are updated **only** by a T selection.

Overview DB73

DB73 Data block	**Turret as change position** Interface NCK–>PLC							
Byte	Bit7	Bit6	Bit5	Bit4	Bit3	Bit2	Bit1	Bit0
DBB 0	INT 8	INT 7	INT 6	INT 5	INT 4	INT 3	INT 2	INT 1
DBB 1	INT 16	INT 15	INT 14	INT 13	INT 12	INT 11	INT 10	INT 9
DBB 2, 3								
DBB n + 0					T0		Perform change (initiation: T no.	Obligatory change
DBB n + 1	Unassigned							
DBB n + 2	Assigned channel (8–bit Int)							
DBB n + 3	Tool management number (8bit–Int)							
DBD n + 4	$P_VDITCP[0], user–assignable parameter 0 (DWord)							
DBD n + 8	$P_VDITCP[1], user–assignable parameter 1 (DWord)							
DBD n + 12	$P_VDITCP[2], user–assignable parameter 2 (DWord)							
DBW n + 16	Reserved							
DBW n + 18	Reserved							
DBW n + 20	Magazine no. of turret (Int)							
DBW n + 22	Location no. of new tool (Int)							
DBW n + 24	Reserved							
DBW n + 26	Location no. of old tool (Int)							
DBW n + 28	Tool new: loc. type (Int)							
DBW n + 30	Tool new: size left (Int)							
DBW n + 32	Tool new: size right (Int)							
DBW n + 34	Tool new: size top (Int)							
DBW n + 36	Tool new: size bottom (Int)							
DBW n + 38	Tool status for tool							
	Tool was in use	Tool with fixed loc. code		Prewarn. limit reached	Measuring tools		Tool enabled	Active tool
DBW n + 40	Tool new: Internal T no. of NCK (Int)							
DBW n + 42	Reserved							

Initial addresses of turrets: Turret 1: n = 4
Turret 2: n = 48
Turret 3: n = 92

n = (m–1)* len + 4 m = Location number of change position
len = 44

Example for change position 3: n = (3–1)*n 44+ 4= 2*44 + 4= 88 + 4= 92

Note

For a description of data blocks DB71 and DB74, please see
References: /FBW/, Description of Functions Tool Management

Description DB73

DB73 – DBX 0.0 – 0.15	**Active status of interface 1–16**	
Edge evaluation	Signal(s) updated: **Conditional**	Signal(s) valid from SW: **2**
Signal state **1**	Associated interface has a valid data block	
Signal state **0**	Operation for this interface has ended. Is reset by FC 7.	

DB73.DBX(n+0).0	**Command code: Obligatory change**	
Edge evaluation	Signal(s) updated: **Conditional**	Signal(s) valid from SW: **2**
Signal status **1**		
Signal status **0**		
Corresponding to...	Position of involved tools	

DB73.DBX(n+0).1	**Command code: Perform change**	
Edge evaluation	Signal(s) updated: **Conditional**	Signal(s) valid from SW: **2**
Signal state **1**	Execute tool change	
Signal state **0**		

DB73.DBB(n+0).3	**T0**	
Edge evaluation	Signal(s) updated: **Conditional**	Signal(s) valid from SW: **2**
Description	Indicates that T0 has been programmed.	

DB73.DBB(n+2)	**Assigned channel**	
Edge evaluation	Signal(s) updated: **Conditional**	Signal(s) valid from SW: **2**
Description	Number of channel from which the T word was programmed.	

DB73.DBB(n+3)	**Tool management no.**	
Edge evaluation	Signal(s) updated: **Conditional**	Signal(s) valid from SW: **2**
Description	Associated tool management number (TO area) of channel	

Note

The bits in DBB (n+0) (obligatory change, execute change,...) are **not** reset by the system. They are current only if the corresponding interface bit in DBB0 is set to ”1”. However, the bits can be reset by the user if necessary.

DB73.DBD(n+4)	**User parameter 0 (DInt)**	
Edge evaluation	Signal(s) updated: **Conditional**	Signal(s) valid from SW: **2**
Description	If you need to send a value to the PLC via the part program, the transfer can be programmed with $P_VDITCP[0]=(value). Parameters 0–2 are passed with the T command.	

DB73.DBD(n+8)	**User parameter 1 (DInt)**	
Edge evaluation	Signal(s) updated: **Conditional**	Signal(s) valid from SW: **2**
Description	If you need to send a value to the PLC via the part program, the transfer can be programmed with $P_VDITCP[1]=(value);.	

DB73.DBD(n+12)	**User parameter 2 (DInt)**	
Edge evaluation	Signal(s) updated: **Conditional**	Signal(s) valid from SW: **2**
Description	If you need to send a value to the PLC via the part program, the transfer can be programmed with $P_VDITCP[2]=(value);.	

DB73.DBW(n+16)	**Reserved**	
Edge evaluation	Signal(s) updated: **Conditional**	Signal(s) valid from SW: **2**

DB73.DBW(n+18)	**Reserved**	
Edge evaluation	Signal(s) updated: **Conditional**	Signal(s) valid from SW: **2**

DB73.DBW(n+20)	**Magazine no. of new tool**	
Edge evaluation	Signal(s) updated: **Conditional**	Signal(s) valid from SW: **2**
Description	Magazine number of the new tool to be used for machining.	
Corresponding to...	DBW(n+22)	

DB73.DBW(n+22)	**Location number of new tool to be loaded**	
Edge evaluation	Signal(s) updated: **Conditional**	Signal(s) valid from SW: **2**
Description	Location number of the new tool to be used for machining.	
Corresponding to...	DBW(n+20)	

DB73.DBW(n+24)	Reserved	
Edge evaluation	Signal(s) updated: **Conditional**	Signal(s) valid from SW: **2**

DB73.DBW(n+26)	Location no. of old tool to be unloaded	
Edge evaluation	Signal(s) updated: **Conditional**	Signal(s) valid from SW: **2**
Description	Location number of the old tool (used up to now for machining)	

DB73.DBW(n+28)	Tool new: Location type
Description	The location type of the new tool is entered here.
Corresponding to...	Tool size: Left, right, top, bottom

DB73.DBW(n+30)	Tool new: Size left (Int)	
Edge evaluation	Signal(s) updated: **Conditional**	Signal(s) valid from SW: **2**
Description	Specification of new tool size **on left** in half locations.	

DB73.DBW(n+32)	Tool new: Size right (Int)	
Edge evaluation	Signal(s) updated: **Conditional**	Signal(s) valid from SW: **2**
Description	Specification of new tool size **on right** in half locations.	

DB73.DBW(n+34)	Tool new: Size top	
Edge evaluation	Signal(s) updated: **Conditional**	Signal(s) valid from SW: **2**
Description	Specification of new tool size **at top** in half locations.	

DB73.DBW(n+36)	Tool new: Size bottom	
Edge evaluation	Signal(s) updated: **Conditional**	Signal(s) valid from SW: **2**
Description	Specification of new tool size **at bottom** in half locations.	

DB73.DBW(n+38)	Tool status for new tool	
Edge evaluation	Signal(s) updated: **Conditional**	Signal(s) valid from SW: **2**
Description	Bit 0: Active tool Bit 1: Tool enabled Bit 2: Tool disabled Bit 3: Measure tool Bit 4: Prewarning limit reached Bit 5: Tool is being changed Bit 6: Tool is fixed–location–coded Bit 7: Tool was in use	

DB73.DBW(n+40)	Tool new: Internal T no. of NCK	
Edge evaluation	Signal(s) updated: **Conditional**	Signal(s) valid from SW: **2**
Description	Display of internal T no. of NCK for the new tool. Tool management variables can be read/written via FB2/FB 3 using this T no.	

DB73.DBW(n+42)	Reserved	
Edge evaluation	Signal(s) updated: **Conditional**	Signal(s) valid from SW: **2**

8.5 Display machine data

Display machine data can be used to activate certain functions and settings at the operator interface.

MD 9450 $MM_WRITE_TOA_FINE_LIMIT
Limit value for fine wear

MD 9478 $MM_TO_OPTION_MASK
Settings for ShopTurn

MD 9639 $MM_CTM_MAX_TOOL_WEAR
Upper input limit for tool wear

MD 9651 $MM_CMM_TOOL_MANAGEMENT
Tool management variant

MD 9652 $MM_CMM_TOOL_LIFE_CONTROL
Tool monitoring

MD 9663 $MM_CMM_TOOL_DISPLAY_IN_DIAM
Radius/diameter display for tool

MD 9667 $MM_CMM_FOLLOW_ON_TOOL_ACTIVE
Tool preselection active

MD 9671 $MM_CMM_TOOL_LOAD_DEFAULT_MAG
Load tool in default magazine

MD 9672 $MM_CMM_FIXED_TOOL_PLACE
Fixed location coding

MD 9673 $MM_CMM_TOOL_LOAD_STATION
Number of loading point

MD 9674 $MM_CMM_ENABLE_TOOL_MAGAZINE
Display of magazine list

MD 9687 $MM_CMM_TOOL_MOVE_DEFAULT_MAG
Relocate tool to default magazine

For the defaults and description of this machine data please refer to section 7.2 ”Display machine data for ShopTurn”.

8.6 Enable spindle and coolant

Here you can assign a direction of rotation (CW/CCW/OFF) and coolant to a tool.

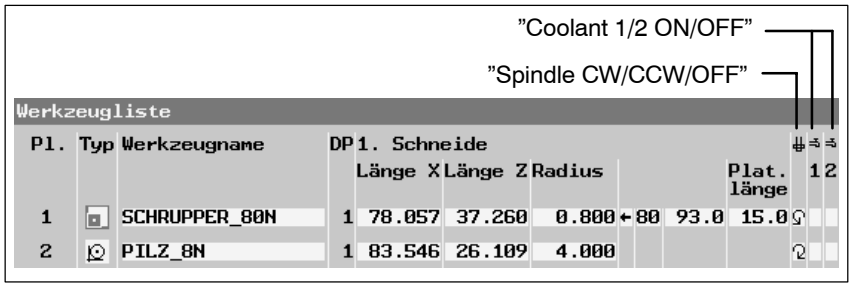


Fig. 8-2 Tool list: Coolant and direction of spindle rotation

Coolants are assigned to the appropriate M commands via the following machine data:
MD 9680 $MM_CMM_M_CODE_COOLANT_I
MD 9681 $MM_CMM_M_CODE_COOLANT_II
MD 9668 $MM_CMM_M_CODE_COOLANT_I_AND_II
MD 9686 $MM_CMM_M_CODE_COOLANT_OFF

8.7 Moving the turret manually

To bring another tool located in the turret into machining position, move the turret manually via the machine control panel. Any manual movement must be reported to the NCK so that the correction values for the new tool can be calculated. You can implement this check–back signal providing that the option "actions that go across modes (asynchronous subprograms ASUB and synchronized actions in all modes)" is set (Order No. 6FC5 800–0AM43–0YB0).

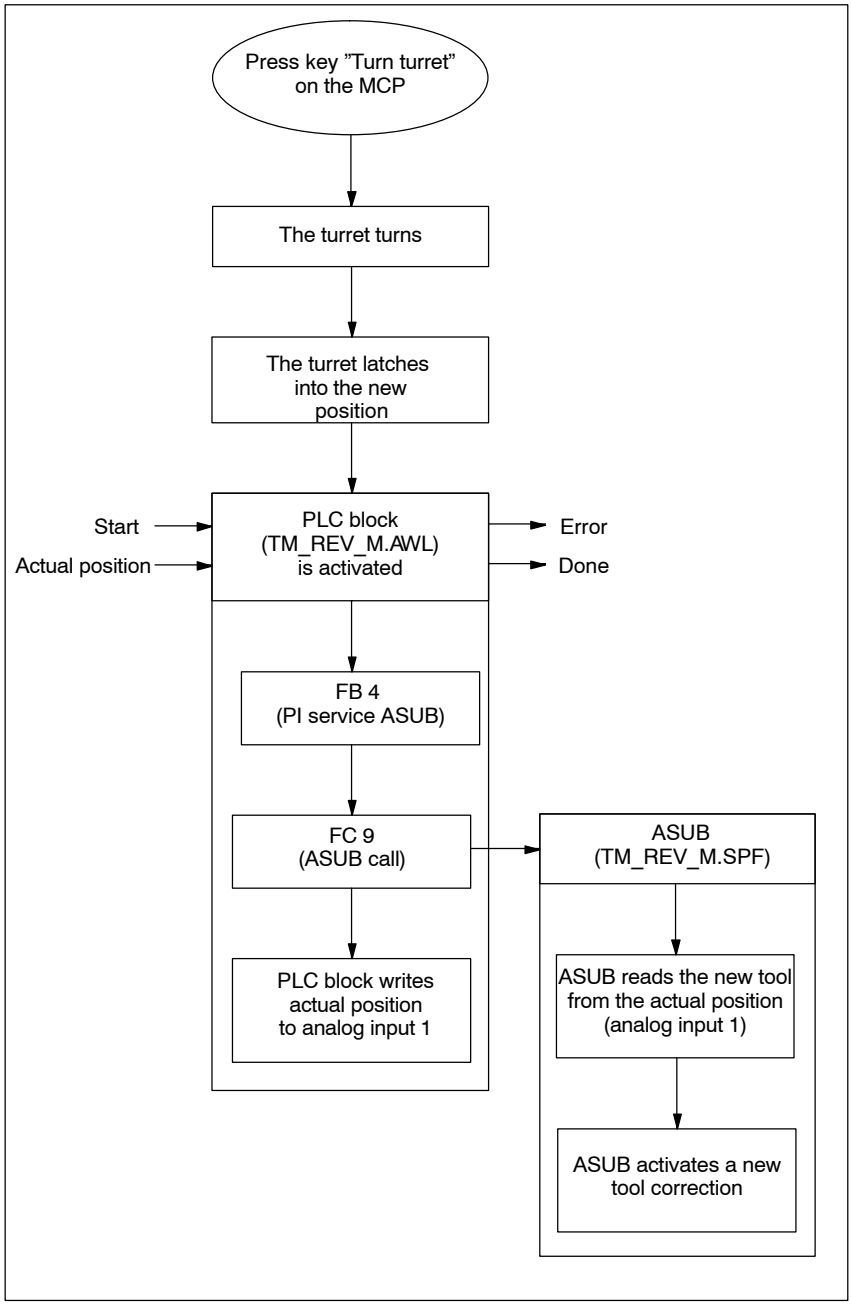


Fig. 8-3 Example of check–back signal from tool number to NCK

The ASUB derives the tool number from the location number (actual position) that it has received from the PLC block and forwards this to the NCK.

The toolbox contains the sources TM_REV_M_GR.AWL in German mnemonics, TM_REV_M_UK.AWL in English mnemonics, and the ASUB TM_REV_M.SPF. The PLC block and ASUB use interrupt 8 and analog input 1. Please set the following machine data:

MD 11602 $MN_ASUP_START_MASK = H3
Ignore stop reasons for ASUB

MD 11604 $MN_ASUP_START_PRIO_LEVEL = H64
Priorities for MD 11602 $MN_ASUP_START_MASK effective

MD 20116 $MC_IGNORE_INHIBIT_ASUP, bit 7 = 1
(ASUB on interrupt 8 can always be executed.)
If another interrupt number is used, the corresponding bit must be set.

MD 10300 $MN_FASTIO_ANA_NUM_INPUTS = Total number of analog inputs used.

MD 10320[0] $MN_FASTIO_ANA_INPUT_WEIGHT = 32767
Weighting of the analog input (index 0 corresponds to input 1)

You might have to adapt the PLC block and the ASUB. Please note the following:

- Assign new block numbers in the symbols table.
- Make sure that the interrupt numbers are identical when you initialize (FB4 call) and call them (FC9 call) (default setting 8).
- If necessary, adapt the interface signals to the analog input.
- Compile and load the TM_REV_M.
- Call up the function block in the user PLC and parameterize it.
- Set interface VAR_INPUT to start and combine the VAR_OUTPUT signals for the check–back signal of the block in your user PLC.
- In your user PLC make sure that the tool turret cannot be restarted manually while an ASUB is being executed. This would otherwise cause the incorrect actual turret position to be detected.

Note

Calling the TM_REV_M.SPF starts a new tool change call (change request at turret interface in DB 73). If all is correct, setpoint position = actual position and no further movement of the turret is triggered. However, if an incorrect actual position is returned, this could cause the turret to swivel again. You should therefore only permit manual movement of the turret in NCK mode JOG and only permit the change request via the interface signal in DB 73 in NCK mode AUTO.

Note

If you permit manual movement of the turret – even if the program has not been interrupted, you must ensure that the ASUB is executed without error and that the interrupted program can be resumed.

8.8 Configuring the operator interface

There are two ways you can modify the existing operator interface for the tool management system:

- In tool management, in addition to the tools, wear, and magazine lists, an additional list can also be activated on the 3rd horizontal softkey; (see section 8.8.1 ”Integrating additional list”).
- In these lists, depending on the tool, all existing or user–defined parameters can be configured; (see section 8.8.2 ”Modifying lists”).

8.8.1 Integrating additional list

To activate an additional list proceed as follows:

- Set machine data
  9478 $MM_TO_OPTION_MASK, Bit 2 = 1
  Activate additional list

  In tool management you can now, by means of the 3rd horizontal softkey, call up a list with the following parameters :
  – ”Length Y”
  – ”Wear length Y”
  – ”H number” .

  Parameter ”H number” will only be displayed if ShopTurn is set up for ISO dialects (see section 11.4 ”ISO dialects”).
- Specifying texts for additional list (optional)
  If you want to change the softkey label (OEM tool list) and header (OEM tool list) of the additional list, you must define new texts (see chapter 8.8.7 ”Defining texts”).

8.8.2 Configuring lists

To modify existing lists (including an integrated additional list) you must proceed as follows:

Activate the configuration file

Set the following machine data:

MD 9478 $MM_TO_OPTION_MASK, bit 8
Analyze the file TO_TURN.INI

The configuration file TO_TURN.INI, in which you configure changes from the standard settings, is analyzed (see chapter 8.8.3 ”Creating the configuration file”).

Activate use of OEM data

If you use user–defined OEM data, set the following machine data:

MD 18080 $MN_MM_TOOL_MANAGEMENT_MASK, Bit 2
Provide memory for user data

MD 18094 $MN_MM_NUM_CC_TDA_PARAM
Number of user–defined parameters

MD 18095 $MN_MM_TYPE_CC_TDA_PARAM[n]=4
Data type (REAL) of user–defined parameters

MD 20310 $MC_TOOL_MANAGEMENT_MASK, Bit 2
Activate user functions

Texts for user–defined parameters (optional)
If you want to change the column headers and cursor texts of the user–defined parameters, define new texts (see chapter 8.8.7 ”Defining texts”).

Adapt the configuration file

Specify all changes changes with respect to the default settings in the configuration file TO_TURN.INI; (see section 8.8.3 ”Creating the configuration file”).

8.8.3 Creating the configuration file

You must save all changes made in the lists with respect to the default setting in configuration file TO_TURN.INI.

Please note the following:

- After the column ”DP number” you can define further columns and also hide existing columns.
- The number of columns displayed is limited by the width of the window because each column has a fixed width. Horizontal scrolling is not possible.
- Per column define cutting edge parameters only, tool parameters only, or magazine parameters only.
- Define the cutting edge parameters in consecutive columns.

The sample file TO_TURN.INI, which can be adapted, is located in ShopTurn on NCU (HMI Embedded sl) under
CYCLE_STORAGE\SEQUENCER_CYCLES\TEMPLATES_ENG.
The adapted file is then found on the CF card (NCU 7x0) under
– oem\sinumerik\hmi\cfg\
– user\sinumerik\hmi\cfg\.

For ShopTurn on PCU 50.3, this example is located on the supplied CD under Tools\TEMPLATES_ENG.
The file is copied to the directory OEM or USER.

Syntax

Entries in the configuration file are subject to the following syntax rules; (see also example at end of this section):

First specify in which tool management list you would like to make changes.
`[DISPLAY_IDENTIFIER]`
DISPLAY_IDENTIFIER: Tool management list

Next define the changes themselves:

- Modify a specific column for all tools:
  `COLUMNx=CONTENTS_IDENTIFIER`
  COLUMN: Column command
  x: Column number, 1 – 12
  CONTENTS_IDENTIFIER: Parameter or property of tool
- Modify a specific column for a specific tool:
  `TOOL_IDENTIFIER = x=CONTENTS_IDENTIFIER`
  TOOL_IDENTIFIER: Tool type
- Define different columns for each tool:
  ```
  TOOL_IDENTIFIER = CONTENTS_IDENTIFIER / CONTENTS_IDENTIFIER / ...
  TOOL_IDENTIFIER = CONTENTS_IDENTIFIER / CONTENTS_IDENTIFIER / ...
  ...
  ```

  The parameters or properties of a tool are listed one after the other by means of a CONTENTS_IDENTIFIER and are separated from one another by a slash "/". If the default setting is to be used for individual columns, you still need to insert a forward slash.

  If, for example, you only want to modify the last few columns, specify the number (x) of the first column you would like to modify and list the respective CONTENTS_IDENTIFIERs for the subsequent columns.
  ```
  TOOL_IDENTIFIER = x=CONTENTS_IDENTIFIER / CONTENTS_IDENTIFIER /
  ..
  ```

Note

Using the COLUMN command, you can first define one column identically for all tools and then adapt it specifically for individual tools.

You can mark comments by means of a semicolon (;) symbol.

If errors occur during evaluation of the configuration file, refer to the TO_INI_F.LOG file for a description of the error. With ShopTurn on NCU (HMI Embedded sl), the file is located in drive I: and with the PCU 50.3 in the directory F:\MMC0W32\TMP.

Identifier

The following tables list the SCREEN_ IDENTIFIERS, TOOL_ IDENTIFIERS, and CONTENTS_ IDENTIFIERS available for defining the columns.

Table 8-2 SCREEN_IDENTIFIER

SCREEN_IDENTIFIER	Tool management list
TOOL_LIST	Tool list
TOOL_LIST_2ND_EDGE	Tool list, further cutting edges
TOOL_WEAR	Tool wear list
TOOL_WEAR_2ND_EDGE	Tool wear list, further cutting edges
TOOL_MAGA	Magazine list
TOOL_LIST_OEM	Additional list
TOOL_LIST_OEM_2ND_EDGE	Further cutting edges

Table 8-3 TOOL_IDENTIFIER

TOOL_IDENTIFIER	Tool
SHANK_END_CUTTER_TR	(End) cutter
END_MILL_CUTTER_TR	Facing tool
POINTED_DRILL_TR	(Twist) drill
ROUGHER	Roughing tool
FINISHER	Finishing tool
PARTING_OFF	Plunge cutter
SCREW_CUTTER	Threading tool
BUTTON	Button tool
STOPPER	Stop
TURN_DRILL	Rotary drill
TO_SCREW_TAP_TR	Screw tap
3DTRACER_TR	3D probe

Table 8-4 CONTENT_ID

CONTENT_ID	Parameter type *	Parameter or property	Field width in number of characters
EMPTY		Empty field	
NOT_USED		Empty column	
LENGTH1	S	Length X	7
LENGTH2	S	Length Z	7
LENGTH3	S	Length Y	7
RADIUS	S	Radius	7
RADIUS_ DIAM	S	Radius with possible diameter calculation	7

Table 8-4 CONTENT_ID

CONTENT_ID	Parameter type *	Parameter or property	Field width in number of characters
RADIUS_ DIAM9	S	Radius with possible diameter calculation – rotary drill bit radius	5
ANGLE_TR	S	Angle	5
CUTTDIR	S	Reference direction for holder angle	1
PLATELEN	S	Tip length	5
PLATEWID	S	Tip width	5
PLATEANG	S	Tip angle	2
PITCH	S	Pitch for screw tap	5
HOLDERANG	S	Holder angle	5
N	S	Number of teeth	2
SPINDLE	S	Spindle direction	1
COOL1	S	Cooling water 1	1
COOL2	S	Cooling water 2	1
DLENGTH1	S	Wear length X	7
DLENGTH2	S	Wear length Z	7
DLENGTH3	S	Wear length Y	7
DRADIUS	S	Wear radius	6
DRADIUS_ DIAM	S	Wear radius with possible diameter calculation	6
T_OR_C	W	Method of wear monitoring	1
P_TIME	S	Tool life	7
PW_TIME	S	Tool life warning limit	7
P_COUNT	S	Quantity	7
PW_COUNT	S	Pre–warning limit for count	7
P_WEAR	S	Wear	7
PW_WEAR	S	Prewarning limit for wear	7
T_LOCKED	W	Tool disabled	1
T_SIZE	W	Oversized tool	1
T_FIXED	W	Tool in fixed location	1
T_STATE	W	Status code: Active/prewarning limit reached/blocked	5

Table 8-4 CONTENT_ID

CONTENT_ID	Parameter type *	Parameter or property	Field width in number of characters
T_SIZE_ LONG	W	Tool size This parameter is not contained in the ShopTurn standard tool list. Column header: LROU Cursor text: Tool size Input: Number of adjacent half locations (maximum 7) to be blocked. The first half location is always the magazine location in which the tool is mounted. The number of adjacent half locations is specified as a four–digit number. The first digit is the number of adjacent locations to the left, the second to the right, the third above, and the fourth below. Requirement input: The tool must be located outside a magazine.	4
T_MAG_ PLACE_TYPE	W	Magazine location type This parameter is not contained in the ShopTurn standard tool list. Column header: Location type Cursor text: Magazine location type Input: Number of magazine location type Entry requirement: The tool must be located outside a magazine.	5
P_LOCKED	M	Magazine location locked	6
MAG_T_ LOCKED	W	Display only: Tool disabled	1
MAG_T_SIZE	W	Display only: Oversized tool	1
MAG_T_ FIXED	W	Display only: Tool in fixed location	1
H_NBR	S	H number of an ISO dialect program	3
TPC1	W	Parameter 1	7
TPC2	W	Parameter 2	7
TPC3	W	Parameter 3	7
TPC4	W	Parameter 4	7
TPC5	W	Parameter 5	7
TPC6	W	Parameter 6	7
TPC7	W	Parameter 7	7
TPC8	W	Parameter 8	7
TPC9	W	Parameter 9	7
TPC10	W	Parameter 10	7

***** Parameter types: S = cutting edge data, W = tool data, M = magazine data.

Parameter ”H number” will only be displayed if ShopTurn is set up for ISO dialects (see section 11.4 ”ISO Dialects”).

Example

```
;==========================================
[TOOL_LIST_OEM]     ; screen OEM tool list;
==========================================
;Default values for columns
COLUMN1 = LENGTH3
COLUMN2 = DLENGTH3
COLUMN3 = TPC1
COLUMN4 = TPC2
COLUMN5 = NOT_USED
COLUMN6 = NOT_USEDY
COLUMN7 = NOT_USED
COLUMN8 = NOT_USED
COLUMN9 = NOT_USED
COLUMN10= NOT_USED
COLUMN11= NOT_USED
COLUMN12= NOT_USED
```

In the example list, the following columns are to be displayed:
Column 1: Tool length Y
Column 2: Wear Y
Column 3: 1. user–specific parameter
Column 4: 2. user–specific parameter

You have set the following machine data for the example shown:

MD 18080 $MN_MM_TOOL_MANAGEMENT_MASK, bit 2
Provide memory for user data

MD 18094 $MN_MM_NUM_CC_TDA_PARAM=2
For each of the user–defined parameters (TPC1 and TPC2)

MD 18095 $MN_MM_TYPE_CC_TDA_PARAM[n]=4
Data type (REAL) of user–defined parameters

MD 20310 $MC_TOOL_MANAGEMENT_MASK, bit 2
Activate user functions

8.8.4 Adapting individual parameters

Using the configuration file to_turn.ini, you can change the default settings for specific tool parameters.

Table 8-5 PARAMETER_IDENTIFIER

PARAMETER_IDENTIFIER	Parameter
T_MAG_PLACE_TYPE	Magazine location type
TPC1	Parameter 1
TPC2	Parameter 2
TPC3	Parameter 3
TPC4	Parameter 4
TPC5	Parameter 5

Table 8-5 PARAMETER_IDENTIFIER

PARAMETER_IDENTIFIER	Parameter
TPC6	Parameter 6
TPC7	Parameter 7
TPC8	Parameter 8
TPC9	Parameter 9
TPC10	Parameter 10

Magazine location type

You can change the input field into a toggle field for the parameter ”Magazine location type”. The following values are entered: You must define the values of the toggle field in the section T_MAG_PLACE_TYPE, by assigning values to cursor texts.

Value = Text–ID

Valid Text–IDs and information on defining texts are provided in Chapter 8.8.7 ”Defining texts”).

Example

```
[T_MAG_PLACE_TYPE]
DEFAULT = 89891
1 = 89868
2 = 89869
5 = 89872
9 = 89876
```

In this example, in the column ”location type” you can select the values 1, 2, 5 or 9 using the ”alternative” softkeys. Here, the values have been randomly selected. Cursor texts that have been saved are displayed for the selected values. If the actual value of the parameter devices from the configured values, then the text, saved as DEFAULT, is displayed as cursor text.

OEM parameter

You can change the input fields into toggle fields for the OEM tool parameters. In so doing, specific values are entered.
You must define the values of the toggle field in Sections TPC1 ... TPC10, by assigning cursor texts to the values (refer to Chapter 8.8.7 ”Defining texts”). Also refer to the example on ”Magazine location type” parameters.

You can define the width of the toggle field or the input field to a specific number of characters. The standard width is 7 characters.

Further, for the input field, you can also specify the data type (integer or double), however, only if you have not specified a toggle field. If you do not change the data type, then, the double data type set as default, is used.

Example

```
[TPC1]
WIDTH = 2
TYPE = INTEGER
```

In this example, the input fields of the column OEM tool parameter 1 are 2 characters wide. You can only enter integer values in the input fields.

8.8.5 Defining texts for the magazine locations on the intermediate buffer

The magazine locations of the buffer are shown, as standard using symbols in the tool manager lists. You can assign texts to the location using the configuration file to_turn.ini.

ID for the buffer:
BUFFER

In the section BUFFER, the magazine locations of the buffer are assigned texts.

Magazine location = Text–ID

Valid text IDs and information on defining texts are provided in Chapter 8.8.7 ”Defining texts”).

Example

```
[BUFFER]
DEFAULT = 89768
1 = 89790
2 = 89791
3 = 89792
```

In this example, three magazine locations of the buffer (spindle, gripper 1 and gripper 2) are assigned texts for the display in the tool manager lists. A default text is displayed for additional magazine locations of the buffer.

8.8.6 Designating a load magazine in the tool list sorted according to magazines

If the tool list is sorted according to magazines, all of the magazine locations are displayed in an increasing sequence according to magazine location and magazine number. Using the configuration file to_turn.ini, a magazine can be highlighted as a load magazine by sorting this magazine so that it is directly at the beginning of the tool list (sorted according to magazines).

ID for the load magazine:
OEM_LOAD_MAGAZINE

Example

[OEM_LOAD_MAGAZINE]
MAGAZINE = 3

In this example, magazine 3 is configured as load magazine,. The magazine locations of magazine 3 are displayed directly at the beginning of the tool list that is sorted according to magazine locations.

8.8.7 Define texts

Texts (softkey name and list header for additional lists, column headers, and cursor texts for user–defined parameters) are assigned to specific text numbers in a text file.

These are subject to the following syntax:
`Text number 0 0 "Text"`

The two parameters 2 and 3 separated by blanks are control characters for text output and must always be set to 0.

The column header in the parameter name can consist of three lines, each line with its own text number; (see Table 8-6).

Table 8-6 Text assignment

Text type	Text number
3rd horizontal softkey	89923
List header	89924
Column header parameter TPC1	89925, 89926, 89927
Cursor text parameter TPC1	89931
Column header parameter TPC2	89928, 89929, 89930
Cursor text parameter TPC2	89932
Column header parameter TPC3	89953, 89954, 89955
Cursor text parameter TPC3	89965
Column header parameter TPC4	89937, 89938, 89939
Cursor text parameter TPC4	89949
Column header parameter TPC5	89940, 89941, 89942

Table 8-6 Text assignment

Text type	Text number
Cursor text parameter TPC5	89950
Column header parameter TPC6	89943, 89944, 89945
Cursor text parameter TPC6	89951
Column header parameter TPC7	89956, 89957, 89958
Cursor text parameter TPC7	89966
Column header parameter TPC8	89946, 89947, 89948
Cursor text parameter TPC8	89952
Column header parameter TPC9	89959, 89960, 89961
Cursor text parameter TPC9	89967
Column header parameter TPC10	89962, 89963, 89964
Cursor text parameter TPC10	89968

Example:
```
89924 0 0 "Tool data"
```

The texts must not exceed the following number of characters:
Softkey: 6
List header: 20
Column header parameter: 7
Cursor text parameter: 45

A line break in the softkey text can be made by inserting two consecutive blanks.

Note

The columns for parameters TPC1 and TPC2 are already preset with example texts; these can be modified.

Magazine location type, OEM parameter, magazine locations of the buffer

The texts for the magazine locations of the buffer and the cursor texts for the parameter, magazine location type and OEM parameter must be located as toggle field in the number range 89700 to 89899.

ShopTurn on NCU (HMI Embedded sl)

With ShopTurn on NCU (HMI Embedded sl), enter the texts and numbers in the text file ALUC.TXT. There is an ALUC.TXT text file in each language directory.

PCU 50.3

With the PCU 50.3 enter the texts and numbers in the text file F:\DH\CUS.DIR\ALUC_xx.COM. If the ALUC_xx.COM file does not exist, you need to create it in the CUS.DIR directory.
The texts are assigned to a language by means of the text file name. In the text file name ”xx” is replaced by one of the following codes:

Table 8-7 Language assignment

Abbreviation XX	Language
gr	German
uk	English
fr	French
it	Italian
sp	Spanish
nl	Dutch
dk	Danish
fi	Finnish
sw	Swedish
pl	Polish
tr	Turkish
ch	Simplified Chinese
tw	Chinese (Traditional)
ko	Korean
hu	Hungarian
po	Brazilian Portuguese
ru	Russian
cz	Czech
ja	Japanese

In the file F:\USER\MBDDE.INI or F:\OEM\MBDDE.INI add the following line in the section [TextFiles]:

```
UserZYK=F:\dh\cus.dir\aluc_
```

8.9 Reading in tool data

Tool data that has been defined on an external tool presetting device can be imported directly into ShopTurn tool management.

Save the tool data as an INI file.

The header in the file must be exactly as follows:

```
;TOOL MAGAZIN ZEROPOINT,TOOL=2,MAGAZIN=0,NPV=0,BNPV=0
```

However, you can modify the values for ”Tool” and ”Magazine”.

Tool=1:	Delete all existing tools in tool management and replace with new tools.
Tool=2:	Add new tools to tool management
Magazine=0:	Do not evaluate magazine location number ($TC_MPP6)
Magazine=1:	Evaluate magazine location number ($TC_MPP6)

Note

Always keep to the precise order (including blanks) in the header. If you enter an incorrect header line, the INI file on opening will be evaluated as a G code program. Starting the program with ”Cycle start” causes existing data in tool management to be overwritten by the data contained in the program.

The tool data must be assigned the following variables in the INI file, where:
x = tool number
y = cutting edge number

Table 8-8 Variable assignment

Variable	Description	Value
$TC_TP1[x]	Duplo number	Number
$TC_TP2[x]	Tool name	Name
$TC_TP3[x]	Number of adjacent half locations on the left to be blocked for oversized tools.	ShopTurn default setting 1 = Do not block adjacent location or 2 = Block half of adjacent location on the left
$TC_TP4[x]	Number of adjacent half locations on the right to be blocked for oversized tools.	ShopTurn default setting 1 = Do not block adjacent location or 2 = Block half of adjacent location on the right
$TC_TP5[x]	Number of adjacent half locations above to be blocked for oversized tools.	ShopTurn default setting 1 = Do not block adjacent location
$TC_TP6[x]	Number of adjacent half locations below to be blocked for oversized tools.	ShopTurn default setting 1 = Do not block adjacent location
$TC_TP7[x]	Magazine location type	Number
$TC_TP8[x]	Tool status	Bit 1 = 1: Tool enabled Bit 2 = 1: Tool disabled Bit 4 = 1: Prewarning limit reached

Table 8-8 Variable assignment

Variable	Description	Value
$TC_TP9[x]	Tool monitoring	Bit 0 = 1: Tool life monitoring active Bit 1 = 1: Workpiece count monitoring active Bit 2 = 1: Wear monitoring active
$TC_TPC1[x]	User–defined parameter 1	
$TC_TPC2[x]	User–defined parameter 2	
$TC_TPC3[x]	User–defined parameter 3	
$TC_TPC4[x]	User–defined parameter 4	
$TC_TPC5[x]	User–defined parameter 5	
$TC_TPC6[x]	User–defined parameter 6	
$TC_TPC7[x]	User–defined parameter 7	
$TC_TPC8[x]	User–defined parameter 8	
$TC_TPC9[x]	User–defined parameter 9	
$TC_TPC10[x]	User–defined parameter 10	
$TC_DP1[x,y]	Tool type	120: Cutter 140: Facing tool 200: Drill 240: Tap 500: Roughing tool 510: Finishing tool 520: Plunge–cutter 540: Threading tool 550: Button 560: Rotary drill 580: 3D probe 750: Stop
$TC_DP2[x,y]	Length of cutting edge	When machining behind the turning center, the tool tip is pointing to: 1: Top right 2: Top left 3: Bottom left 4: Botton right 5: Right 6: Top 7: Left 8: Bottom 9: Tool tip = tool nose center
$TC_DP3[x,y]	Length or Length X	Number [mm] Number [mm]
$TC_DP4[x,y]	Length 2 or Length Z	Number [mm] Number [mm]
$TC_DP5[x,y]	Length 3 or Length Y	Number [mm] Number [mm]
$TC_DP6[x,y]	Radius or internal radius (facing tool)	Number [mm] Number [mm]
$TC_DP7[x,y]	External radius	Number [degrees]
$TC_DP8[x,y]	Tip length	Number [mm]
$TC_DP9[x,y]	Tip width	Number [mm]
$TC_DP10[x,y]	Holder angle	Number [degrees]

Table 8-8 Variable assignment

Variable	Description	Value
$TC_DP11[x,y]	Reference direction for holder angle or tool angle/chamfer angle (facing tool)	1: Traversing motion in negative direction, x– 2: Traversing motion in positive direction, x+ 3: Longitudinal travel in negative direction, z– 4: Longitudinal travel in positive direction, z+ Number [degrees]
$TC_DP12[x,y]	Wear length or Wear length X	Number [mm] Number [mm]
$TC_DP13[x,y]	Wear length 2 or Wear length Z	Number [mm] Number [mm]
$TC_DP14[x,y]	Wear length 3 or Wear length Y	Number [mm] or Number [mm]
$TC_DP15[x,y]	Wear radius	Number [mm]
$TC_DP24[x,1] $TC_DP24[x,y] $TC_DP24[x,y]	Number of teeth (milling tool) or tool tip angle (drill) or tool clearance angle (turning tool) ShopTurn calculates the tip angle from the tool clearance angle and displays this in tool management: Tip angle = 180 – tool clearance angle – holder angle.	Number Number [degrees] Number [degrees]
$TC_DP25[x,1]	Direction of spindle rotation Coolant M function	Bit 8 and bit 9 = 0: Spindle stop Bit 8 = 1: Spindle rotates clockwise Bit 9 = 1: Spindle rotates counter–clockwise Bit 10 = 1: Cooling water 1 ON Bit 11 = 1: Cooling water 2 ON Bit 0 = 1: M function 1 Bit 1 = 1: M function 2 Bit 2 = 1: M function 3 Bit 3 = 1: M function 4
$TC_DPH[x,y]	H number of an ISO dialect program	Number
$TC_MOP1[x,y]	Tool life warning limit	Number [min]
$TC_MOP2[x,y]	Tool life	Number [min]
$TC_MOP3[x,y]	Pre–warning limit for count	Number
$TC_MOP4[x,y]	Quantity	Number
$TC_MOP5[x,y]	Prewarning limit for wear	Number [mm]
$TC_MOP15[x,y]	Maximum wear	Number [mm]
$TC_MPP6[n,m]	Magazine location number	Number n: Magazine number m: Magazine location number

For bit–coded parameters you must specify a hexadecimal value.

The H number for ISO dialect programs will only be evaluated if ShopTurn is set up for ISO dialects (see Section 11.4 ”ISO Dialects”).

If you do not define parameters, tool management will subsequently set them to zero.

You must program the command ”M30” at the end of the file.

For information about importing tool data into tool management, please refer to:
References: /BATsl/, ShopTurn Operating/Programming Guide

Example

```
;TOOL MAGAZINE ZEROPOINT,TOOL=2,MAGAZINE=0,NPV=0,BNPV=0
$TC_TP1[1]=1                ;Duplo number
$TC_TP2[1]=ROUGHING TOOL    ;Tool ”ROUGHING TOOL”
$TC_TP3[1]=1                ;Adjacent location on left unoccupied
$TC_TP4[1]=1                ;Adjacent location on right unoccupied
$TC_TP5[1]=1                ;Adjacent location above unoccupied
$TC_TP6[1]=1                ;Adjacent location below unoccupied
$TC_TP7[1]=1                ;Magazine location type
$TC_TP8[1]=2                ;Tool enabled
$TC_TP9[1]=1                ;Tool life monitoring
$TC_DP1[1,1]=500            ;Tool type, roughing tool
$TC_DP2[1,1]=3              ;Cutting edge position (left bottom)
$TC_DP3[1,1]=35.92          ;Length X
$TC_DP4[1,1]=67.89          ;Length Z
$TC_DP6[1,1]=0.6            ;Radius
$TC_DP8[1,1]=14             ;Tip length
$TC_DP10[1,1]=93            ;Holder angle
$TC_DP11[1,1]=3.0           ;Reference direction for holder angle
                            ;–Z direction
$TC_DP24[1,1]=2             ;Tool clearance angle
...
M30                         ;End of program
```

■

Additional Functions

9

9.1 Measuring cycle

9.1.1 Brief description

Overview

You can use a measuring cycle for automatic measurement of tools on turning machines with ShopTurn.

This is achieved simply by connecting a touch trigger probe to the control.

References:

/FB2/, Function Manual on Expansion Functions, Measuring (M5);
/GDsl/, Device Manual NCU, SINUMERIK 840D sl
/IDsl/, Commissioning Manual CNC: NCK, PLC, drive, SINUMERIK 840D sl,

Measuring cycle

The measuring cycle is supplied with ShopTurn.

You have to adapt the measuring cycle data to the specific characteristics of the machine.

Function test

The probe operates internally with the command MEAS.

The function test of the probe is conducted using a part program.

References: /PGA/, Programming Manual on Production Planning;
/BNM/, Programming Manual on Measuring cycles

9.1.2 Display machine data for measuring cycle

9749 MD number	CMM_ENABLE_MEAS_T_AUTO Enable automatic tool measurement		
Default setting: 1	Min. input limit: 0	Max. input limit: 1	
Changes effective as from NOW	Protection level: 3/4	Units: –	
Data type: WORD	Valid as of software version: ShopTurn 6.4		
Significance:	This MD enables the ”Automatic tool measurement” function on the user interface. 0 = ”Automatic tool measurement” function is not displayed 1 = ”Automatic tool measurement” function is displayed		

9751 MD number	CMM_MEAS_T_PROBE_INPUT Measuring input for tool probe		
Default setting: 0	Min. input limit: 0	Max. input limit: 1	
Changes effective as from NOW	Protection level: 3/4	Units: –	
Data type: BOOL	Valid as of software version: ShopTurn 6.3		
Significance:	This MD is used to stipulate the number of the input for the tool probe for the main spindle. 0 = Measuring input 1 is activated 1 = Measuring input 2 is activated		

9754 MD number	CMM_MEAS_DIST_TOOL_LENGTH Maximum measurement distance for tool length for rotating Spindle		
Default setting: 10	Min. input limit: 0.001	Max. input limit: 1000	
Changes effective as from NOW	Protection level: 3/4	Units: mm	
Data type: DOUBLE	Valid as of software version: ShopTurn 6.4		
Significance:	This MD stipulates the maximum measurement distance before and after the expected switching position (tool length) for measuring the tool length with rotating spindle. If no switching signal is output within this range the error message ”Probe does not switch” is output.		

9759 MD number	CMM_MAX_CIRC_SPEED_ROT_SP Maximum circumferential speed for tool measurement for rotating Spindle		
Default setting: 100	Min. input limit: 1	Max. input limit: 200	
Changes effective as from NOW	Protection level: 3/4	Units: m/min	
Data type: DOUBLE	Valid as of software version: ShopTurn 6.3		
Significance:	This MD is used to stipulate the maximum permissible circumferential speed of the tools to be measured for tool measurement with rotating spindle. The permissible spindle speed for carrying out tool measurement is calculated according to this MD.		

9760 MD number	CMM_MAX_SPIND_SPEED_ROT_SP Maximum speed for tool measurement for rotating Spindle		
Default setting: 1000	Min. input limit: 100	Max. input limit: 25000	
Changes effective as from NOW	Protection level: 3/4	Units: rev/min	
Data type: DOUBLE	Valid as of software version: ShopTurn 6.3		
Significance:	This MD is used to stipulate the maximum permissible rotational speed of the tools to be measured for tool measurement with rotating spindle.		

9771 MD number	CMM_MAX_FEED_ROT_SP Maximum feed for tool measurement for rotating Spindle		
Default setting: 20	Min. input limit: 1	Max. input limit: 1000	
Changes effective as from NOW	Protection level: 3/4	Units: mm/min	
Data type: DOUBLE	Valid as of software version: ShopTurn 6.4		
Significance:	This MD is used to stipulate the feed rate for tool measurement with rotating spindle.		

9772 MD number	CMM_T_PROBE_MEASURING_DIST Measurement distance for tool measurement with stationary spindle		
Default setting: 10	Min. input limit: 1	Max. input limit: 1000	
Changes effective as from NOW	Protection level: 3/4	Units: mm	
Data type: DOUBLE	Valid as of software version: ShopTurn 6.4		
Significance:	This MD is used to stipulate the measurement distance for tool measurement with stationary spindle and during probe calibration.		

9773 MD number	CMM_T_PROBE_MEASURING_FEED Feed rate for tool measurement with stationary spindle		
Default setting: 300	Min. input limit: 10	Max. input limit: 5000	
Changes effective as from NOW	Protection level: 3/4	Units: mm/min	
Data type: DOUBLE	Valid as of software version: ShopTurn 6.4		
Significance:	This MD is used to stipulate the feed for tool measurement with stationary spindle and during probe calibration.		

9828 MD number	ST_MEAS_T_PROBE_INPUT_SUB Input number for tool probe for counterspindle		
Default setting: 1	Min. input limit: 0	Max. input limit: 1	
Changes effective as from NOW	Protection level: 3/4	Units: –	
Data type: UNSIGNED WORD	Valid as of software version: ShopTurn 6.3		
Significance:	This MD is used to stipulate the number of the input for the tool probe for the counter–spindle. 0 = Measuring input 1 is activated 1 = Measuring input 2 is activated		

9.2 Network link

Option

The function ”Manage up to 4 additional network drives” is an option with Order No. 6FC5 800−0AP01−0YB0.

For installing the network, please refer to

References: /IHA/, Commissioning Base Software and HMI Advanced, SINUMERIK 840D sl/840D/840Di;
/IHE/, Commissioning Base Software and HMI Embedded sl, SINUMERIK 840D sl

9.3 Cylinder surface transformation, end face machining

9.3.1 General

The cylinder surface transformation and end face machining functions can be used to machine the peripheral and end faces of a turned part.

The functions ”cylinder surface transformation” (Tracyl) and ”face end machining” (Transmit) are a software option set on the CNC–ISO operator interface. The Order No. is: 6FC5 800–0AM27–0YB0.

These functions must also be set up via machine data.

- Face–end machining main spindle: Transformation 1
  MD 24100 $MC_TRAFO_TYPE_1=256 (without Y axis)
  MD 24100 $MC_TRAFO_TYPE_1=257 (with Y axis)
- Face–end machining counter–spindle: Transformation 2
  MD 24200 $MC_TRAFO_TYPE_2=256 (without Y axis)
  MD 24200 $MC_TRAFO_TYPE_2=257 (with Y axis)
- Cylinder surface transformation main spindle: Transformation 3
  without groove side offset (without Y axis):
  MD 24300 $MC_TRAFO_TYPE_3=512
  with groove side offset (with Y axis):
  MD 24300 $MC_TRAFO_TYPE_3=513
  with groove side offset and Y correction:
  MD 24300 $MC_TRAFO_TYPE_3=514
- Cylinder surface transformation counter–spindle: Transformation 4
  without groove side offset (without Y axis):
  MD 24400 $MC_TRAFO_TYPE_4=512
  with groove side offset (with Y axis):
  MD 24400 $MC_TRAFO_TYPE_4=513
  with groove side offset and Y correction:
  MD 24400 $MC_TRAFO_TYPE_4=514

Note

For each of the individual transformations you must also set up other machine data (see following sections).

The ”cylinder surface transformation” and ”end face machining” functions are automatically integrated in the ShopTurn cycles, with the exception of ”straight line” and ”circle”. For these two cycles you can call the functions in the Program operating area under Straight line, Arc, Tool.
References: /BATsl/, ShopTurn Operating/Programming Guide

9.3.2 Example: X axis and Z axis, main spindle and tool spindle

For example, for a turning machine with X and Z axes, main spindle (C1) and tool spindle (WZ) you must configure the following machine data:

2008020080 $MC_AXCONF_CHANAX_NAME_TAB[0]="XC"
Channel axis XC
20080 $MC_AXCONF_CHANAX_NAME_TAB[1]="ZC"
Channel axis ZC
20080 $MC_AXCONF_CHANAX_NAME_TAB[2]="C1"
Channel axis C1
20080 $MC_AXCONF_CHANAX_NAME_TAB[3]="WZ"
Channel axis WZ

General settings for transformations:
10602 $MN_FRAME_GEOAX_CHANGE_MODE=1
Take coordinate transformations in total frame into account when switching between geometry axes
24040 $MC_FRAME_ADAPT_MODE=H7
Adaptation of active frames
28082 $MC_MM_SYSTEM_FRAME_MASK, Bit 6=1
Configuring of channel–specific system frames included in channel calculation

Face end machining

Data set for face end machining (main spindle):
24100 $MC_TRAFO_TYPE_1=256
Definition of 1st transformation in channel:
TRANSMIT main spindle
24110 $MC_TRAFO_AXES_IN_1[0]=1
Channel axis perpendicular to rotary axis (XC) for 1st transformation
24110 $MC_TRAFO_AXES_IN_1[1]=3
Channel axis of rotary axis (C1) for 1st transformation
24110 $MC_TRAFO_AXES_IN_1[2]=2
Channel axis parallel to rotary axis (ZC) for 1st transformation
24120 $MC_TRAFO_GEOAX_ASSIGN_TAB_1[0]=1
1st channel axis (X) for 1st transformation
24120 $MC_TRAFO_GEOAX_ASSIGN_TAB_1[1]=3
2nd channel axis (Y) for 1st transformation
24120 $MC_TRAFO_GEOAX_ASSIGN_TAB_1[2]=2
3rd channel axis (Z) for 1st transformation
24900 $MC_TRANSMIT_ROT_AX_OFFSET_1=0
Offset of rotary axis for 1st TRANSMIT transformation
24905 $MC_TRANSMIT_ROT_AX_FRAME_1=2
Axial offset of rotary axis is taken into account during TRANSMIT
24910 $MC_TRANSMIT_ROT_SIGN_IS_PLUS_1=0
Arithmetic sign of rotary axis for 1st TRANSMIT transformation
24911 $MC_TRANSMIT_POLE_SIDE_FIX_1=1
Restriction of working area in front of/behind pole,
1st TRANSMIT transformation
24920 $MC_TRANSMIT_BASE_TOOL_1[0]=0
Vector of basic tool for 1st TRANSMIT transformation

Cylinder surface transformation without groove side offset

Data set for cylinder surface transformation without groove side offset (main spindle):

24300	$MC_TRAFO_TYPE_3=512 Definition of 3rd transformation in channel: TRACYL main spindle
24310	$MC_TRAFO_AXES_IN_3[0]=1 Channel axis perpendicular to rotary axis (XC) for 3rd transformation
24310	$MC_TRAFO_AXES_IN_3[1]=3 Channel axis of rotary axis (C1) for 3rd transformation
24310	$MC_TRAFO_AXES_IN_3[2]=2 Channel axis parallel to rotary axis (ZC) for 3rd transformation
24320	$MC_TRAFO_GEOAX_ASSIGN_TAB_3[0]=1 1st channel axis (X) for 3rd transformation
24320	$MC_TRAFO_GEOAX_ASSIGN_TAB_3[1]=3 2nd channel axis (Y) for 3rd transformation
24320	$MC_TRAFO_GEOAX_ASSIGN_TAB_3[2]=2 3rd channel axis (Z) for 3rd transformation
24800	$MC_TRACYL_ROT_AX_OFFSET_1=0 Offset of rotary axis for 1st TRACYL transformation
24805	$MC_TRACYL_ROT_AX_FRAME_1=2 Axial offset of rotary axis is taken into account during TRACYL
24810	$MC_TRACYL_ROT_SIGN_IS_PLUS_1=1 Arithmetic sign of rotary axis for 1st TRACYL transformation
24820	$MC_TRACYL_BASE_TOOL_1[n]=0 Vector of basic tool for 1st TRACYL transformation

9.3.3 Example: X axis and Z axis, main spindle and tool spindle, Y axis

For example, for a turning machine with X, Z, and Y axes, main spindle (C1) and tool spindle (WZ) you must configure the following machine data:

2008020080 $MC_AXCONF_CHANAX_NAME_TAB[0]="XC"
Channel axis XC
20080 $MC_AXCONF_CHANAX_NAME_TAB[1]="ZC"
Channel axis ZC
20080 $MC_AXCONF_CHANAX_NAME_TAB[2]="C1"
Channel axis C1
20080 $MC_AXCONF_CHANAX_NAME_TAB[3]="WZ"
Channel axis WZ
20080 $MC_AXCONF_CHANAX_NAME_TAB[5]="YC"
Channel axis YC

General settings for transformations:
10602 $MN_FRAME_GEOAX_CHANGE_MODE=1
Take coordinate transformations in total frame into account when switching between geometry axes
24040 $MC_FRAME_ADAPT_MODE=H7
Adaptation of active frames
28082 $MC_MM_SYSTEM_FRAME_MASK, Bit 6=1
Configuring of channel–specific system frames included in channel calculation

Face end machining

Data set for face end machining (main spindle):
24100 $MC_TRAFO_TYPE_1=257
Definition of 1st transformation in channel:
TRANSMIT main spindle
24110 $MC_TRAFO_AXES_IN_1[0]=1
Channel axis perpendicular to rotary axis (XC) for 1st transformation
24110 $MC_TRAFO_AXES_IN_1[1]=3
Channel axis of rotary axis (C1) for 1st transformation
24110 $MC_TRAFO_AXES_IN_1[2]=2
Channel axis parallel to rotary axis (ZC) for 1st transformation
24120 $MC_TRAFO_GEOAX_ASSIGN_TAB_1[0]=1
1st channel axis (X) for 1st transformation
24120 $MC_TRAFO_GEOAX_ASSIGN_TAB_1[1]=3
2nd channel axis (Y) for 1st transformation
24120 $MC_TRAFO_GEOAX_ASSIGN_TAB_1[2]=2
3rd channel axis (Z) for 1st transformation
24900 $MC_TRANSMIT_ROT_AX_OFFSET_1=0
Offset of rotary axis for 1st TRANSMIT transformation
24905 $MC_TRANSMIT_ROT_AX_FRAME_1=2
Axial offset of rotary axis is taken into account during TRANSMIT
24910 $MC_TRANSMIT_ROT_SIGN_IS_PLUS_1=0
Arithmetic sign of rotary axis for 1st TRANSMIT transformation
24911 $MC_TRANSMIT_POLE_SIDE_FIX_1=1
Restriction of working area in front of/behind pole,
1st TRANSMIT transformation
24920 $MC_TRANSMIT_BASE_TOOL_1[0]=0
Vector of basic tool for 1st TRANSMIT transformation

Cylinder surface transformation with groove side offset

Data set for cylinder surface transformation with groove side offset (main spindle):

24300	$MC_TRAFO_TYPE_3=513 Definition of 3rd transformation in channel: TRACYL main spindle
24310	$MC_TRAFO_AXES_IN_3[0]=1 Channel axis perpendicular to rotary axis (XC) for 3rd transformation
24310	$MC_TRAFO_AXES_IN_3[1]=3 Channel axis of rotary axis (C1) for 3rd transformation
24310	$MC_TRAFO_AXES_IN_3[2]=2 Channel axis parallel to rotary axis (ZC) for 3rd transformation
24310	$MC_TRAFO_AXES_IN_3[3]=6 Channel axis parallel to cylinder peripheral surface and perpendicular to rotary axis (ZC) for 3rd transformation
24320	$MC_TRAFO_GEOAX_ASSIGN_TAB_3[0]=1 1st channel axis (X) for 3rd transformation
24320	$MC_TRAFO_GEOAX_ASSIGN_TAB_3[1]=3 2nd channel axis (Y) for 3rd transformation
24320	$MC_TRAFO_GEOAX_ASSIGN_TAB_3[2]=2 3rd channel axis (Z) for 3rd transformation
24800	$MC_TRACYL_ROT_AX_OFFSET_1=0 Offset of rotary axis for 1st TRACYL transformation
24805	$MC_TRACYL_ROT_AX_FRAME_1=2 Axial offset of rotary axis is taken into account during TRACYL
24810	$MC_TRACYL_ROT_SIGN_IS_PLUS_1=1 Arithmetic sign of rotary axis for 1st TRACYL transformation
24820	$MC_TRACYL_BASE_TOOL_1[n]=0 Vector of basic tool for 1st TRACYL transformation

9.3.4 Example: X axis and Z axis, main spindle, tool spindle and counter–spindle

For example, for a turning machine with X and Z axes, main spindle (C1), tool spindle (WZ), and counter–spindle (C2) you must configure the following machine data:

2008020080 $MC_AXCONF_CHANAX_NAME_TAB[0]="XC"
Channel axis XC
20080 $MC_AXCONF_CHANAX_NAME_TAB[1]="ZC"
Channel axis ZC
20080 $MC_AXCONF_CHANAX_NAME_TAB[2]="C1"
Channel axis C1
20080 $MC_AXCONF_CHANAX_NAME_TAB[3]="WZ"
Channel axis WZ
20080 $MC_AXCONF_CHANAX_NAME_TAB[4]="C2"
Channel axis C2

General settings for transformations:
10602 $MN_FRAME_GEOAX_CHANGE_MODE=1
Take coordinate transformations in total frame into account when switching between geometry axes
24040 $MC_FRAME_ADAPT_MODE=H7
Adaptation of active frames
28082 $MC_MM_SYSTEM_FRAME_MASK, Bit 6 = 1
Configuring of channel–specific system frames included in channel calculation

Face end machining

Data set for face end machining (counter–spindle):
24200 $MC_TRAFO_TYPE_2=256
Definition of 2nd transformation in channel:
TRANSMIT counter–spindle
24210 $MC_TRAFO_AXES_IN_2[0]=1
Channel axis perpendicular to rotary axis (XC) for 2nd transformation
24210 $MC_TRAFO_AXES_IN_2[1]=5
Channel axis of rotary axis (C2) for 2nd transformation
24210 $MC_TRAFO_AXES_IN_2[2]=2
Channel axis parallel to rotary axis (ZC) for 2nd transformation
24220 $MC_TRAFO_GEOAX_ASSIGN_TAB_2[0]=1
1st channel axis (X) for 2nd transformation
24220 $MC_TRAFO_GEOAX_ASSIGN_TAB_2[1]=5
2nd channel axis (Y) for 2nd transformation
24220 $MC_TRAFO_GEOAX_ASSIGN_TAB_2[2]=2
3rd channel axis (Z) for 2nd transformation
24950 $MC_TRANSMIT_ROT_AX_offset_2=0
Offset of rotary axis for 2nd TRANSMIT transformation
24955 $MC_TRANSMIT_ROT_AX_FRAME_2=2
Axial offset of rotary axis is taken into account during TRANSMIT
24960 $MC_TRANSMIT_ROT_SIGN_IS_PLUS_2=0
Arithmetic sign of rotary axis for 2nd TRANSMIT transformation
24961 $MC_TRANSMIT_POLE_SIDE_FIX_2=1
Restriction of operating area in front of/behind the pole, 2nd TRANSMIT transformation
24970 $MC_TRANSMIT_BASE_TOOL_2[0]=0
Vector of basic tool for 2nd TRANSMIT transformation

Cylinder surface transformation without groove side offset

Data set for cylinder surface transformation on counter–spindle (without groove side offset):

24400	$MC_TRAFO_TYPE_4=512 Definition of 4th transformation in channel: TRACYL counter–spindle
24410	$MC_TRAFO_AXES_IN_4[0]=1 Channel axis perpendicular to rotary axis (XC) for 4th transformation
24410	$MC_TRAFO_AXES_IN_4[1]=5 Channel axis of rotary axis (C2) for 4th transformation
24410	$MC_TRAFO_AXES_IN_4[2]=2 Channel axis parallel to rotary axis (ZC) for 4th transformation
24420	$MC_TRAFO_GEOAX_ASSIGN_TAB_4[0]=1 1st channel axis (X) for 4th transformation
24420	$MC_TRAFO_GEOAX_ASSIGN_TAB_4[1]=5 2nd channel axis (Y) for 4th transformation
24420	$MC_TRAFO_GEOAX_ASSIGN_TAB_4[2]=2 3rd channel axis (Z) for 4th transformation
24850	$MC_TRACYL_ROT_AX_offset_2=0 Offset of rotary axis for 2nd TRACYL transformation
24855	$MC_TRACYL_ROT_AX_FRAME_2=2 Axial offset of rotary axis is taken into account during TRACYL
24860	$MC_TRACYL_ROT_SIGN_IS_PLUS_2=1 Arithmetic sign of rotary axis for 2nd TRACYL transformation
24870	$MC_TRACYL_BASE_TOOL_2[n]=0 Vector of basic tool for 2nd TRACYL transformation

9.3.5 Example: X axis and Z axis, main spindle, tool spindle, and counter–spindle, Y axis

For example, for a turning machine with X, Z, and Y axes, main spindle (C1), tool spindle (WZ), and counter–spindle (C2) you must configure the following machine data:

20080 $MC_AXCONF_CHANAX_NAME_TAB[0]=”XC”
Channel axis XC
20080 $MC_AXCONF_CHANAX_NAME_TAB[1]=”ZC”
Channel axis ZC
20080 $MC_AXCONF_CHANAX_NAME_TAB[2]=”C1”
Channel axis C1
20080 $MC_AXCONF_CHANAX_NAME_TAB[3]=”WZ”
Channel axis WZ
20080 $MC_AXCONF_CHANAX_NAME_TAB[4]=”C2”
Channel axis C2
20080 $MC_AXCONF_CHANAX_NAME_TAB[5]=”YC”
Channel axis YC

General settings for transformations:
10602 $MN_FRAME_GEOAX_CHANGE_MODE=1
Take coordinate transformations in total frame into account when switching between geometry axes
24040 $MC_FRAME_ADAPT_MODE=H7
Adaptation of active frames
28082 $MC_MM_SYSTEM_FRAME_MASK, Bit 6 = 1
Configuring of channel–specific system frames included in channel calculation

Face end machining

Data set for face end machining (counter–spindle):

24200	$MC_TRAFO_TYPE_2=257 Definition of 2nd transformation in channel: TRANSMIT counter–spindle
24210	$MC_TRAFO_AXES_IN_2[0]=1 Channel axis perpendicular to rotary axis (XC) for 2nd transformation
24210	$MC_TRAFO_AXES_IN_2[1]=5 Channel axis of rotary axis (C2) for 2nd transformation
24210	$MC_TRAFO_AXES_IN_2[2]=2 Channel axis parallel to rotary axis (ZC) for 2nd transformation
24220	$MC_TRAFO_GEOAX_ASSIGN_TAB_2[0]=1 1st channel axis (X) for 2nd transformation
24220	$MC_TRAFO_GEOAX_ASSIGN_TAB_2[1]=5 2nd channel axis (Y) for 2nd transformation
24220	$MC_TRAFO_GEOAX_ASSIGN_TAB_2[2]=2 3rd channel axis (Z) for 2nd transformation
24950	$MC_TRANSMIT_ROT_AX_offset_2=0 Offset of rotary axis for 2nd TRANSMIT transformation
24955	$MC_TRANSMIT_ROT_AX_FRAME_2=2 Axial offset of rotary axis is taken into account during TRANSMIT
24960	$MC_TRANSMIT_ROT_SIGN_IS_PLUS_2=0 Arithmetic sign of rotary axis for 2nd TRANSMIT transformation
24961	$MC_TRANSMIT_POLE_SIDE_FIX_2=1 Restriction of operating area in front of/behind the pole, 2nd TRANSMIT transformation
24970	$MC_TRANSMIT_BASE_TOOL_2[0]=0 Vector of basic tool for 2nd TRANSMIT transformation

Cylinder surface transformation with groove side offset

Data set for cylinder surface transformation with groove side offset (counter–spindle):

24400	$MC_TRAFO_TYPE_4=513 Definition of 4th transformation in channel: TRACYL counterspindle
24410	$MC_TRAFO_AXES_IN_4[0]=1 Channel axis perpendicular to rotary axis (XC) for 4th transformation
24410	$MC_TRAFO_AXES_IN_4[1]=5 Channel axis of rotary axis (C2) for 4th transformation
24410	$MC_TRAFO_AXES_IN_4[2]=2 Channel axis parallel to rotary axis (ZC) for 4th transformation
24410	$MC_TRAFO_AXES_IN_4[3]=6 Channel axis parallel to cylinder peripheral surface and perpendicular to rotary axis (ZC) for 4th transformation
24420	$MC_TRAFO_GEOAX_ASSIGN_TAB_4[0]=1 1st channel axis (X) for 4th transformation
24420	$MC_TRAFO_GEOAX_ASSIGN_TAB_4[1]=5 2nd channel axis (Y) for 4th transformation
24420	$MC_TRAFO_GEOAX_ASSIGN_TAB_4[2]=2 3rd channel axis (Z) for 4th transformation
24850	$MC_TRACYL_ROT_AX_OFFSET_2=0 Offset of rotary axis for 2nd TRACYL transformation
24855	$MC_TRACYL_ROT_AX_FRAME_2=2 Axial offset of rotary axis is taken into account when TRACYL is active
24860	$MC_TRACYL_ROT_SIGN_IS_PLUS_2=1 Sign of rotary axis for 2nd TRACYL transformation
24870	$MC_TRACYL_BASE_TOOL_2[n]=0 Vector of basic tool for 2nd TRACYL transformation

9.4 Inclined Y axis

If your machine has an inclined Y axis (i.e. the Y axis is not perpendicular to the other axes), you can still program the entire machining sequence in the Cartesian coordinate system. The control uses the ”Inclined axis” function (Traang) to transform the Cartesian coordinates to the traveling motions of the inclined axis.

The function ”Inclined Axis” (Traang) is a software option which you need to set in the CNC–ISO user interface. The Order No. is:
6FC5 800–0AM28–0YB0.

You also need to set up the ”Inclined axis” function (Traang) via machine data.
References: /FB2/, Function Manual on Extended Functions,
M1, Kinematic Transformations

Once the ”Inclined axis” function has been set up in the ShopTurn user interface, it is automatically integrated in the ShopTurn cycles. This means you can select machining level ”Face Y” or ”Surface Y” in the masks and enter the position for the inclined Y axis in Cartesian coordinates.
References: /BATsl/, ShopTurn Operating/Programming Guide

Example

For example, for a turning machine with X and Z axes and inclined Y axis, main spindle (C), and tool spindle (WZ) you must configure the following machine data:

20050	$MC_AXCONF_GEOAX_ASSIGN_TAB[0]=1 1st real geometric axis (X axis)
20050	$MC_AXCONF_GEOAX_ASSIGN_TAB[1]=0 2nd real geometric axis (Y axis) not configured
20050	$MC_AXCONF_GEOAX_ASSIGN_TAB[2]=2 3rd real geometric axis (Z axis)
20110	$MC_RESET_MODE_MASK, Bit 0 = 1, Bit 7 = 0 TRAANG retained after booting
20112	$MC_START_MODE_MASK, Bit 7 = 1 TRAANG retained after ”Cycle start”
20118	$MC_GEOAX_CHANGE_RESET=1 Allow automatic switching of geometric axes
20140	$MC_TRAFO_RESET_VALUE=5 TRAANG is always active after reset
20144	$MC_TRAFO_MODE_MASK, bit 0 = 1 TRAANG runs in background (persistent process) and is not displayed in the operator interface

20070 $MC_AXCONF_MACHAX_USED[4]=5
Channel axis YC = 5th machine axis
20080 $MC_AXCONF_CHANAX_NAME_TAB[0]=”XC”
1st channel axis = XC
20080 $MC_AXCONF_CHANAX_NAME_TAB[1]=”ZC”
2nd channel axis = ZC
20080 $MC_AXCONF_CHANAX_NAME_TAB[2]=”C”
3rd channel axis = C
20080 $MC_AXCONF_CHANAX_NAME_TAB[3]=”WZ”
4th channel axis = WZ
20080 $MC_AXCONF_CHANAX_NAME_TAB[4]=”YC”
5th channel axis = YC

Data set for ”Inclined axis”:
24430 $MC_TRAFO_TYPE_5=1024
Transformation 5: TRAANG
24432 $MC_TRAFO_AXES_IN_5[0]=5
1st transformation axis with transformation 5 = channel axis YC
24432 $MC_TRAFO_AXES_IN_5[1]=1
2nd transformation axis with transformation 5 = channel axis XC
24432 $MC_TRAFO_AXES_IN_5[2]=2
3rd transformation axis with transformation 5 = channel axis ZC
24434 $MC_TRAFO_GEOAX_ASSIGN_TAB_5[0]=1
1st geometric axis with transformation 5 = channel axis XC
24434 $MC_TRAFO_GEOAX_ASSIGN_TAB_5[1]=5
2nd geometric axis with transformation 5 = channel axis YC
24434 $MC_TRAFO_GEOAX_ASSIGN_TAB_5[2]=2
3rd geometric axis with transformation 5 = channel axis ZC
24436 $MC_TRAFO_INCLUDES_TOOL_5=0
Tool included when 5th transformation is active
24700 $MC_TRAANG_ANGLE_1=55
Angle between 1st and 2nd transformation axes

Data set for linking (TRACON) face end machining on main spindle (TRANSMIT) and ”Inclined axis” (TRAANG):
24440 $MC_TRAFO_TYPE_6=8192
Transformation 6: TRACON
24444 $MC_TRAFO_GEOAX_ASSIGN_TAB_6[0]=1
1st geometric axis with transformation 6 = channel axis XC
24444 $MC_TRAFO_GEOAX_ASSIGN_TAB_6[1]=3
2nd geometric axis with transformation 6 = channel axis C
24444 $MC_TRAFO_GEOAX_ASSIGN_TAB_6[2]=2
3rd geometric axis with transformation 6 = channel axis ZC
24995 $MC_TRACON_CHAIN_1[0]=1
Number of transformation TRANSMIT (main spindle) for linking
24995 $MC_TRACON_CHAIN_1[1]=5
Number of transformation TRAANG for linking

Data set for linking (TRACON) from cylinder surface transformation main spindle and ”Inclined axis” (TRAANG):

24450	$MC_TRAFO_TYPE_7=8192 Transformation 7: TRACON
24454	$MC_TRAFO_GEOAX_ASSIGN_TAB_7[0]=1 1st geometric axis with transformation 7 = channel axis XC
24454	$MC_TRAFO_GEOAX_ASSIGN_TAB_7[1]=3 2nd geometric axis with transformation 7 = channel axis C
24454	$MC_TRAFO_GEOAX_ASSIGN_TAB_7[2]=2 3rd geometric axis with transformation 7 = channel axis ZC
24996	$MC_TRACON_CHAIN_1[0]=1 Number of transformation TRACYL (main spindle) for linking
24996	$MC_TRACON_CHAIN_1[3]=5 Number of transformation TRAANG for linking

9.5 Measuring cycle support in the G code editor

In ShopTurn you can integrate screen forms which provide support for programming measuring cycles into the G code editor. These cycles can then be recompiled automatically.

The ”measuring cycles” function is a software option with Order No. 6FC5 800–0AP28–0YB0.

For more information on measuring cycle support, please refer to:
References: /BNM/, Programming Manual on Measuring Cycles

For commissioning, proceed as follows:

ShopTurn on NCU (HMI Embedded sl)

- Remove the semicolon ”;” preceding the following lines in file STANDARD_CYCLES\COMMON.COM:

```
;sc8406=aeditor.com     ;Measuring cycles for turning (horizontal softkey 6 on
                        ;the expanded softkey menu in the
                        ;G code editor)
;sc8407=aeditor.com     ;Measuring cycles for milling (horizontal softkey 7
                        ;on the expanded softkey menu in the
                        ;G code editor)
```

  By doing this, you create the connection between the softkeys used to call the measuring cycles support and the configuration file of the support screens.

- Remove the semicolon ”;” preceding the following line in file STANDARD_CYCLES\COMMON.COM:

```
;sc617=startup.com      ;Startup operating area (horizontal
                        ;softkey 7 on the expanded softkey bar)
```

  You can modify the characteristics of the measuring cycle support in the startup operating area.

- Restart the machine.

- If required, modify the characteristics for the measuring cycle support in the ”Commissioning” → ”>” → ”Measuring cycles” menu.

PCU 50.3

Requirements:
When HMI Advanced was commissioned, the files AEDITOR.COM and STARTUP.COM must have been stored in the STANDARD CYCLES (CST.DIR) directory.
when ShopTurn started up the archive ST_CYC.ARC must have been transferred from the PCU 50.3 to the NCK. The COMMON.COM file must have been copied automatically to the STANDARD_CYCLES (CST.DIR) directory.

- Copy the archive MCSUPP from the ARCHIVES\CYCLE_ARCHIVES\MCYC.
- If you want to use the measurement result displays, you need to copy the MCRESULT archive from the ARCHIVES\CYCLE_ARCHIVES\MCYC directory.
- If the versions of the standard measuring cycles supplied with ShopTurn (see file SIEMENSD.RTF or SIEMENSE.RTF on the software CD) differ from the HMI Advanced measuring cycles (see ARCHIVES\CYCLE_ARCHIVES\MCYC\VERSION.ARC) in the first 4 digits (e.g. 06.02), you need to copy the archive MCYCTURN from the ARCHIVES\CYCLE_ARCHIVES\MCYC directory.
- Remove the semicolon ";" preceding the following lines in file STANDARD_CYCLES\COMMON.COM:

```
;sc8406=aeditor.com    ;Measuring cycles for turning (horizontal softkey 6 on
                       ;the expanded softkey menu in the
                       ;G code editor)
;sc8407=aeditor.com;   ;Measuring cycles for milling (horizontal softkey 7
                       ;on the expanded softkey menu in the
                       ;G code editor)
```

  Remove the semicolon ";" preceding the following lines in the file STANDARD_CYCLES\AEDITOR.COM:

```
;HS15=($83531,,se1)
;PRESS(HS15)
;  LS("F_mess","MZ_SKL.COM",1)
;END_PRESS
```

  By doing this, you create the connection between the softkeys used to call the measuring cycles support and the configuration file of the support screens.

- In the file STANDARD_CYCLES\STARTUP.COM remove the semicolon ";" preceding the following lines:

```
;HS15=($83070,,se1)
;PRESS(HS15)
; LS("Messz")
;END_PRESS
```

  This allows you to modify the characteristics of the measuring cycle support in the Startup operating area.
- Restart the PCU 50.3.
- If required, modify the characteristics for the measuring cycle support in the "Commissioning" → ">" → "Measuring cycles" menu.

9.6 Counter–spindle

If your turning machine has a counter–spindle, you can machine workpieces with turning, drilling, and milling functions on the front and reverse side without having to rechuck the workpiece manually.

Before machining the reverse side, the counter–spindle must grip the workpiece, pull it out of the main spindle, and position it at the new machining position.

In order to use the counter–spindle on your machine,
you need the software options ”travel to fixed stop” (Order No. 6FC5 800–0AM01–0YB0) and ”synchronous spindle” (Order No. 6FC5 800–0AM14–0YB0).

layout

When setting up the counter–spindle you need to take the following machine data into account:
9803 $MM_ST_INDEX_AXIS_4: Axis index slide counter–spindle
9806 $MM_ST_INDEX_SPINDLE_SUB: Axis index counter–spindle
9812 $MM_ST_GEAR_STEPS_SPINDLE_SUB: Gear steps counter–spindle
9823 $MM_ST_DISPL_DIR_SUB_SPIND_M3: Direction of rotation with M3
9825 $MM_ST_DISPL_DIR_SUB_C_AX_INV: Direction of rotation C axis with M3

The position to which the counter–spindle travels when the program starts is defined in the following machine data:
9851 $MM_ST_CYCLE_SUB_SP_WORK_POS: Retraction position of the counter–spindle

If the counter–spindle is to travel to fixed stop when gripping, ShopTurn will evaluate the following machine data:
9852 $MM_ST_CYCLE_SUB_SP_DIST: Travel to fixed stop, distance
9853 $MM_ST_CYCLE_SUB_SP_FEED: Travel to fixed stop, feed
9854 $MM_ST_CYCLE_SUB_SP_FORCE: Travel to fixed stop, force

Between traveling to the fixed stop and gripping, the counter–spindle can retract a short distance to counteract compressive stress in the workpiece.
9857 $MM_ST_CYCLE_RET_DIST_FIXEDSTOP: Retraction distance for relieving compressive stress after traveling to fixed stop

After gripping you can cut off the workpiece. Before doing so, the counter–spindle can travel back a short distance with the workpiece to exert tensile stress on the workpiece. This relieves pressure on the tool when cutting off.
9858 $MM_ST_CYCLE_RET_DIST_PART_OFF: Retraction distance for tensile stress before cut–off

After cut–off, you can carry out a cut–off check with ShopTurn using the function ”Travel to fixed stop”. You can activate/deactivate the cut–off check by means of the following machine data:
9859 $MM_ST_CYCLE_PART_OFF_CTRL_DIST: Cut–off check, distance
9860 $MM_ST_CYCLE_PART_OFF_CTRL_FEED: Cut–off check, feed
9861 $MM_ST_CYCLE_PART_OFF_CTRL_FORC: Cut–off check, force

The cut–off is successful if travel to fixed stop fails. Alarms 20091 ”Axis %1 did not reach fixed stop” and 20094 ”Axis %1 function was aborted” must therefore be deactivated in the machine data:
37050 $MA_FIXED_STOP_ALARM_MASK = 2
This machine data must be set in the ”Machine data” mask in operating area ”Tools – zero offset”; you need to adapt it to the specific axis.
If, however, the specified force was achieved during the cut–off control (i.e. travel to fixed stop was successful), alarm 61255 ”Error at cut–off: Tool break?” is issued.

Note

The ”Travel to fixed stop” function can also be used when gripping the spindle (see above). If travel to fixed stop does not succeed when gripping, an alarm will of course also be issued. Instead of alarms 20091 and 20094 the alarm 61254 ”Error during travel to fixed stop” will be issued.

Dimensions

To define the reference point for moving the counter–spindle, you must first inform the counter–spindle of the dimensions. You can either enter the dimensions in the following machine data or in the menu ”Tools – zero offset” → ”>” → ”Spindles”. Changes to the machine data automatically become effective in the menu and vice versa.
9829 $MM_ST_SPINDLE_CHUCK_TYPES: Spindle chuck selection
9831 $MM_ST_SPINDLE_PARA_ZL1: Chuck dimensions for counter–spindle
9832 $MM_ST_SPINDLE_PARA_ZL2: Stop dimension for counter–spindle
9833 $MM_ST_SPINDLE_PARA_ZL3: Jaw dimension for counter–spindle

Machine manufacturer cycle

If you would like to perform one of the following actions, you need to adapt the machine manufacturer cycle ST_CUST.SPF (see section 11.2 ”Machine manufacturer cycles):

- Switch between spindle and C axis mode for the main spindle or counter–spindle
- Open, close, flush chuck (main spindle/counter–spindle)
- Change default settings for the main spindle and counter–spindle

Simulation

If your programming includes M functions to open or close the chuck on the main spindle or counter–spindle, the simulation will only display these actions if you observe the following:

In the machine data you must assign the M functions to cycles which will then perform the appropriate function.
The functions performed by individual cycles are listed in the table below:

Table 9-1 Cycles for opening/closing the chuck

Cycle	Function
chuck1cl.spf	Main spindle, close chuck
chuck1os.spf	Main spindle, open chuck when spindle is stationary
chuck1or.spf	Main spindle, open chuck when spindle is rotating
chuck2cl.spf	Counter–spindle, close chuck
chuck2os.spf	Counter–spindle, open chuck when spindle is stationary
chuck2or.spf	Counter–spindle, open chuck when spindle is rotating

Please proceed as follows:

- Transfer the necessary cycles from the directory CYCLES\SC\PROG\TEMPLATES to the user cycles directory.
- Assign the M functions to cycles in the following machine data:
  MD 10715 $MN_M_NO_FCT_CYCLE[n] = Number of M function
  MD 10716 $MN_M_NO_FCT_CYCLE_NAME[n] = Name of cycle
- Execute a ”Power ON” in the NCK.

Simulation example

Sample program:

```
G54
...                                  ; Machine front face
G0 G90 X200                          ; Retract tool
M280                                 ; Open chuck on counter–spindle
COUPDEF (S3,S4,1,1,"noc","dv")       ; Define coupling
COUPON (S3,S4,1)                     ; Coupling ON
G0 Z3=300                            ; Position counter–spindle
G1 G91 F1000 Z3=–40                  ; Gripper position
M281                                 ; Close chuck on counter–spindle
M240                                 ; Open chuck on main spindle
G1 G91 F1000 Z3=60                   ; Pull
G0 G90 Z3=600                        ; Machining position
COUPOF (S3,S4)                       ; Coupling OFF
G55                                  ; Activate rear face WO with
                                     ; mirroring
...                                  ; Machine rear face
M30                                  ; End of program
```

To obtain the sample program above, set the following machine data:
$MN_M_NO_FCT_CYCLE[1]=**M280**
$MN_M_NO_FCT_CYCLE_NAME[1]=chuck2os
$MN_M_NO_FCT_CYCLE[2]=**M281**
$MN_M_NO_FCT_CYCLE_NAME[2]=chuck2cl
$MN_M_NO_FCT_CYCLE[3]=**M240**
$MN_M_NO_FCT_CYCLE_NAME[3]=chuck1os

For further information about machining with the counter–spindle, please refer to:
References: /BATsl/, ShopTurn Operating/Programming Guide

9.7 Turning machines with a B axis

Where turning machines with an additional rotary axis B are concerned, the tool can be aligned or swiveled on the X/Y plane. If your machine has a counterspindle, you can alternate machining on the main spindle and counterspindle with, e.g., a tool.
The basic setting, in which all tools are measured, must be B = 0.

Setup

When setting up your machine, you must take the following machine data into account:
MD 9808 $MM_ST_INDEX_AXIS_B

The function CUTMOD is used to create the necessary tool offsets.

References: PG/, "Programming Manual Fundamentals"

The Swivel function is activated using MD 9723 $MM_CMM_ENABLE_SWIVELLING_HEAD = 1.

For aligning and swiveling purposes, the following swivel data records must be taken into account:

- Swivel data record for aligning and swiveling tools on the main spindle and counterspindle (Turning, Face Y/C, Surface Y/C).
- Swivel data record for machining inclined surfaces when milling on the main spindle (face B).
- Swivel data record for machining inclined surfaces when milling on the counterspindle (face B).

When commissioning swiveling, you must select the "Yes" setting in the "B axis kinematics" field in the "Commissioning Rotary axis parameters" menu.

References: PGZ/, Programming Guide Cycles, chapter "Swiveling – Cycle800

Measuring a tool

The tool is measured with special brackets for β and γ, which can be set in the input mask. The rotary axes can be put in the requisite positions through the NC start.
Before setting the length, these positions are compared with the specified values. If the values differ from one another by more than the value of the MD 36000 STOP_LIMIT_COARSE (coarse precise stop), the message "Tool orientation Beta contradictory, actuate NC start" (or gamma) is output.
The tolerance limit is approved because there may be slight deviations from the target value, for example as a result of an axis jamming.

Transformations

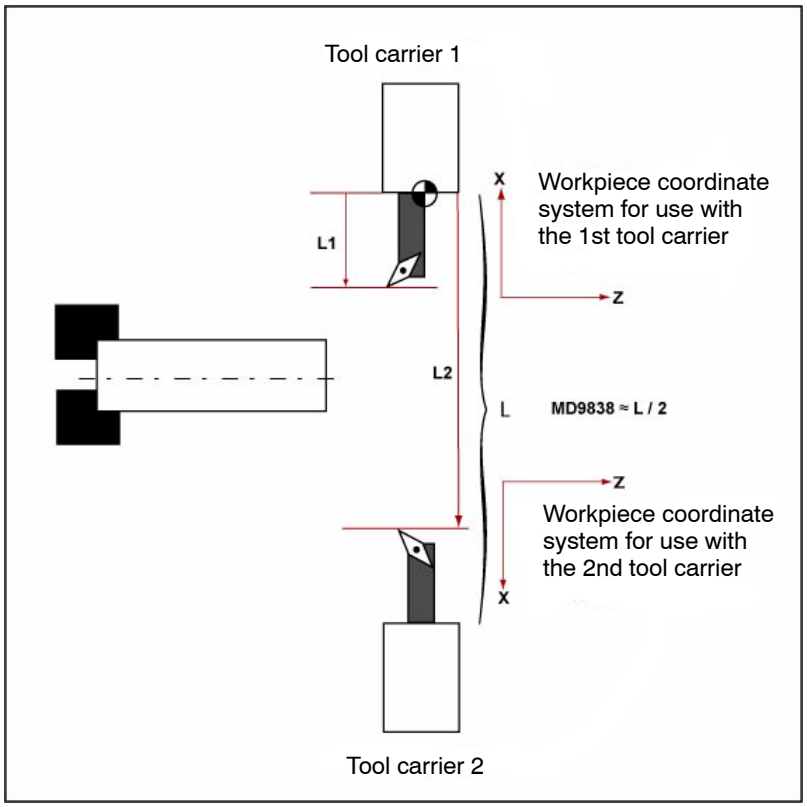
To use the function "Tracking tool (hold tool tip when swiveling)", you must commission the relevant TRAORI.

9.8 Working with two tool carriers

It is possible to work on a single turning machine with two tool carriers, both of which are mounted on an X axis. The tool holders might be turrets, multifix, or a combination of both.

Since both tool adapters are mounted on the same axis, it is only ever possible to work with one tool at a time. The workpiece is located between the two tool adapters.

All tools have the same reference point (generally on tool carrier 1), regardless of which tool carrier they are located on. For this reason, the tool lengths of the second tool carrier are always longer than those of the tools on the first tool carrier. The main machining work is carried out in the negative range of the X axis.



A tool length limit is defined using the MD 9838 $MM_ST_BORDER_TOOL_LEN_X_REV_2. Tools can be automatically assigned to the two tool carriers by means of this:
Tool length< MD 9838: Tool carrier1
Tool length ≥ MD 9838: Tool carrier2

If the tool length is ≥ MD 9838, mirroring takes place.

For the mirroring used for the X and Y axes and offsets of the C axis, the following machine data must be configured:

MD 24006 $MC_CHSFRAME_RESET_MASK='H11'; Bit 4 = 1
MD 24007 $MC_CHSFRAME_RESET_CLEAR_MASK='H00' ; Bit 4 = 0
MD 28082 $MC_MM_SYSTEM_FRAME_MASK='H31'; Bit 4 = 1
MD 28083 $MC_MM_SYSTEM_DATAFRAME_MASK='H1F'; Bit 4 = 1

Due to the coordinate system being mirrored, the following machine/setting data must be set:

MD 21202 $MC_LIFTFAST_WITH_MIRROR=1
Mirroring with LIFTFAST

SD 42910 $SC_MIRROR_TOOL_WEAR=1
Mirroring of tool wear

MD 9478 Bit 19 = 1; if no turret is being used.

G code programs

Note

If there are a large number of old G code programs that were programmed without mirroring of the X axis, you should consider setting mirroring for G code programs.

If the relevant mode of operation of the second tool carrier (with mirroring of the X axis) is also to be used for G code programs, further settings must be made.

To activate mirroring when loading tools on the second tool carrier, the following T replacement must be set:

10717 $MN_T_NO_FCT_CYCLE_NAME=”F_T_REV2”

If the necessary mirror images are also to be activated automatically when programming the TRASMIT command, this command must be replaced by a subprogram (e.g., in the manufacturer directory):

10712 $MN_NC_USER_CODE_CONF_NAME_TAB[0]=”TRANSMIT”
10712 $MN_NC_USER_CODE_CONF_NAME_TAB[1]=”TRANSM”

TRANSMIT.SPF
PROC TRANSMIT(INT_NR)
F_T_REV2
TRANSM(_NR)
F_T_REV2
RET

Note

In the status display, TRANSM is now displayed instead of TRANSMIT.

Supplementary condition

The ”Taper turning” option is not enabled when working with two tool carriers. It is not possible to use the turning machine with B axis at the same time.

■

Space for your notes

Customized User Interface 10

10.1 Configuring the custom boot screen

With the ShopTurn variant with PCU 50.3 you can create your own customized image (e.g. company logo, etc.) which will then appear when the control boots up.
To do so create your own image in 16–color mode. The maximum size of the image, for the OP010/OP010C/OP010S/OP012, is 224 x 224 pixels and, for the OP015, 352 x 352 pixels.

For the procedure for configuring this and exchanging the boot–up image, please refer to:

References: /IHA/, Commissioning Base Software and HMI Advanced, SINUMERIK 840D sl/840Di/840D

10.2 Configuring the custom screen

You can use the ”operator interface extension” software to build your own custom screens with specific functional expansions (e.g. custom cycles and measuring cycles) or just create your own screen layout (static screens).

References: /IHA/, Commissioning Base Software and HMI Advanced, Operator Interface Extension, SINUMERIK 840D sl/840Di/840D

Screen form properties

In ShopTurn you cannot specify any dimensions for configuring the main part of the screen. The position and size of the various main parts of the screen are fixed according to the operating area and mode. This only applies for screens that can be accessed via entry softkeys.

Table 10-1 Fixed dimensions of ShopTurn screens

Exit screen	Softkey	Size	Dimensions
Manual operating mode – Zero point workpiece/Measure workpiece	Vertical softkey 1 ... Vertical softkey 7		X–pos. = 0 Y–pos. = 272 height = 158 pixels width = 556 pixels
Manual operating mode	Horizontal softkey 8	Large	X–pos. = 0 Y–pos. = 34 height = 396 pixels width = 556 pixels
Manual operating mode	Horizontal softkey 8	Medium	X–pos. = 0 Y–pos. = 78 height = 352 pixels width = 556 pixels
Manual operating mode	Horizontal softkey 8	Small	X–pos. = 0 Y–pos. = 254 height = 176 pixels width = 556 pixels
Automatic mode	Horizontal softkey 6	Large	X–pos. = 0 Y–pos. = 34 height = 396 pixels width = 556 pixels
Automatic mode	Horizontal softkey 6	Medium	X–pos. = 0 Y–pos. = 78 height = 352 pixels width = 556 pixels
Automatic mode	Horizontal softkey 6	Small	X–pos. = 0 Y–pos. = 254 height = 176 pixels width = 556 pixels
Program operating area – Drilling/milling	Vertical softkey 6		X–pos. = 33 Y–pos. = 52 height = 378 pixels width = 523 pixels
Program operating area – Miscellaneous	Vertical softkey 4		X–pos. = 33 Y–pos. = 52 height = 378 pixels width = 523 pixels

Table 10-1 Fixed dimensions of ShopTurn screens

Exit screen	Softkey	Size	Dimensions
Program operating area – G code editor	Horizontal softkey 2 ... Horizontal softkey 6		X–pos. = 0 Y–pos. = 34 height = 396 pixels width = 556 pixels
Alarms operating area	Horizontal softkey 7 Horizontal softkey 8		X–pos. = 0 Y–pos. = 34 height = 396 pixels width = 556 pixels
Operating area – Tools/ Zero offsets	Horizontal softkey 7		X–pos. = 0 Y–pos. = 34 height = 396 pixels width = 556 pixels

Generate code

Do not enter a target directory for the program operating area and the manual mode. ShopTurn defines the target file.

Entry softkeys

The customized static screens can be displayed with the following entry softkeys, the SC number being the link between the entry softkey and the configuration file of the screen.
In manual and automatic modes you have the choice between different sized screens that fill the following window areas:

- Large: Status area, position display, or program display and input window
- Medium: Position display or program display and input window
- Small: Input window

Table 10-2 Entry softkeys for screens

Exit screen	Softkey	Size	SCxxxx
Manual operating mode	Horizontal softkey 8	Large	SC818
	Horizontal softkey 8	Medium	SC8181
	Horizontal softkey 8	Small	SC8182
Automatic mode	Horizontal softkey 6	Small	SC826
	Horizontal softkey 6	Medium	SC8261
	Horizontal softkey 6	Large	SC8262
Messages/Alarms operating area	Horizontal softkey 7		SC857
Messages/Alarms operating area	Horizontal softkey 8		SC858
Operating area – Tools/ Zero offsets	Horizontal softkey 7		SC867

Note

The screen size is fixed in ShopTurn (see above ”Screen Properties”). Take note of this in case you would like to program background images, for example, for screens you configured yourself.

Note

Open a user–configured screen before calling a softkey bar.

You can call your own cycles by means of the following access softkeys:

Table 10-3 Entry softkeys for cycles

Exit screen	Softkey	SCxxxx
Program operating area – Drilling	Vertical softkey 6	SC8426
Program operating area – Turning	Vertical softkey 6	SC9436
Program operating area – Milling	Vertical softkey 6	SC9456
Program operating area – Miscellaneous	Vertical softkey 4	SC8454

Note

In manual mode you can also assign your own user cycle to horizontal softkey 8. The NC code generated is then collected and you can then start the cycle with ”Cycle Start”; (see section 10.2.3. ”Integrating measuring cycles).

You can use the following entry softkeys to display your own measuring cycles, which are to replace the ShopTurn measuring cycles:

Table 10-4 Entry softkeys for measuring cycles

Exit screen	Softkey	SCxxxx
Manual operating mode – Zero point workpiece	Vertical softkey 1	SC8131
Manual operating mode – Zero point workpiece	Vertical softkey 2	SC8132
Manual operating mode – Zero point workpiece	Vertical softkey 3	SC8133
Manual operating mode – Zero point workpiece	Vertical softkey 4	SC8134
Manual operating mode – Zero point workpiece	Vertical softkey 5	SC8135
Manual operating mode – Zero point workpiece	Vertical softkey 6	SC8136
Manual operating mode – Zero point workpiece	Vertical softkey 7	SC8137
Manual operating mode – Tool measurement	Vertical softkey 1	SC8141
Manual operating mode – Tool measurement	Vertical softkey 2	SC8142
Manual operating mode – Tool measurement	Vertical softkey 3	SC8143
Manual operating mode – Tool measurement	Vertical softkey 4	SC8144
Manual operating mode – Tool measurement	Vertical softkey 5	SC8145
Manual operating mode – Tool measurement	Vertical softkey 6	SC8146
Manual operating mode – Tool measurement	Vertical softkey 7	SC8147

10.2.1 Transferring cycles to the machining plan

You can transfer all user cycles that you have inserted in the Program operating area (including measuring cycles) to the work plan and edit them there. However, the cycles will not be displayed as programming graphics.

Use the ”operator interface extension” software to define the cycle. Please note that the cycle must contain the Generate Code (GC) function and an OUTPUT method for NC code to be generated and sent to the NC for execution.

If you call and validate the cycle on the ShopTurn user interface, the cycle will automatically appear in the work plan. The screen form name of the cycle will appear in the machining plan as plain text.

Sample cycles

Sample cycles are available for the following applications:

- Tail stock (TAILSTOK)
- Part gripper (RECEPTAC)
- Bar loader (BARLOAD)

If you want to use these cycles, adapt them to the characteristics of your machine.

Directory CYCLES\SC\PROG\TEMPLATES contains the archive file EQUIPMENT.ARC which lists the cycles and operator interfaces. The associated texts are imported together with the alarm texts when you install ShopTurn.
A readme file with information about how to install is also located in the same directory as the archive file.

The cycles for tail stock, part gripper, and bar loader can be seen in the Program operating area under Miscellaneous —> Device. You will also find the tail stock cycle in Manual mode (horizontal softkey 8).

Sample cycles – linking ”bar loader”

The ”bar loader” cycle can be linked with a cut–off or stock removal cycle; (see Sample cycle).

Note

If you want to use the sample cycle for bar loader, you must not change its name (”BARLOAD”).

10.2.2 Linking cycles into the machining plan

You can link user cycles into the work plan with the ShopTurn cycles ”Positions”.

Name

The name of the cycles to be linked in the work plan is mandatory and must not be changed:
F_DR_Ox with x = 1 to 8

Variables

The following variables must be configured within the cycles:

First you must define three variables that are not used until the program step is inserted in the ShopTurn work plan; i.e. you must assign the attribute invisible (wr0) to the variable input field. The variable sequence is as follows:

1. Variable for the next tool, data type String (S)
2. Variable for the position identifier in objects, data type Integer
3. Variable for the sequencer text that is to be displayed in the ShopTurn work schedule, data type character string (S)
   The easiest way to assign a value to this variable is in a LOAD method.

Two variables for cycles processing must also be defined as well as one variable for each toggle field which has a different display content on the interface (e.g. mm/rev) than the internally transferred quantity (e.g. 1).
You need to assign the attribute invisible (wr0) to these variables too as they do not have their own input field. You can place the variables in any order.

Note

When an OUTPUT method is used, all variables described above must be configured within this method so that they can all be interpreted on recompilation.

Once–only machining

If a particular work step is to be performed not in all positions but only once, the following user variable must be set in the associated cycle F_ONETIME = 1.
This also provides the possibility of implementing modal changes for subsequent technologies.

Example

Directory CYCLES\SC\PROG\TEMPLATES contains a sample cycle F_DR_O1.SPF. The associated user interface is configured in file F_DR_O1.COM. The German and English texts are stored in files F_DR_TXD.COM and F_DR_TXE.COM. A number of help displays are also included with the sample cycle.
Please see the README.TXT file for information on how to install the cycle.

10.2.3 Integrating measuring cycles

Manual operating mode

If you wish to integrate your own measuring cycles in Manual mode, vertical softkey 8 (VS8) in the new cycles screen acquires a special meaning. All actions that the new measuring cycle is supposed to perform after the ”cycle start” key has been pressed must be defined in a PRESS method for VS8. If you press the ”cycle start” button, NC code is generated that is written and executed in a program, i.e. the function Generate Code (GC) must be programmed in the PRESS method for VS8. This in turn means that an OUTPUT method must be defined.

Note

The user does not have to press VS8; it is triggered internally when the ”cycle start” key is pressed. The VS8 must therefore not be labeled.

Example

Directory CYCLES\SC\PROG\TEMPLATES contains the archive file F_MS_O1.ARC containing sample cycle F_MS_O1.SPF. The associated user interface is configured in file F_MS_O1.COM. In Files F_MS_TXD.COM and F_MS_TXE.COM contain the German and English texts, and file EDGE_Z.BMP contains a help display.
Please see the README.TXT file for information on how to install the cycle.

”Program” operating area

If you want to link user measuring cycles into the operating area program, proceed as described in Section 10.2.1 ”Transferring Cycles to the Machining Plan”.
A vertical softkey menu will then be created behind the entry softkey ”Miscellaneous” → ”Zero point workpiece”, via which you can call up your measuring cycles. Such a vertical softkey menu already exists behind the ”Miscellaneous” → ”Measure tool” entry softkey.

10.3 ShopTurn Open (PCU 50.3)

In ShopTurn Open, the HMI Advanced operating areas ”Parameter” (without tool management and zero offsets), ”Services”, ”Diagnosis”, and ”Commissioning” are located on the extended horizontal softkey bar.

The extended horizontal softkey bar also includes the ”Machine Service” and ”Exit” softkeys for commissioning/diagnosis. These two softkeys are visible only when protection level 1 is selected.

In ShopMill Open you can also integrate Windows applications in the standard menu bar via softkeys.
You must also install the Windows applications in the OEM directory and define them in
the REGIE.INI file. This does not apply to Siemens additional products.
You must install those in the ADD_ON directory.
Refer to the ShopTurn DVD for details.

References: User’s Guide HMI Programming Package Part 1 (BN)

10.3.1 Basic menu bar

Jump into application

There are various methods by which you can integrate Windows applications into the basic menu bar:

- Horizontal softkey 4, i.e. the Messages/Alarms operating area is replaced
- User–assignable softkeys 7 and 8
- User–assignable softkeys 1 to 8 in the extended horizontal softkey menu

Return jump

There are 3 methods by which you can jump from the Windows application to another operating area:

- Using the ”Menu Select” key on the operator panel, you can return to the basic menu bar and then select another operating area via a softkey.
- You can configure the keys ”Position”, ”Program”, ”Offset”, ”Program Manager”, ”Alarm” and ”Custom” on the operator panel to branch directly into another operating area or sub–menu.
- You can configure so–called ”PLC keys” to branch into another operating area or sub–menu.

References: /IHA/, Commissioning Base Software and HMI Advanced, Operator Interface Extension
keyword ”OP Hotkeys” and ”PLC Keys”,
SINUMERIK 840D sl, 840D/840Di

10.4 Operator interface for ShopTurn on the NCU (HMI Embedded sl)

Default setting

The operating areas ”Machine” and ”Program” are by default hidden without manufacturer password. In the operating area ”Parameter” only the setting data and the zero offsets can be seen without manufacture password.

You can change the display by means of the following machine data

9272 MM_USER_CLASS_APPLICATION [1] = 1
Machine area

9272 MM_USER_CLASS_APPLICATION [3] = 1
Program area

9273 MM_USER_CLASS_APP_PARAMETER [1] = 1
Tool list

9273 MM_USER_CLASS_APP_PARAMETER [2] = 1
Tool wear

9273 MM_USER_CLASS_APP_PARAMETER [3] = 1
OEM tool data

9273 MM_USER_CLASS_APP_PARAMETER [4] = 1
Magazine

9273 MM_USER_CLASS_APP_PARAMETER [6] = 1
R parameters

9273 MM_USER_CLASS_APP_PARAMETER [10] = 1
User data

9414 MM_TM_KIND_OF_TOOLMANAGEMENT = 1
Graphical representation of the tool management

10.5 User status display

You can display machine states controlled via the PLC as custom user icons in the program status line.

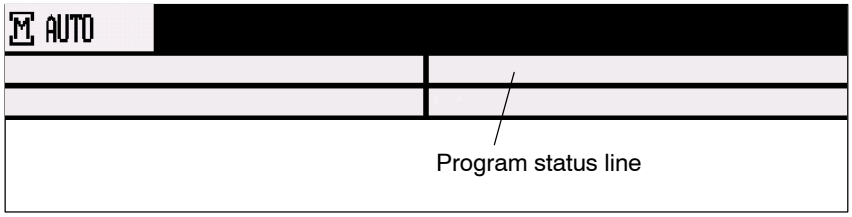


Fig. 10-1 Program status line

You need to activate the display of user icons in the display MD 9052 SHOW_CHANNEL_SPANNING_STATE.
The program path of the currently selected program is shown in the line below together with the program name.

16 display positions are defined in the program status bar.

User symbols

The custom user icons must meet the following requirements:

- Colors: 16–color mode
- Size: OP010/OP010C/OP010S: 16 x 16 pixels
  OP012: 20 x 20 pixels
  OP015: 27 x 26 pixel (height x width)
- File name: 8 characters
- Format: BMP

For ShopTurn on PCU 50.3 (HMI Advanced), you should store user symbols in the directory F:\DH\CUS.DIR\HLP.DIR.

For ShopTurn on NCU (HMI Embedded sl), you should place the icons in the following folders:

Table 10-5 Directory for saving user–specific icons

Storage folder	OP/resolution	Size of icons (height x width)
ico640	OP 010 (640 x 480)	16 x 16 pixels
ico800	OP 012 (800 x 600)	20 x 20 pixels
ico1024	OP 015 (1024 x 768)	27 x 26 pixels

The user–specific images are stored in directories (icoxxx) in the following sub–directories:

- Manufacturer: \oem\sinumerik\hmi\ico\icoxxx
- User: \oem\sinumerik\hmi\ico\icoxxx

10.5.1 Configuring the user status display

The cross–channel status display is allocated and configured with user symbols in the HEADER.INI file.

ShopTurn on PCU 50.3

The HEADER.INI file is located in the directory F:\HMI_ADV and must be copied either to the directory F:\OEM or F:\USER.

In the section `UserIcons` of the HEADER.INI file, enter the names of the user symbols and the signal for the control of the symbol selection.

Note

If you have already made these entries for HMI Advanced, you do not need to make them again for ShopTurn.

```
[User icons]
UI_0= <Icons_00.bmp>, <Position>
              UI_0:          Identifier
              Icons_00.bmp:  Name of the user icon
              Position:      Position for display (1 to 16)
...
UI_31= <Icons_31.bmp>, <Position>

USER_ICON_BASE = DBx.DBBy
              DBx.DBBy:      Signal defined by user for controlling icon
                             selection
```

The user icons are addressed bit by bit, i.e. if bit **n** is set in signal DBx.DBBy, then the user icon with identifier UI_**n** is displayed.
If the bit is reset by the PLC, the associated user icon is deleted in the program status display.

If several user symbols are assigned to the same position, the user symbol with the highest identifier number is displayed.
You do not need to specify blank positions.

Additional information can be found in

References: /IHA/, Commissioning Base Software and HMI Advanced, SINUMERIK 840D sl/840D/840Di

ShopTurn on NCU (HMI Embedded sl)

The CompactFlash card contains the HEADER.INI file in the directory siemens\sinumerik\hmi\cfg\

Copy the HEADER.INI file to the directory oem\sinumerik\hmi\cfg.
Open the file using the editor and assign the desired positions to the user symbols.

Additional information can be found in

References: /IHE/, Commissioning Base Software and HMI Embedded sl, SINUMERIK 840D sl

10.6 OP hotkeys, PLC keys

OP hotkeys

You can configure the keys ”Position”, ”Program”, ”Offset”, ”Program Manager”, ”Alarm”, and ”Custom” on the operator panel to branch to an operating area of your choice.

PLC keys

You can configure so–called ”PLC keys” to branch to particular operating areas.

Configuration

You can switch to the following operating areas with the OP hotkeys or PLC keys:

- ShopTurn on NCU (HMI Embedded sl)
  From ShopTurn to any operating area of ShopTurn.
- PCU 50.3
  From ShopTurn to any other operating area in ShopTurn.
  From ShopTurn to any other operating area in HMI Advanced and vice versa.

Information about the configuration can be found in:
References: /IHA/, Commissioning Base Software and HMI Advanced, Operator Interface Extension
keyword ”OP Hotkeys” and ”PLC Keys”,
SINUMERIK 840D sl, 840D/840Di

Note

The configuration of the ”PLC Keys” in ShopTurn is carried out in the file KEYS.INI, not via the DB 19 interface.
The file KEYS.INI must feature the following entry in the section [HMI_INI_FILES]:

```
Task6 = shopturn.ini
```

■

Miscellaneous 11

11.1 Machine data for protection levels

Lockable data areas

The machine manufacturer or end user can set operator panel machine data to assign the appropriate protection levels for individual functions and data areas For a number of data types, it is possible to input different protection levels for read and write access authority.
The machine data which can be locked by means of protection levels in Shop-Turn is listed below:

Operator panel machine data	Access to
9182 USER_CLASS_INCH_METRIC	Switch over between Inch/metric
9200 USER_CLASS_READ_TOA	Read tool offsets
9201 USER_CLASS_WRITE_TOA_GEO	Write tool geometry (including type and cutting edge)
9202 USER_CLASS_WRITE_TOA_WEAR	Write tool wear (without limit value)
9203 USER_CLASS_WRITE_FINE	Write fine tool wear and fine offsets
9206 USER_CLASS_WRITE_TOA_SUPVIS	Change limit values for tool monitoring
9210 USER_CLASS_WRITE_ZOA	Write rough offsets
9215 USER_CLASS_WRITE_SEA	Write setting data
9216 USER_CLASS_READ_PROGRAM (protection level must be ≤ 6)	Read program
9217 USER_CLASS_WRITE_PROGRAM (protection level must be ≤ 5)	Write/edit program
9218 USER_CLASS_SELECT_PROGRAM (protection level must be ≤ 7)	Enable program selection
9222 USER_CLASS_WRITE_RPA	Write R parameters
9252 USER_CLASS_TM_SKTOOLLOAD	Enable loading of a tool
9253 USER_CLASS_TM_SKTOOLUNLOAD	Enable unloading of a tool
9254 USER_CLASS_TM_SKTOOLMOVE	Enable relocation of a tool
9258 USER_CLASS_TM_SKNCNEWTOOLE	Enable setting up of new cutting edges

9259 USER_CLASS_TM_SKNCDELTOOL	Enable deleting of a tool
9264 USER_CLASS_TM_SKTLNEWTOOL	Enable setting up of a new tool
9510 USER_CLASS_DIRECTORY1_P	Enable network drive 1
9511 USER_CLASS_DIRECTORY2_P	Enable network drive 2
9512 USER_CLASS_DIRECTORY3_P	Enable network drive 3
9513 USER_CLASS_DIRECTORY4_P	Enable network drive 43
9890 ST_USER_CLASS_MEAS_T_CAL	Calibrate tool probe

11.2 Machine manufacturer cycle

The machine manufacturer cycle ST_CUST.SPF is called by the ShopTurn cycles. Adapt the cycle if you want to perform one of the following actions:

- Switch between spindle and C axis mode of the main spindle or counter–spindle.
  Marks _M1, _M2, _M21, _M22
  (see also Section 9.6 ”Counter–spindle”)
- Clamp or release the rotary axes (main spindle/counter–spindle).
  Marks _M3, _M4, _M23, _M24
  (see also MD 9843 $MM_ST_ENABLE_SPINDLE_CLAMPING)
- Open, close, flush chuck (main spindle /counter–spindle).
  Marks _M5 to _M8, _M25 to _M29
  (see also Section 9.6 ”Counter–spindle”)
- Engage/disengage rotating tool (make/break connection to drive).
  Marks _M41, _M42
- Configure special functions for switchover between machining planes. (You do not have to make any settings for cylinder surface transformation or face end machining with the C axis.)
  Marks _M61 to _M68
- Position, extend, or retract receptacle for cutting off.
  Marks _M100, _M101, _M102
  (see also MD 9841 $MM_ST_ENABLE_PART_OFF_RECEPT)
- Configure special functions for tool changing. (These special functions are called after the T command is output.)
  Marks _M110, _M111, _M112
- Change default for coupling between main spindle and counter–spindle.
  Mark _M120
  (see also Section 9.6 ”Counter–spindle”)
- Set special properties for program start or program end.
  Marks _M131, _M135, _M136

The cycle is located in directory CYCLES\SC\PROG\TEMPLATES.

11.3 Spindle control

For configuration of the spindle control, please refer to:

References: /FB1/, Function Manual, Basic Machine, Spindles (S1)

End of program

ShopTurn distinguishes between M2/M30 (end of a ShopTurn program) and the M function set in MD 10714 $MN_M_NO_FCT_EOP (end of a program created in ”Manual” or ”MDI” mode).
Setting MD 35040 $MA_SPIND_ACTIVE_AFTER_RESET=2 causes the NCK to switch off the spindle in response to M2/M30 but not in response to the M function programmed in MD 10714 $MN_M_NO_FCT_EOP.
One of the purposes of this functionality is to allow continuous starting of the spindle in manual mode (e.g. for scratching).

ShopTurn uses the following M functions for end of program:

- M function from MD 10714 $MN_M_NO_FCT_EOP: End of main program, spindle continues
- M2, M30: End of main program and return jump to beginning of program, spindle stops
- M17: End of subprogram and return jump to main program, spindle continues operating

Direction of rotation

In order to ensure that the rotational direction of the spindle and C axis is correctly displayed in the ShopTurn interface and executed directly for programmed ShopTurn functions you must enter a number of mutually coordinated settings. You must base these settings on the actual direction of rotation of the spindle/C axis on the machine.

If you want to use functions cylinder surface transformation and end face machining, commissioning must have been correctly performed for the functions. The following machine data in particular must be correctly set:
24810 $MC_TRACYL_ROT_SIGN_IS_PLUS_1
24860 $MC_TRACYL_ROT_SIGN_IS_PLUS_2
24910 $MC_TRANSMIT_ROT_SIGN_IS_PLUS_1
24960 $MC_TRANSMIT_ROT_SIGN_IS_PLUS_2

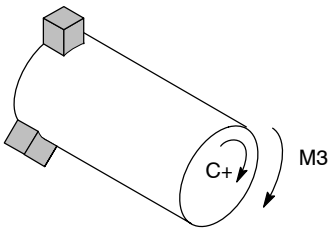
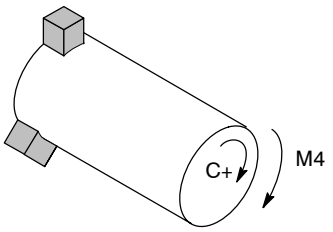
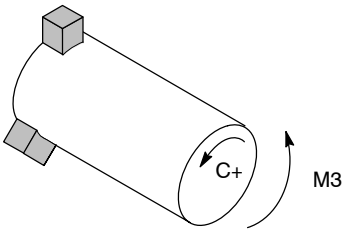
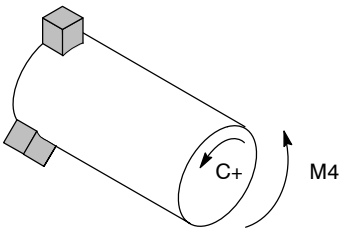
In display machine data you stipulate which direction of rotation of the spindle/C axis is to be displayed for M function M3 in the ShopTurn interface.

MD 9822 $MM_ST_DISPL_DIR_MAIN_SPIND_M3
MD 9823 $MM_ST_DISPL_DIR_SUB_SPIND_M3
MD 9824 $MM_ST_DISPL_DIR_MAIN_C_AX_INV
MD 9825 $MM_ST_DISPL_DIR_SUB_C_AX_INV

The direction of spindle rotation (M3/M4) is assigned to the positive rotational direction of the C axis via interface signal DB3n.DBX17.6 (where n = index of the relevant C axis). The bit defines whether M3 and C+ rotate in the same direction (=0) or in opposite directions (=1).

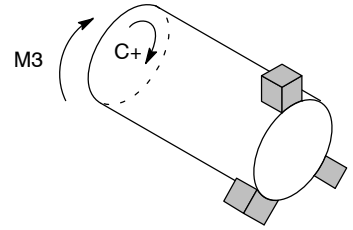
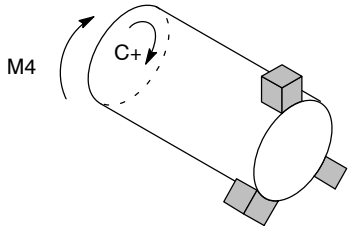
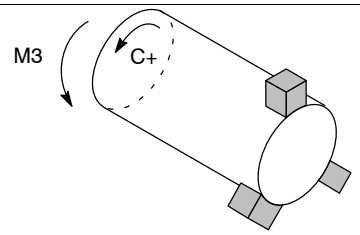
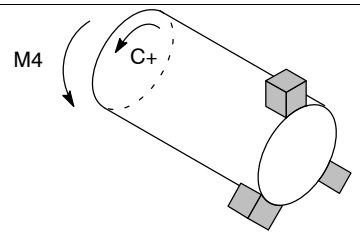
The following settings are possible for the main spindle and the counter–spindle; the settings for machine data 9822 and 9823 depend on the direction in which the user views the coordinate axis. The settings for machine data 9824 and 9825 and the interface signal DB3n.DBX17.6 must be observed.

Table 11-1 Direction of rotation of main spindle

Main spindle direction of rotation	MD 9822*	MD 9824	DB3n.DBX17.6
	0	0	0
	1	0	1
	1	1	0
	0	1	1

The setting of machine data marked with "*" assumes that the viewing direction was towards the negative coordinate axis. If on the other hand you are looking towards the positive coordinate axis, you need to reverse the values, i.e. swap "0" and "1".
Machine data 9822 is only relevant for display in the ShopTurn interface, not for correct operation on the machine.

Table 11-2 Direction of rotation of counter–spindle

Counter–spindle direction of rotation	MD 9823*	MD 9825	DB3n.DBX17.6
	1	0	0
	0	0	1
	0	1	0
	1	1	1

The setting of machine data marked with "*" assumes that the viewing direction was towards the negative coordinate axis. If on the other hand you are looking towards the positive coordinate axis, you need to reverse the values, i.e. swap "0" and "1".
Machine data 9823 is only relevant for display in the ShopTurn interface, not for correct operation on the machine.

11.4 ISO dialects

You can also create and run ISO dialect programs under ShopTurn. For more information on how to set up ISO dialects, please refer to the following documentation:

References: /FBFA/, Function Manual on ISO Dialects for SINUMERIK

11.5 Automatically generated programs

ShopTurn creates certain programs automatically during operation.

Directory of part programs:

CMM_SINGLE The program is created when a function is executed in ”Manual” mode. The program is started via an asynchronous subprogram.

INPUT_DATA_MM
INPUT_DATA_IN The last parameter values entered in a screen are saved in these programs, depending on the unit of measurement. (INPUT_DATA_MM = values with the unit ”mm”; INPUT_DATA_IN = values with the unit ”Inch”)

Directory commissioning:

REM_DATA.TRC Data which needs to remain stored when the machine is next switched off (e.g. the last selected program) is saved in this program.

11.6 Manual machine

Option

”Manual machine” is an option (Order No. 6FC5 800–0AP11–0YB0).

The ”Manual machine” option allows work to be carried out with large cycles in manual mode, without the need to write programs for this purpose.

For a more detailed description of the cycles available in ”Manual” mode, and how to work with the ”Manual machine” option, please refer to:

References: /BATsl/, Operation/Programming ShopTurn

Manual operation in the ”Manual” mode is based on the jogging functions in the NCK.

Tool selection

For tool selection enter the name of the tool and start by means of ”Cycle start”.

- Turret revolver
  With ”Cycle start” the turret revolver starts to rotate.
- Multifix
  With ”Cycle start” tool correction is activated.

Feedrate

SD 42600 JOG_FEED_PER_REV_SOURCE is used to stipulate whether one or two feed fields should be available in the basic screen ”Manual”.

- 42600 JOG_FEED_PER_REV_SOURCE = 0
  One feedrate field is offered.
  The entry field can be switched from mm/min to mm/rev. SD 41100 JOG_REV_IS_ACTIVE is then altered from 0 (mm/min) to 1 (mm/rev).

- 42600 JOG_FEED_PER_REV_SOURCE = –3
  Two separate feedrate fields are offered for mm/min and mm/rev.
  The JOG function only uses the mm/rev feedrate when the main spindle is rotating. Otherwise, it uses the mm/min value.

At the same time, the axis–specific setting data SD 43300 must be set so that it is identical for each of the relevant axes.

Feedrate override

With the MD 12200 $MN_RUN_OVERRIDE_0 = 1, it is possible to move the handwheels even when there is a 0% feedrate override.

Rotational speed (main spindle)

The speed entry field can be switched between rpm and m/min.
SD 43206 SA_SPIND_SPEED_TYPE is set accordingly to the appropriate G code (G94, G95, G96).

- rpm
  SD 43200 SPIND_S is written.
- m/min
  SD 43202 SPIND_CONSTCUT_S is written.

You have the following possibilities for starting the spindle:

- MD 9897 MM_ST_OPTION_MASK_MAN_FUNC Bit 0 = 1
  Spindle start by means of ”Cycle start”.
  Select rpm or m/min and enter the desired value.
  Select the direction of rotation and start the spindle by means of ”Cycle start”.
- MD 9897 MM_ST_OPTION_MASK_MAN_FUNC Bit0 = 0
  Spindle start by means of external switch.
  Select rpm or m/min and enter the desired value.
  DB3x.DBB30.1 starts the spindle rotating clockwise.
  DB3x.DBB30.2 starts the spindle rotating counter–clockwise.
  DB3x.DBB30.0 stops the spindle.

Rotational speed (driven tool)

The rotational speed is displayed in rpm.
SD 43206 SA_SPIND_SPEED_TYPE is set.
SD 43200 SPIND_S is written.

You have the following possibilities for starting the spindle:

- MD 9897 MM_ST_OPTION_MASK_MAN_FUNC Bit 0 = 1
  Spindle start by means of ”Cycle start”.
  In the input field enter the desired value for the rotational speed.
  Select the direction of rotation and start the spindle by means of ”Cycle start”.
- MD 9897 MM_ST_OPTION_MASK_MAN_FUNC Bit0 = 0
  Spindle start by means of external switch.
  In the input field enter the desired value for the rotational speed.
  DB3x.DBB30.1 starts the spindle rotating clockwise.
  DB3x.DBB30.2 starts the spindle rotating counter–clockwise.
  DB3x.DBB30.0 stops the spindle.

11.7 Action log

You can log sequences of operations in ShopTurn in the action log to obtain a history of events for later examination.

More detailed information about the action log can be found in:
References: /IHE/, Commissioning Base Software and HMI Embedded sl, SINUMERIK 840D sl

Please refer to directory DOCUMENTATION\WINDOWLISTS on the ShopTurn DVD for the assignment between ID numbers listed in the log and the corresponding ShopTurn windows.

Note

If you are using a PCU 50.3 with HMI Advanced and wish to log ShopTurn operating sequences, you must enter the settings for the action log directly in MD 9012 $MM_ACTION_LOG_MODE. The log is stored in the F:\MMCOW32\TMP_AC_LOG.BIN file. No support for display or readout of this file is provided.

11.8 Version display

You can see the ShopTurn version in the boot screen.

The version data of the installed system software are output in a version display.

- For this purpose switch to the CNC ISO operator interface.
- Select the menu ”Diagnostics” —> ”Service displays” —> ”Version”.

More detailed information about the version display can be found in the following documentation:

References: /IHE/, Commissioning Base Software and HMI Embedded sl, SINUMERIK 840D sl;
/BAD/ Operating Manual HMI Advanced, SINUMERIK 840D sl/840D/840Di
BEMsl/ Operating Manual HMI Embedded sl, SINUMERIK 840D sl

■

Abbreviations

A

ASUB	Asynchronous subroutine
BAG	Mode group
BUFF	Buffer
CNC	Computerized Numerical Control
COM	Communication Component of NC control that performs and coordinates communication.
DB	Data block
DBB	Data Block Byte
DBD	Data Block Double Word
DBW	Data Block Word
DBX	Data Block Bit
FC	Function Call, function block on the PLC
GUD	Global User Data
HPU	Handheld Programming Unit
IBN	Commissioning
MCP	Machine control panel
MD	Machine data
MDA	Manual Data Automatic
MPF	Main program file: Part program (main program)
MPI	Multi–Port Interface
NC	Numerical Control The NC control comprises the components NCK, PLC, PCU and COM.

NCK	Numerical Control Kernel Component of NC control that executes programs and basically coordinates movements for the machine tool.
NCU	Numerical Control Unit: NC module
OB	Organization block in the PLC
OP	Operator Panel
PC	Personal computer
PCU	Personal Computer Unit Component of NC control allowing communication between operator and machine.
PG	Programming device
PLC	Programmable Logic Control Component of NC control for processing machine tool control logic
RAM	Random Access Memory, i.e. program memory that can be read and written to
SD	Setting data
SK	Softkey
SPF	Sub Program File
STL	Statement List
SW	Software
TM	Tool management
T No.	Tool number
TOA	Tool Offset Active: Identifier for tool offsets
WZ	Tool

■

Index

O

P

R

S

T

U

V

W

To
SIEMENS AG
A&D MC MS1
Postfach 3180

D–91050 Erlangen

Phone: +49 (0) 180 5050 – 222 [Hotline]
Fax: +49 (0) 9131 98 – 63315 [Documentation]
mailto:docu.motioncontrol@siemens.com

Suggestions

Corrections

For Publication/Manual:

SINUMERIK 840D sl

ShopTurn

Manufacturer/Service Documentation

Commissioning Manual

Order No.: 6FC5397–5AP10–2BA0
Edition: 01/2008

If you find any errors when reading this publication, please use this form to notify us. Suggestions for improvement are also welcome.

From

Name

Address of your company/department

Address

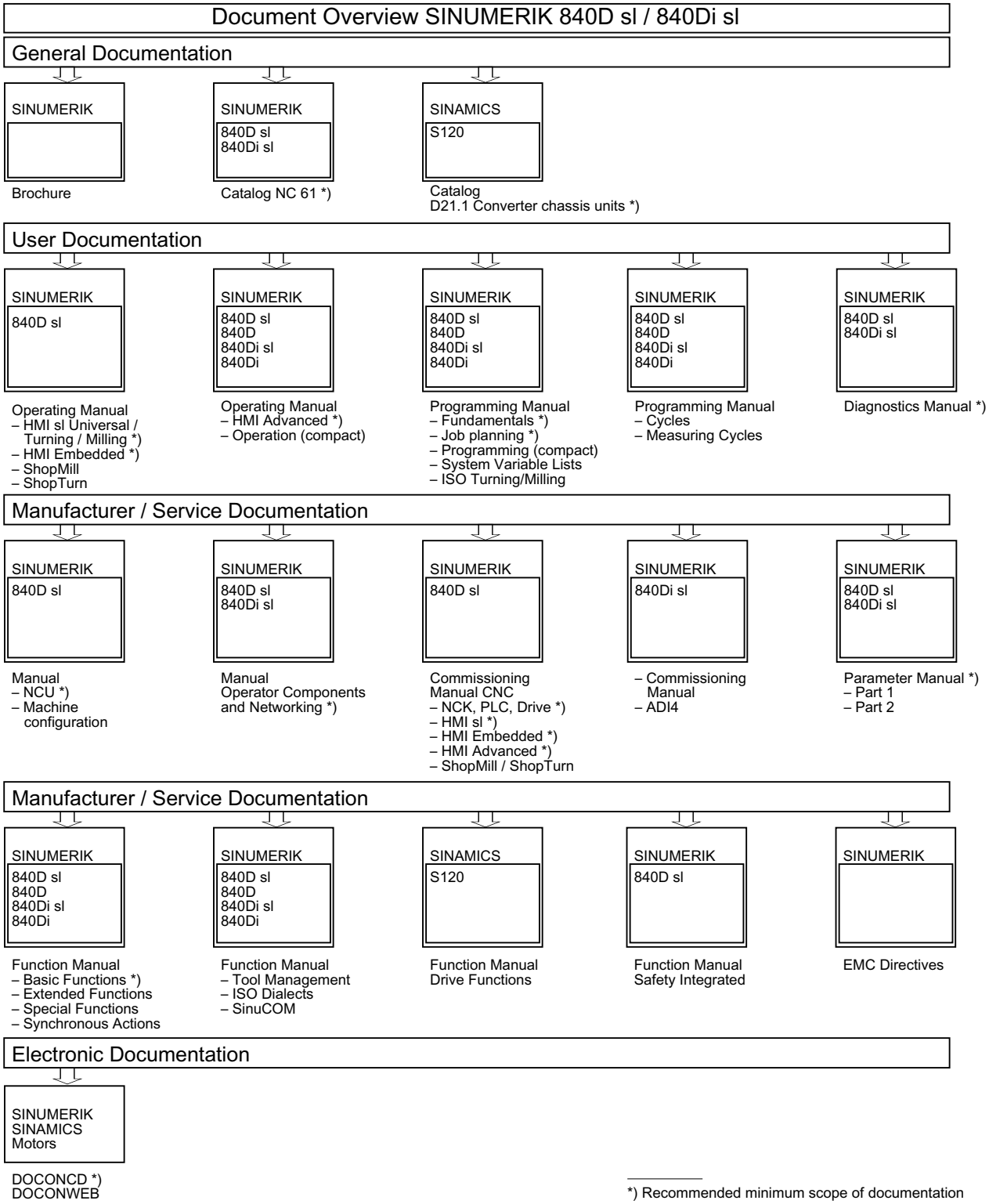
Zip code: City:

Phone: /

Fax: /

Suggestions and/or corrections

Overview



*) Recommended minimum scope of documentation