SIEMENS

SINUMERIK

SINUMERIK ONE Universal

Operating Manual

Introduction 1
Fundamental safety instructions 2
Fundamentals 3
Multi-touch operation 4
Setting up the machine 5
Working in manual mode 6
Machining the workpiece 7
Simulating machining 8
Creating a G code program 9
Multi-channel view 10
Collision avoidance 11
Tool management 12
Managing programs 13
Teaching in a program 14
Handheld Terminal HT 10 for multi-touch operation 15
Ctrl-Energy 16
Editing the PLC user program 17
Service and diagnostics 18

Valid for
Control
SINUMERIK ONE

CNC software version V6.25 SP1
SINUMERIK Operate for PCU/PC V6.25 SP1

07/2025
A5E48384812B AK

Legal information

Warning notice system

This manual contains notices you have to observe in order to ensure your personal safety, as well as to prevent damage to property. The notices referring to your personal safety are highlighted in the manual by a safety alert symbol, notices referring only to property damage have no safety alert symbol. These notices shown below are graded according to the degree of danger.

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

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

⚠ **CAUTION**
indicates that minor personal injury can result if proper precautions are not taken.

NOTICE
indicates that property damage can result if proper precautions are not taken.

If more than one degree of danger is present, the warning notice representing the highest degree of danger will be used. A notice warning of injury to persons with a safety alert symbol may also include a warning relating to property damage.

Qualified Personnel

The product/system described in this documentation may be operated only by **personnel qualified** for the specific task in accordance with the relevant documentation, in particular its warning notices and safety instructions. Qualified persons are those who, because of their training and experience, are familiar with the installation, assembly, commissioning, operation, decommissioning and disassembly of the product and can recognize risks and avoid possible hazards.

Proper use of Siemens products

Note the following:

⚠ **WARNING**
Siemens products may only be used for the application described in the catalog and the associated usage information. If products and components from other manufacturers are used, these must be recommended or approved by Siemens. Proper transport, storage, installation, assembly, commissioning, operation and maintenance are required to ensure that the products operate safely and without any problems. The permissible ambient conditions must be complied with. The information in the relevant documentation must be observed.

Trademarks

All names identified by ® are registered trademarks of Siemens Aktiengesellschaft. The remaining trademarks in this publication may be trademarks whose use by third parties for their own purposes could violate the rights of the owner.

Disclaimer of Liability

We have reviewed the contents of this publication to ensure consistency with the hardware and software described. Since variance cannot be precluded entirely, we cannot guarantee full consistency. However, the information in this publication is reviewed regularly and any necessary corrections are included in subsequent editions.

Siemens Aktiengesellschaft
Digital Industries
Postfach 48 48
90026 NÜRNBERG
GERMANY

A5E48384812B AK
Ⓟ 07/2025 Subject to change

Copyright © Siemens 2019 - 2025.
All rights reserved

Table of contents

1 Introduction 13

1.1 About SINUMERIK 13

1.2 About this documentation 14

1.3 Documentation on the internet 15
1.3.1 Documentation overview SINUMERIK ONE 15
1.3.2 Documentation overview SINUMERIK operator components 15

1.4 Feedback on the technical documentation 17

1.5 mySupport documentation 18

1.6 Service and Support 19

1.7 Using OpenSSL 21

1.8 Compliance with the General Data Protection Regulation 22

2 Fundamental safety instructions 23

2.1 General safety instructions 23

2.2 Warranty and liability for application examples 24

2.3 Cybersecurity information 25

3 Fundamentals 27

3.1 Product overview 27

3.2 Operator components 28
3.2.1 Touch panels 28
3.2.2 Machine control panels 29
3.2.2.1 Overview 29
3.2.2.2 Controls on the machine control panel 29

3.3 Operator interface 34
3.3.1 Screen layout 34
3.3.2 Status display 35
3.3.3 Actual value window 37
3.3.4 T,F,S window 39
3.3.5 Current block display 41
3.3.6 Operation via softkeys and buttons 43
3.3.7 Key combinations on a PC keyboard 44
3.3.8 Entering or selecting parameters 45
3.3.9 Pocket calculator 47
3.3.10 Pocket calculator functions 47
3.3.11 Context menu 50
3.3.12 Changing the user interface language 50
3.3.13 Entering Chinese characters 51
3.3.13.1 Function - input editor 51
3.3.13.2 Entering Asian characters 52

3.3.13.3 Editing the dictionary ... 54
3.3.14 Entering Korean characters ... 55
3.3.15 Protection levels ... 58
3.3.16 Work station safety ... 60
3.3.17 Cleaning mode ... 60
3.3.18 Display live image from a camera ... 61
3.3.19 Online help in SINUMERIK Operate ... 62

4 Multi-touch operation ... 65

4.1 Touch-sensitive user interface ... 65
4.2 Finger gestures ... 66
4.3 Multitouch user interface ... 69
4.3.1 Screen layout ... 69
4.3.2 Function key block ... 70
4.3.3 Further operator touch controls ... 71
4.3.4 Virtual keyboard ... 71
4.3.5 Special "tilde" character ... 72
4.4 Expansion with side screen ... 73
4.4.1 Overview ... 73
4.4.2 Sidescreen with standard windows ... 73
4.4.3 Standard widgets ... 75
4.4.4 "Actual value" widget ... 76
4.4.5 "Zero point" widget ... 76
4.4.6 "Alarms" widget ... 76
4.4.7 "NC/PLC variables" widget ... 76
4.4.8 "Axle load" widget ... 77
4.4.9 "Tool" widget ... 77
4.4.10 "Service life" widget ... 78
4.4.11 "Program runtime" widget ... 78
4.4.12 Widget "Camera 1" and "Camera 2" ... 78
4.4.13 Sidescreen with pages for the ABC keyboard and/or machine control panel ... 79
4.4.14 Example 1: ABC keyboard in the sidescreen ... 80
4.4.15 Example 2: Machine control panel in the sidescreen ... 81
4.5 SINUMERIK Operate Display Manager ... 82
4.5.1 Overview ... 82
4.5.2 Screen layout ... 83
4.5.3 Operator controls ... 83

5 Setting up the machine ... 87

5.1 Switching on and switching off ... 87
5.2 Approaching a reference point ... 88
5.2.1 Referencing axes ... 88
5.2.2 User agreement ... 89
5.3 Modes of operation ... 91
5.3.1 General information ... 91
5.3.2 Modes groups and channels ... 93
5.3.3 Channel switchover ... 93
5.4 Settings for the machine ... 95
5.4.1 Switching over the coordinate system (MCS/WCS) ... 95

5.4.2 Switching the unit of measurement ... 95
5.4.3 Setting the zero offset ... 97

5.5 Zero offsets ... 99
5.5.1 Display active zero offset ... 100
5.5.2 Displaying the zero offset "overview" ... 101
5.5.3 Displaying and editing base zero offset ... 102
5.5.4 Displaying and editing settable zero offset ... 103
5.5.5 Displaying and editing details of the zero offsets ... 103
5.5.6 Deleting a zero offset ... 105

5.6 Monitoring axis and spindle data ... 107
5.6.1 Specify working area limitations ... 107
5.6.2 Editing spindle data ... 107

5.7 Displaying setting data lists ... 109

5.8 Handwheel assignment ... 110

5.9 MDA ... 112
5.9.1 Loading an MDA program from the Program Manager ... 112
5.9.2 Saving an MDA program ... 113
5.9.3 Editing/executing a MDI program ... 114
5.9.4 Deleting an MDA program ... 115

6 Working in manual mode ... 117

6.1 General ... 117

6.2 Selecting a tool and spindle ... 118
6.2.1 T, S, M windows ... 118
6.2.2 Selecting a tool ... 119
6.2.3 Starting and stopping a spindle manually ... 120
6.2.4 Position spindle ... 121

6.3 Traversing axes ... 122
6.3.1 Traverse axes by a defined increment ... 122
6.3.2 Traversing axes by a variable increment ... 123

6.4 Positioning axes ... 125

6.5 Manual retraction ... 126

6.6 Default settings for manual mode ... 127

7 Machining the workpiece ... 129

7.1 Starting and stopping machining ... 129

7.2 Selecting a program ... 131

7.3 Program running-in ... 132

7.4 Displaying the current program block ... 134
7.4.1 Current block display ... 134
7.4.2 Displaying a basic block ... 135
7.4.3 Display program level ... 136

7.5 Correcting a program ... 138

7.6 Repositioning axes ... 139

7.7 Starting execution at a specific point ... 141
7.7.1 Use block search ... 141
7.7.2 Continuing program from search target ... 143
7.7.3 Simple search target definition ... 143
7.7.4 Defining an interruption point as search target ... 144
7.7.5 Entering the search target via search pointer ... 144
7.7.6 Parameters for block search in the search pointer ... 145
7.7.7 Block search mode ... 146

7.8 Controlling the program run ... 148
7.8.1 Program control ... 148
7.8.2 Use Powerride for program control ... 150
7.8.3 Skip blocks ... 151

7.9 Overstore ... 152

7.10 Editing a program ... 154
7.10.1 Searching in programs ... 154
7.10.2 Replacing program text ... 156
7.10.3 Copying/pasting/deleting program blocks ... 157
7.10.4 Renumbering a program ... 159
7.10.5 Creating a program block ... 160
7.10.6 Opening additional programs ... 161
7.10.7 Editor settings ... 162

7.11 Display and edit user variables ... 167
7.11.1 Global R parameters ... 168
7.11.2 R parameters ... 169
7.11.3 Displaying global user data (GUD) ... 171
7.11.4 Displaying channel GUDs ... 172
7.11.5 Displaying local user data (LUD) ... 173
7.11.6 Displaying program user data (PUD) ... 174
7.11.7 Searching for user variables ... 174

7.12 Display G and auxiliary functions ... 177
7.12.1 Selected G functions ... 177
7.12.2 All G functions ... 179
7.12.3 G functions for mold making ... 179
7.12.4 Auxiliary functions ... 180

7.13 Displaying superimpositions ... 182

7.14 Displaying status of synchronized actions ... 183

7.15 Displaying the program runtime and counting workpieces ... 185

7.16 Settings for the automatic mode ... 187

7.17 Working with DXF files ... 190
7.17.1 Overview ... 190
7.17.2 Displaying CAD drawings ... 191
7.17.2.1 Open a DXF file ... 191
7.17.2.2 Cleaning a DXF file ... 191
7.17.2.3 Enlarging or reducing the CAD drawing ... 192
7.17.2.4 Changing the section ... 193
7.17.2.5 Rotating the view ... 193
7.17.2.6 Displaying/editing information for the geometric data ... 194

7.17.3 Importing and editing a DXF file ... 195
7.17.3.1 General procedure ... 195
7.17.3.2 Setting the tolerance ... 195
7.17.3.3 Assigning the machining plane ... 196
7.17.3.4 Selecting the machining range / deleting the range and element ... 196
7.17.3.5 Saving the DXF file ... 197
7.17.3.6 Specifying a reference point ... 198
7.17.3.7 Accepting contours ... 199

7.18 Mold making view ... 202
7.18.1 Starting the mold making view ... 204
7.18.2 Adapting the mold making view ... 204
7.18.3 Specifically jump to the program block ... 205
7.18.4 Searching for program blocks ... 206
7.18.5 Changing the view ... 206
7.18.5.1 Enlarging or reducing the graphical representation ... 206
7.18.5.2 Moving and rotating the graphic ... 207
7.18.5.3 Modifying the viewport ... 208

8 Simulating machining ... 211

8.1 Overview ... 211

8.2 Simulation prior to machining of the workpiece ... 219

8.3 Simultaneous recording prior to machining of the workpiece ... 221

8.4 Simultaneous recording during machining of the workpiece ... 222

8.5 Setting the model quality ... 223

8.6 Different views of the workpiece ... 224
8.6.1 Overview ... 224
8.6.2 Plan view ... 224
8.6.3 3D view ... 225
8.6.4 Side view ... 225

8.7 Editing the simulation display ... 227
8.7.1 Blank display ... 227
8.7.2 Showing and hiding the tool path ... 227

8.8 Program control during simulation ... 228
8.8.1 Changing the feedrate ... 228
8.8.2 Simulating the program block by block ... 229

8.9 Modifying and adapting the simulation graphics ... 230
8.9.1 Enlarging or reducing the graphical representation ... 230
8.9.2 Panning a graphical representation ... 231
8.9.3 Rotating the graphical representation ... 231
8.9.4 Modifying the viewport ... 232
8.9.5 Defining cutting planes ... 232

8.10 Displaying simulation alarms ... 234

9 Creating a G code program ... 235

9.1 Graphical programming ... 235

9.2 Program views ... 236

9.3 Program structure ... 240

9.4 Fundamentals ... 241
9.4.1 Machining planes ... 241
9.4.2 Current planes in cycles and input screens ... 241
9.4.3 Programming a tool (T) ... 242

9.5 Generating a G code program ... 243

9.6 Blank input ... 244
9.6.1 Overview ... 244
9.6.2 Calling the input screen ... 245
9.6.3 Blank input ... 245

9.7 Selection of the cycles via softkey ... 247

10 Multi-channel view ... 249

10.1 Multi-channel view ... 249

10.2 Multi-channel view in the "Machine" operating area ... 250

10.3 Setting the multi-channel view ... 253

11 Collision avoidance ... 255

11.1 Activate collision avoidance ... 257

11.2 Set collision avoidance ... 258

12 Tool management ... 261

12.1 Lists for the tool management ... 261

12.2 Magazine management ... 263

12.3 Tool types ... 264

12.4 Tool dimensioning ... 267

12.5 Tool list ... 275
12.5.1 Creating a new tool ... 277
12.5.2 Additional data ... 278
12.5.3 Managing several cutting edges ... 280
12.5.4 Deleting a tool ... 281
12.5.5 Loading and unloading tools ... 282
12.5.6 Selecting a magazine ... 283
12.5.7 Code carrier connection ... 284
12.5.8 Managing a tool in a file ... 286

12.6 Tool wear ... 289
12.6.1 Reactivating a tool ... 291

12.7 Tool data OEM ... 293

12.8 Magazine ... 294
12.8.1 Position magazine ... 296
12.8.2 Relocating a tool ... 296
12.8.3 Deleting / unloading / loading / relocating all tools ... 297

12.9 Tool details ... 299
12.9.1 Displaying tool details ... 299

12.9.2 Cutting edge data ... 299
12.9.3 Tool data ... 301
12.9.4 Monitoring data ... 302
12.10 Changing a tool type ... 304
12.11 Sorting tool management lists ... 305
12.12 Filtering the tool management lists ... 307
12.13 Specific search in the tool management lists ... 309
12.14 Multiple selection in the tool management lists ... 311
12.15 Working with multitool ... 312
12.15.1 Tool list for multitool ... 312
12.15.2 Create multitool ... 313
12.15.3 Equipping multitool with tools ... 315
12.15.4 Removing a tool from multitool ... 316
12.15.5 Deleting multitool ... 317
12.15.6 Loading and unloading multitool ... 317
12.15.7 Reactivating the multitool ... 318
12.15.8 Relocating a multitool ... 319
12.15.9 Positioning a multitool ... 320
12.16 Settings for tool lists ... 321
13 Managing programs ... 323
13.1 Overview ... 323
13.1.1 NC memory ... 325
13.1.2 Local drive ... 326
13.1.3 USB drives ... 327
13.1.4 FTP drive ... 328
13.2 Opening and closing a program ... 329
13.3 Executing a program ... 331
13.4 Creating a directory/program/job list/program list ... 333
13.4.1 File and directory names ... 333
13.4.2 Creating a new directory ... 333
13.4.3 Creating a new workpiece ... 334
13.4.4 Creating a new G code program ... 335
13.4.5 Storing any new file ... 336
13.4.6 Creating a job list ... 337
13.4.7 Creating a program list ... 338
13.5 Creating templates ... 340
13.6 Searching directories and files ... 341
13.7 Displaying the program in the Preview ... 343
13.8 Selecting several directories/programs ... 344
13.9 Copying and pasting a directory/program ... 346
13.10 Deleting a directory/program ... 348
13.11 Changing file and directory properties ... 349

13.12 Set up drives ... 351
13.12.1 Overview ... 351
13.12.2 Setting up drives ... 351
13.13 Viewing PDF documents ... 358
13.14 EXTCALL ... 361
13.15 Execution from external memory (EES) ... 364
13.16 Backing up data ... 365
13.16.1 Generating an archive in the Program Manager ... 365
13.16.2 Generating an archive via the system data ... 366
13.16.3 Reading in an archive in the Program Manager ... 368
13.16.4 Read in archive from system data ... 369
13.17 Setup data ... 371
13.17.1 Reading-in set-up data ... 373
13.18 Recording tools and determining the demand ... 375
13.18.1 Overview ... 375
13.18.2 Opening tool data ... 376
13.18.3 Checking the loading ... 377
13.19 Backing up parameters ... 379
13.20 RS-232-C ... 382
13.20.1 Reading-in and reading-out archives via a serial interface ... 382
13.20.2 Setting V24 in the program manager ... 383
14 Teaching in a program ... 385
14.1 Overview ... 385
14.2 Select teach in mode ... 387
14.3 Processing a program ... 388
14.3.1 Inserting a block ... 388
14.3.2 Editing a block ... 388
14.3.3 Selecting a block ... 389
14.3.4 Deleting a block ... 389
14.4 Teach sets ... 391
14.4.1 Input parameters for teach-in blocks ... 392
14.5 Settings for teach-in ... 394
15 Handheld Terminal HT 10 for multi-touch operation ... 395
15.1 HT 10: Overview ... 395
15.2 Machine control panel menu ... 398
15.3 Virtual keyboard ... 400
15.4 Calibrating the touch panel ... 401
16 Ctrl-Energy ... 403
16.1 Functions ... 403
16.2 Ctrl-E analysis ... 405
16.2.1 Displaying energy consumption ... 405

16.2.2 Displaying the energy analyses ... 406
16.2.3 Assigning language-dependent names for auxiliary units ... 407
16.2.4 Making settings for measurements ... 408
16.2.5 Measuring and saving the energy consumption ... 409
16.2.6 Evaluating measurement results ... 411
16.2.6.1 Overview ... 411
16.2.6.2 Displaying the measurement history ... 411
16.2.6.3 Displaying consumption values ... 412
16.2.6.4 Comparing usage values ... 413
16.2.6.5 Displaying the long-term measurement of the energy consumption ... 413
16.2.7 Manage measurements ... 414

16.3 Ctrl-E profiles ... 416
16.3.1 Creating and editing energy-saving profiles ... 416
16.3.2 Creating and editing energy-saving profiles ... 419
16.3.3 Using the energy-saving profile ... 422

17 Editing the PLC user program ... 425

17.1 Introduction ... 425

17.2 Displaying and editing PLC properties ... 426
17.2.1 Displaying PLC properties ... 426
17.2.2 Resetting the processing time ... 426
17.2.3 Loading modified PLC user program ... 426

17.3 Displaying and editing PLC and NC variables ... 428

17.4 Displaying and editing PLC signals in the status list ... 434

17.5 View of the program blocks ... 435
17.5.1 Displaying information on the program blocks ... 435
17.5.2 Structure of the user interface ... 436
17.5.3 Control options ... 437
17.5.4 Displaying the program status ... 438
17.5.5 Changing the address display ... 439
17.5.6 Enlarging/reducing the ladder diagram ... 439
17.5.7 Program block ... 440
17.5.7.1 Displaying and editing the program block ... 440
17.5.7.2 Displaying local variable table ... 441
17.5.7.3 Creating a program block ... 441
17.5.7.4 Opening a program block in the window ... 443
17.5.7.5 Displaying/canceling the access protection ... 443
17.5.7.6 Editing block properties subsequently ... 444
17.5.8 Editing a program block ... 444
17.5.8.1 Editing the PLC user program ... 444
17.5.8.2 Editing a program block ... 445
17.5.8.3 Deleting a program block ... 447
17.5.8.4 Inserting and editing networks ... 447
17.5.8.5 Editing network properties ... 448
17.5.9 Displaying the network symbol information table ... 449

17.6 Displaying symbol tables ... 451

17.7 Displaying cross references ... 452

17.8 Searching for operands ... 454

18 Service and diagnostics 455

18.1 Alarm, error, and system messages 455
18.1.1 Displaying alarms 455
18.1.2 Displaying an alarm log 457
18.1.3 Displaying messages 458
18.1.4 Sorting, alarms, faults and messages 459

18.2 PLC and NC variables 460
18.2.1 Displaying and editing PLC and NC variables 460
18.2.2 Saving and loading screen forms 465

18.3 Accessing PLC data 466

18.4 Creating screenshots 467

18.5 Version 468
18.5.1 Displaying version data 468
18.5.2 Save information 469

18.6 Logbook 471
18.6.1 Displaying and editing the logbook 471
18.6.2 Making a logbook entry 472

18.7 Remote diagnostics 474
18.7.1 Setting remote access 474
18.7.2 Permit modem 475
18.7.3 Request remote diagnostics 476
18.7.4 Exit remote diagnostics 477

Index 479

1 Introduction

1.1 About SINUMERIK

From simple, standardized CNC machines to premium modular machine designs – the SINUMERIK CNCs offer the right solution for all machine concepts. Whether for individual parts or mass production, simple or complex workpieces – SINUMERIK is the highly dynamic automation solution, integrated for all areas of production. From prototype construction and tool design to mold making, all the way to large-scale series production.

Visit our website for more information SINUMERIK (https://www.siemens.com/sinumerik).

1.2 About this documentation

Target group

This documentation is intended for users of universal machines running the SINUMERIK Operate software.

Benefits

The Operating Manual helps users familiarize themselves with the control elements and commands. Guided by the manual, users are capable of responding to problems and taking corrective action.

Standard scope

This documentation only describes the functionality of the standard version. This may differ from the scope of the functionality of the system that is actually supplied. Please refer to the ordering documentation only for the functionality of the supplied drive system.

It may be possible to execute other functions in the system which are not described in this documentation. This does not, however, represent an obligation to supply such functions with a new control or when servicing.

For reasons of clarity, this documentation cannot include all of the detailed information on all product types. Further, this documentation cannot take into consideration every conceivable type of installation, operation and service/maintenance.

The machine manufacturer must document any additions or modifications they make to the product themselves.

Websites of third-party companies

This document may contain hyperlinks to third-party websites. Siemens is not responsible for and shall not be liable for these websites and their content. Siemens has no control over the information which appears on these websites and is not responsible for the content and information provided there. The user bears the risk for their use.

1.3 Documentation on the internet

1.3.1 Documentation overview SINUMERIK ONE

Comprehensive documentation about the functions provided in SINUMERIK ONE Version 6.13 and higher is provided in the Documentation overview SINUMERIK ONE (https://support.industry.siemens.com/cs/ww/en/view/109768483).



You can display documents or download them in PDF and HTML5 format.

The documentation is divided into the following categories:

- User: Operating
- User: Programming
- Manufacturer/Service: Functions
- Manufacturer/Service: Hardware
- Manufacturer/Service: Configuring/Commissioning
- Manufacturer/Service: Safety Integrated
- Information and training
- Manufacturer/Service: SINAMICS

1.3.2 Documentation overview SINUMERIK operator components

Comprehensive documentation about the SINUMERIK operator components is provided in the Documentation overview SINUMERIK operator components (https://support.industry.siemens.com/cs/document/109783841/technische-dokumentation-zu-sinumerik-bedienkomponenten?dti=0&lc=en-WW).

You can display documents or download them in PDF and HTML5 format.

The documentation is divided into the following categories:

- Operator panels
- Handheld unit/Mini handheld devices
- Machine control panels
- Further operator components

An overview of the most important documents, entries and links on the topic of "SINUMERIK" is provided at SINUMERIK Overview - Topic Page (https://support.industry.siemens.com/cs/document/109766201/sinumerik-an-overview-of-the-most-important-documents-and-links?dti=0&lc=en-WW).

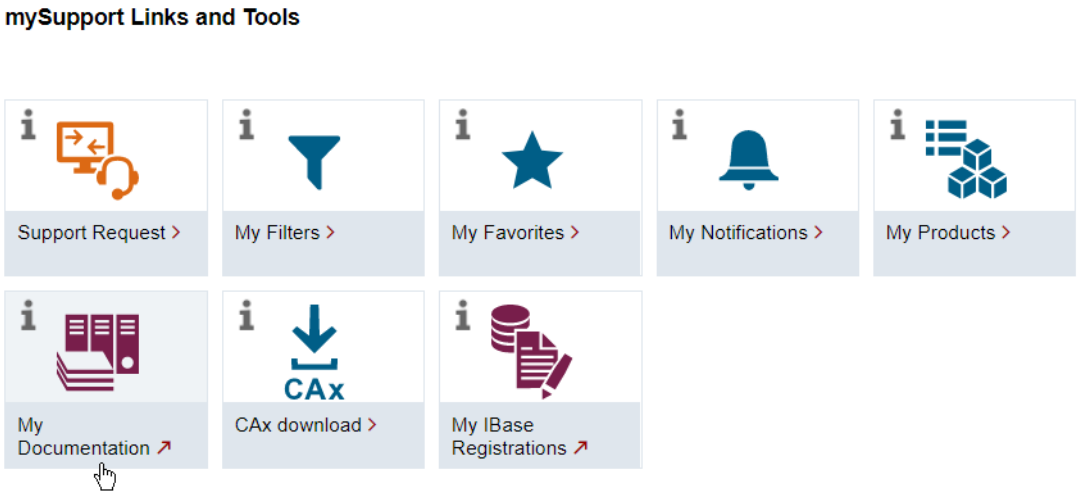
1.4 Feedback on the technical documentation

If you have any questions, suggestions, or corrections regarding the technical documentation published in the Siemens Industry Online Support, use the "Provide feedback" link which appears at the end of the entry.

1.5 mySupport documentation

With the "mySupport documentation" web-based system you can compile your own individual documentation based on Siemens content, and adapt it for your own machine documentation.

To start the application, click on the "My Documentation" tile on the "mySupport links and tools" (https://support.industry.siemens.com/cs/ww/en/my) portal page:



The configured manual can be exported in RTF, PDF or XML format.

Note

Siemens content that supports the mySupport documentation application can be identified by the presence of the "Configure" link.

1.6 Service and Support

Product support

You can find more information about products on the internet:

Product support (https://support.industry.siemens.com/cs/ww/en/)

The following is provided at this address:

- Up-to-date product information (product announcements)
- FAQs (frequently asked questions)
- Manuals
- Downloads
- Newsletters with the latest information about your products
- Global forum for information and best practice sharing between users and specialists
- Local contact persons via our Contacts at Siemens database (→ "Contact")
- Information about field services, repairs, spare parts, and much more (→ "Field Service")

Technical support

Country-specific telephone numbers for technical support are provided on the internet at address (https://support.industry.siemens.com/cs/ww/en/sc/4868) in the "Contact" area.

If you have any technical questions, please use the online form in the "Support Request" area.

Training

You can find information on SITRAIN at the following address (https://www.siemens.com/sitrain).
SITRAIN offers training courses for automation and drives products, systems and solutions from Siemens.

Siemens support on the go



With the award-winning "Industry Online Support" app, you can access more than 300,000 documents for Siemens Industry products – any time and from anywhere. The app can support you in areas including:

- Resolving problems when implementing a project
- Troubleshooting when faults develop
- Expanding a system or planning a new system

Furthermore, you have access to the Technical Forum and other articles from our experts:

- FAQs
- Application examples
- Manuals
- Certificates
- Product announcements and much more

The "Industry Online Support" app is available for Apple iOS and Android.

1.7 Using OpenSSL

This product can contain the following software:

- Software developed by the OpenSSL project for use in the OpenSSL toolkit
- Cryptographic software created by Eric Young.
- Software developed by Eric Young

You can find more information on the internet:

- OpenSSL (https://www.openssl.org)
- Cryptsoft (https://www.cryptsoft.com)

1.8 Compliance with the General Data Protection Regulation

Overview

Siemens observes standard data protection principles, in particular the data minimization rules (privacy by design).

For the SINUMERIK Operate, this means:

The product processes/saves the following personal data:

- FullName (optional): Only if user management is activated
- User name + password: Only if user management is activated
- UserID: Only if user management is activated
- IP address
- Security events
- Time stamp

It does not involve data from the personal or private sphere.

The above data is required for the user log-in function. The storage of data is appropriate and limited to what is necessary, as it is essential for the identification of the authorized operator. (Mandatory here are: user name + password; the FullName is optional.)

The above-mentioned data cannot be stored anonymously or – with the exception of the user name – pseudonymously, as otherwise the purpose of identifying the operating personnel cannot be achieved.

Our products do not automatically delete the data mentioned above. The data and logs can be deleted manually by authorized personnel.

The above data is secured against loss of integrity and confidentiality by Industry State-of-the-Art Product Security mechanisms.

Fundamental safety instructions

2

2.1 General safety instructions

⚠ WARNING

Danger to life if the safety instructions and residual risks are not observed

If the safety instructions and residual risks in the associated hardware documentation are not observed, accidents involving severe injuries or death can occur.

- Observe the safety instructions given in the hardware documentation.
- Consider the residual risks for the risk evaluation.

⚠ WARNING

Malfunctions of the machine as a result of incorrect or changed parameter settings

As a result of incorrect or changed parameterization, machines can malfunction, which in turn can lead to injuries or death.

- Protect the parameterization against unauthorized access.
- Handle possible malfunctions by taking suitable measures, e.g. EMERGENCY STOP or EMERGENCY OFF.

2.2 Warranty and liability for application examples

Application examples are not binding and do not claim to be complete regarding configuration, equipment, or any eventuality which may arise. Application examples do not represent customer-specific solutions, but merely serve to provide assistance with typical tasks.

As the user you yourself are responsible for ensuring that the products described are operated correctly. Application examples do not relieve you of your responsibility for safe handling when using, installing, operating and maintaining the equipment.

2.3 Cybersecurity information

Siemens provides products and solutions with industrial cybersecurity functions that support the secure operation of plants, systems, machines and networks.

In order to protect plants, systems, machines and networks against cyber threats, it is necessary to implement – and continuously maintain – a holistic, state-of-the-art industrial cybersecurity concept. Siemens' products and solutions constitute one element of such a concept.

Customers are responsible for preventing unauthorized access to their plants, systems, machines and networks. Such systems, machines and components should only be connected to an enterprise network or the internet if and to the extent such a connection is necessary and only when appropriate security measures (e.g. firewalls and/or network segmentation) are in place.

For additional information on industrial cybersecurity measures that may be implemented, please visit
https://www.siemens.com/cybersecurity-industry.

Siemens' products and solutions undergo continuous development to make them more secure. Siemens strongly recommends that product updates are applied as soon as they are available and that the latest product versions are used. Use of product versions that are no longer supported, and failure to apply the latest updates may increase customer's exposure to cyber threats.

To stay informed about product updates, subscribe to the Siemens Industrial Cybersecurity RSS Feed under
https://new.siemens.com/cert.

Further information is provided on the Internet:

Configuration Manual Industrial Cybersecurity (https://support.industry.siemens.com/cs/ww/en/view/109975311)

⚠ WARNING

Unsafe operating states resulting from software manipulation

Software manipulations, e.g. viruses, Trojans, or worms, can cause unsafe operating states in your system that may lead to death, serious injury, and property damage.

- Keep the software up to date.
- Incorporate the automation and drive components into a state-of-the-art, integrated industrial cybersecurity concept for the installation or machine.
- Make sure that you include all installed products in the integrated industrial cybersecurity concept.
- Protect files stored on exchangeable storage media from malicious software by with suitable protection measures, e.g. virus scanners.
- Carefully check all cybersecurity-related settings once commissioning has been completed.

Fundamentals

3

3.1 Product overview

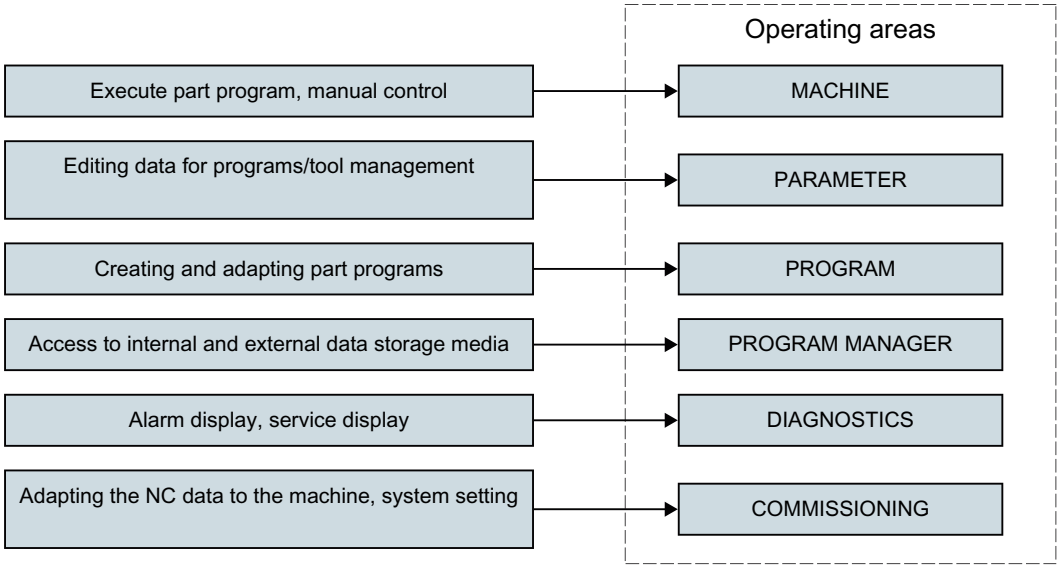
The SINUMERIK control system is a CNC (Computerized Numerical Control) for machine tools.

You can use the CNC to implement the following basic functions in conjunction with a machine tool:

- Create can adapt part programs
- Execute part programs
- Manual control
- Access internal and external data media
- Edit data for programs
- Manage tools, zero points and further user data required in programs
- Diagnose control system and machine

Operating areas

The basic functions are grouped in the following operating areas in the control:

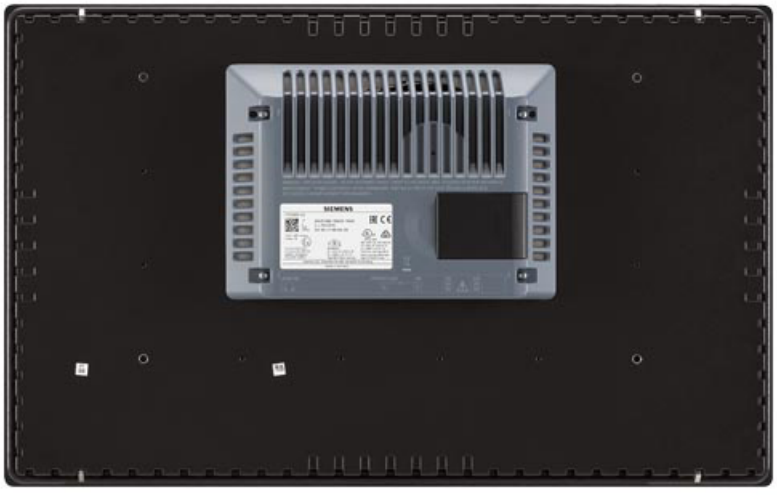


3.2 Operator components

3.2.1 Touch panels

The SIMATIC ITC2200 V3 Industrial Thin Client can be used to display and use the user interface of SINUMERIK Operate:





The "SINUMERIK Operate Generation 2" user interface has been optimized for multi-touch operation. You can execute all actions by touch and finger gestures. Using SINUMERIK Operate is much quicker with touch operation and finger gestures.

Machine manufacturer

Please observe the information provided by the machine manufacturer.

The following operator panel fronts, handheld devices and SINUMERIK control systems can be operated with the "SINUMERIK Operate Generation 2" user interface:

- SIMATIC HMI Industrial Thin Clients ITC V3, ITC V3 PRO firmware V3.2 (https://support.industry.siemens.com/cs/ww/en/view/109801145)
- SIMATIC HMI Industrial Flat Panels V2 IFP V2, IFP V2 PRO, IFP V2 ETH (https://support.industry.siemens.com/cs/ww/en/view/109767495)
- Handheld Terminal HT 10 (https://support.industry.siemens.com/cs/ww/en/view/109782248)

More information on multi-touch operation can be found under Multi-touch operation (Page 65).

3.2.2 Machine control panels

3.2.2.1 Overview

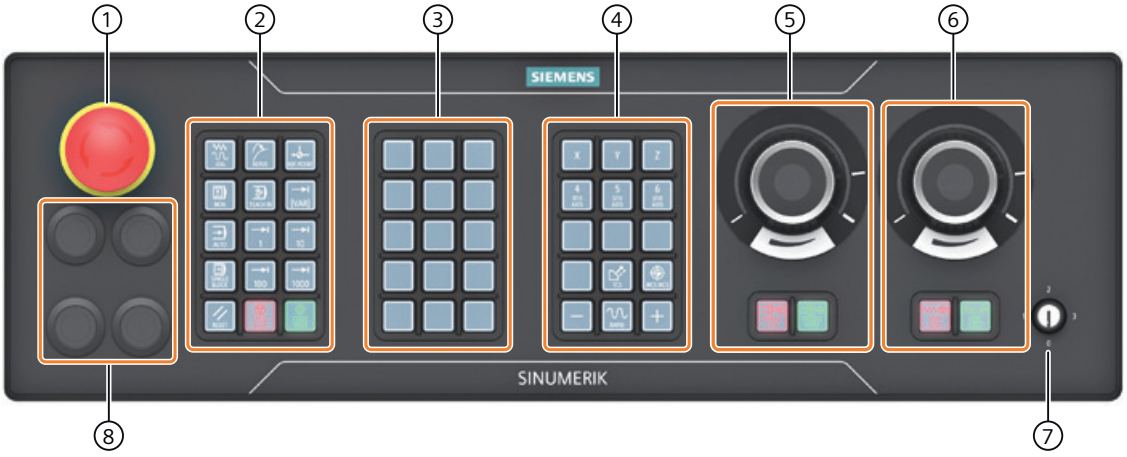
The machine tool can be equipped with a machine control panel by Siemens or with a specific machine control panel from the machine manufacturer.

You use the machine control panel to initiate actions on the machine tool such as traversing an axis or starting the machining of a workpiece.

3.2.2.2 Controls on the machine control panel

In this example, the MCP 2200c machine control panel is used to illustrate the operator controls and displays of a Siemens machine control panel.

Overview



① EMERGENCY STOP button
② Keypad 1 (operating mode block)
③ Keypad 2 (customer keys)
④ Keypad 3 (axis block)
⑤ Powerride 2 (spindle) + keypad 3.2
⑥ Powerride 1 (feed) + keypad 3.1
⑦ Key-operated authorization switch
⑧ 4x mounting space for 22.5 mm elements with slide-in labels

Figure 3-1 Operator controls

Labeling of the keys

The keys are designed with replaceable caps for machine-specific adaptations. The key caps can be freely inscribed using a laser. Alternatively, you can use transparent key caps and insert labels.

Operator controls

EMERGENCY STOP button



Press the button in situations where:

- life is at risk.
- there is the danger of a machine or workpiece being damaged.

All drives will be stopped with the greatest possible braking torque.



Machine manufacturer

For additional responses to pressing the EMERGENCY STOP button, please refer to the machine manufacturer's instructions.

RESET



- Stop processing the current programs.
  The NCK control remains synchronized with the machine. It is in its initial state and ready for a new program run.
- Cancel alarm.

Program control



<SINGLE BLOCK>
Single block mode on/off.



<CYCLE START>
The key is also referred to as NC Start.
Execution of a program is started.



<CYCLE STOP>
The key is also referred to as NC Stop.
Execution of a program is stopped.

Operating modes, machine functions



<JOG>
Select "JOG" mode.



<TEACH IN>
Selecting the "Teach In" function



<MDI>
Select "MDI" mode.



<AUTO>
Select "AUTO" mode.



<REPOS>
Repositions, re-approaches the contour.



<REF POINT>
Approach reference point.



Inc <VAR> (Incremental Feed Variable)
Incremental mode with variable increment size.



Inc (incremental feed)
Incremental mode with predefined increment size of 1, ..., 10000 increments.

...





Machine manufacturer
A machine data code defines how the increment value is interpreted.

Traversing axes with rapid traverse override and coordinate switchover



Axis keys
Selects an axis.

...





Direction keys
Select the traversing direction.

...





<RAPID>
Traverse axis in rapid traverse while pressing the direction key.



<WCS MCS>
Switches between the workpiece coordinate system (WCS) and machine coordinate system (MCS).

Spindle control with override switch



<SPINDLE STOP>
Stop spindle.



<SPINDLE START>
Spindle is enabled.

Feed control with override switch



<FEED STOP>

Stops execution of the running program and shuts down axis drives.



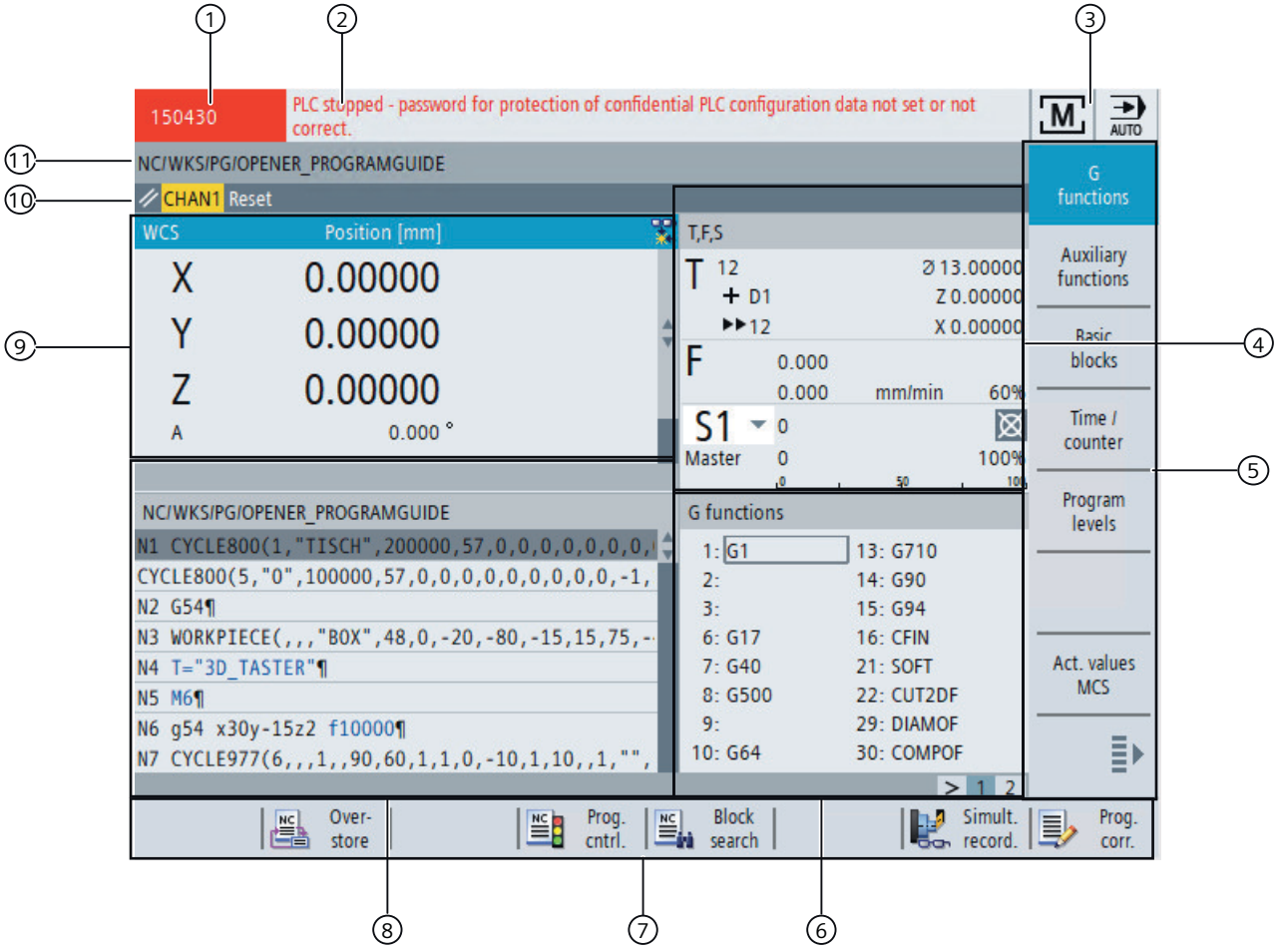
<FEED START>

Enable for program execution in the current block and enable for ramp-up to the feedrate value specified by the program.

3.3 Operator interface

3.3.1 Screen layout

Overview



① Alarm number
② Alarm/message line
③ Active operating area and mode
④ Display for

- Active tool T
- Current feedrate F
- Active spindle with current status (S)
- Spindle utilization rate in percent
- Name of active tool holder with display of a rotation in space and plane
- Name of the active kinematic transformation

⑤ Vertical softkey bar
⑥ Display of active G functions, all G functions, auxiliary functions and input window for different functions (for example, skip blocks, program control).
⑦ Horizontal softkey bar
⑧ Operating window with program block display
⑨ Position display of the axes in the actual values window
⑩ Channel state and program control
⑪ Program name

3.3.2 Status display

The status display includes the most important information about the current machine status and the status of the NCK. It also shows alarms as well as NC and PLC messages.

Depending on your operating area, the status display is made up of several lines:

- Large status display
  The status display is made up of three lines in the "Machine" operating area.
- Small status display
  In the "Parameter", "Program", "Program Manager", "Diagnosis" and "Start-up" operating areas, the status display consists of the first line from the large display.

Status display of "Machine" operating area

First line

Ctrl-Energy - power display

Display	Meaning
	The machine is not productive.
	The machine is productive and energy is being consumed.
	The machine is feeding energy back into the supply system.
The power display must be switched on in the status line. **Additional information** on configuring is provided in the Ctrl-Energy System Manual.	

Active operating area

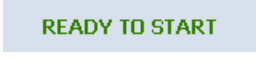
Display	Meaning
	"Machine" operating area In touch mode you can switch over the operating area here.
	"Parameter" operating area
	"Program" operating area

Display	Meaning
	"Program manager" operating area
	"Diagnostics" operating area
	"Startup" operating area

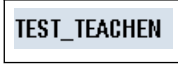
Active mode or function

Display	Meaning
	"Jog" mode
	"MDA" mode
	"AUTO" mode
	"TEACH IN" function
	"REPOS" function
	"REF POINT" function

Alarms and messages

Display	Meaning
	Alarm display The alarm numbers are displayed in white lettering on a red background. The associated alarm text is shown in red lettering. An arrow indicates that several alarms are active. An acknowledgment icon shows how to acknowledge or delete the alarm.
	NC or PLC message Message numbers and texts are shown in black lettering. An arrow indicates that several messages are active.
	Messages from NC programs do not have numbers and appear in green lettering.

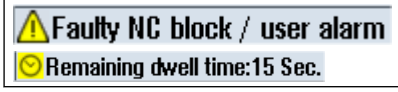
Second line

Display	Meaning
	Program path and program name

The displays in the second line can be configured.

Machine manufacturer
Please observe the information provided by the machine manufacturer.

Third line

Display	Meaning
	Channel status display. If the machine has several channels, then the channel name is also displayed. If there is only one channel, then only "Reset" is displayed as channel status. In touch mode you can switch over the channel here
	Channel status display: The program was canceled with "Reset". The program is executed. The program was interrupted with "Stop".
	Display of active program controls: PRT No axis motion DRY Dry run feedrate RG0: reduced rapid traverse M01: programmed stop 1 M101: programmed stop 2 (the designation is variable) SB1: Single block, coarse (program stops only after blocks that perform a machine function) SB2: Calculation block (program stops after each block) SB3: Single block, fine (program also only stops after blocks which perform a machine function in cycles) CST: configured stop (program stops at stop-relevant locations, which you defined before the program starts)
	Channel operational messages: Stop: An operator action is generally required. Wait: No operator action is required.

The machine manufacturer settings determine which program controls are displayed.

Machine manufacturer
Please observe the information provided by the machine manufacturer.

3.3.3 Actual value window

The actual values of the axes and their positions are displayed.

Work/Machine

The displayed coordinates are based on either the machine coordinate system or the workpiece coordinate system. The machine coordinate system (Machine), in contrast to the workpiece coordinate system (Work), does not take any work offsets into consideration.

You can use the "Machine actual values" softkey to toggle between the machine coordinate system and the workpiece coordinate system.

The actual value display of the positions can also refer to the SZS coordinate system (settable zero system). However the positions are still output in the Work.

The SZS coordinate system corresponds to the Work coordinate system, reduced by certain components ($P_TRAFRAME, $P_PFRAME, $P_ISO4FRAME, $P_CYCFRAME), which are set by the system when machining and are then reset again. By using the SZS coordinate system, jumps into the actual value display are avoided that would otherwise be caused by the additional components.



Machine manufacturer

Observe the information provided by the machine OEM.

Maximize display



Press the ">>" and "Zoom act. val." softkeys.



Display overview

Display	Meaning
Header columns	
Work/Machine	Display of axes in selected coordinate system.
Position	Position of displayed axes.
Display of distance-to-go	The distance-to-go for the current NC block is displayed while the program is running. When a compressor function is active, polynomial blocks are generated that are then taken into account in the distance to go display instead of the original blocks.
Feed/override	The feed acting on the axes, as well as the override, are displayed in the full-screen version.
REPOS offset	The distances traversed in manual mode are displayed. This information is only displayed when you are in the "REPOS" function.

Display	Meaning	
Collision avoidance		Collision avoidance is activated for JOG, MDI and AUTO modes.
		Collision avoidance is deactivated for JOG, MDI and AUTO modes.
Footer	Display of active work offsets and transformations. The T, F, S values are also displayed in the full-screen version.	

See also

Zero offsets (Page 99)

Set collision avoidance (Page 258)

3.3.4 T,F,S window

The most important data concerning the current tool, the feedrate (path feed or axis feed in JOG) and the spindle is displayed in the T, F, S window.

In addition to the "T, F, S" window name, the following information is also displayed:

Display	Meaning
BC (example)	Name of the tool carrier
Turning (example)	Name of the active kinematic transformation
	Active tool carrier rotated in the plane
	Active tool carrier swiveled in space

Tool data

Display	Meaning
T	
Tool name	Name of the current tool
Location	Location number of the current tool
D	Cutting edge of the current tool The tool is displayed with the associated tool type symbol corresponding to the actual coordinate system in the selected cutting edge position. If the tool is swiveled, then this is taken into account in the display of the cutting edge position. In DIN-ISO mode the H number is displayed instead of the cutting edge number.
H	H number (tool offset data record for DIN-ISO mode) If there is a valid D number, this is also displayed.

Display	Meaning
Ø	Diameter of the current tool
R	Radius of the current tool
L	Length of the actual tool
Z	Z value of the current tool
X	X value of the current tool

Feed data

Display	Meaning
F	
	Feed disable
	Actual feed value If several axes traverse, is displayed for: • "JOG" mode: Axis feed for the traversing axis • "MDA" and "AUTO" mode: Programmed axis feed
Rapid traverse	G0 is active
0.000	No feed is active
Override	Display as a percentage

Spindle data

Display	Meaning
S	
S1	Spindle selection, identification with spindle number and main spindle
Speed	Actual value (when spindle turns, display increases) Setpoint (always displayed, also during positioning)
Symbol    	Spindle status Spindle not enabled Spindle is turning clockwise Spindle is turning counterclockwise Spindle is stationary
Override	Display as a percentage
Spindle utilization rate 	Display between 0 and 100% The upper limit value can be greater than 100%. See machine manufacturer's specifications. Display of maximum remaining time of spindle use at the current spindle load. Time and symbol are displayed if the remaining time is less than 120 seconds.

Note

Display of logical spindles

If the spindle converter is active, logical spindles are displayed in the workpiece coordinate system. When switching over to the machine coordinate system, the physical spindles are displayed.

Machine manufacturer

Please observe the information provided by the machine manufacturer.

3.3.5 Current block display

The window of the current block display shows the program blocks currently being executed.

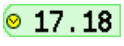
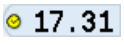
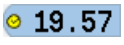
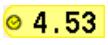
Display of current program

The following information is displayed in the running program:

- The workpiece name or program name is entered in the header line.
- The program block which is just being processed appears colored.

Display of the machining times

If you set that the machining times are to be recorded in the settings for automatic mode, the measured times are shown at the end of the line as follows:

Display	Meaning
Light green background 	Measured machining time of the program block (automatic mode)
Green background 	Measured machining time of the program group (automatic mode)
Light blue background 	Estimated machining time of the program block (simulation)
Blue background 	Estimated machining time of the program group (simulation)
Yellow background 	Wait time (automatic mode or simulation)

Highlighting of selected G code commands or keywords

In the program editor settings, you can specify whether selected G code commands are to be highlighted in color. The following colors are used as standard:

Display	Meaning
Blue font M30¶	D, S, F, T, M and H functions
Red font G0¶	"G0" motion command
Green font G1¶	"G1" motion command
Blue-green font G3¶	"G2" or "G3" motion command
Gray font ; Kommentar¶	Comment

Machine manufacturer

You can define further highlight colors in the "sleditorwidget.ini" configuration file.
Please refer to the machine manufacturer's instructions.

Editing a program directly

In the Reset state, you can edit the current program directly.



1. Press the <INSERT> key.
2. Place the cursor at the relevant position and edit the program block.
   Direct editing is only possible for G code blocks in the NC memory, not for external execution.



3. Press the <INSERT> key to exit the program and the edit mode again.

See also

Correcting a program (Page 138)

Settings for the automatic mode (Page 187)

3.3.6 Operation via softkeys and buttons

Operating areas/operating modes

The user interface consists of different windows featuring eight horizontal and eight vertical softkeys.

You operate the softkeys with the keys next to the softkey bars.

You can display a new window or execute functions using the softkeys.

The operating software is sub-divided into six operating areas (machine, parameter, program, program manager, diagnosis, startup), three operating modes and four functions (JOG, MDI, AUTO, TEACH IN, REF. POINT, REPOS, single block).

Changing the operating area



Press the <MENU SELECT> key and select the desired operating area using the horizontal softkey bar.

You can call the "Machine" operating area directly using the key on the operator panel.



Press the <MACHINE> key to select the "machine" operating area.

Changing the operating mode

You can select a mode or function directly with the keys on the machine control panel or the vertical softkeys in the main menu.

General keys and softkeys



When the > symbol appears to the right of the dialog line on the user interface, you can change the horizontal softkey bar within an operating area. To do so, press the menu forward key.

The > symbol indicates that you are in the expanded softkey bar.

Pressing the key again will take you back to the original horizontal softkey bar.



Use the ">>" softkey to open a new vertical softkey bar.



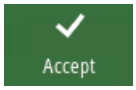
Use the "<<" softkey to return to the previous vertical softkey bar.



Use the "Return" softkey to close an open window.



Use the "Cancel" softkey to exit a window without accepting the entered values and return to the next highest window.



When you have entered all the necessary parameters in the parameter screen form correctly, you can close the window and save the parameters using the "Accept" softkey. The values you entered are applied to a program.



Use the "OK" softkey to initiate an action immediately, e.g. to rename or delete a program.

See also

Channel switchover (Page 93)

3.3.7 Key combinations on a PC keyboard

You can use the following combinations on an external PC keyboard:

Key combination on a PC keyboard	Operate function/equivalent key on a SINUMERIK-OP	Meaning
<SHIFT+F11>	Position	In operating area "Machine", switches to the actual position display
<END>[1)]	Program	Switches into operating area "Program Manager"
<Page DOWN>[1)]	Offset	Switches into operating area "Parameter > Work offset"
<HOME>[1)]	Prg.Manager	Calls the Program Manager
<Page Up>[1)]	Alarm	Calls operating area "Diagnostics"
<SHIFT+F12> or cursor down[1)]	Custom	Switches into operating area "Custom" - user-specific user interface
<SHIFT+F10>	M Machine	Calls the "Machine" operating area.
<F10>	Menu Select	Calls the main menu to select the operating area
<F12>	Help	Calls the context-sensitive online help for the selected window or machine data
<ESC>	ALARM CANCEL	Clears alarms and messages that are marked with the special symbol
<Shift> + <F9>	>	Menu forward key Advances in the extended horizontal softkey bar.
<F9>	^	Menu back key Returns to the higher-level menu.
<F11>	1...n CHANNEL	Switches further for multiple channels

1) Key is located on numeric keypad: <NumLock> must be deactivated.

3.3.8 Entering or selecting parameters

When setting up the machine and during programming, you must enter various parameter values in the entry fields. The background color of the fields provides information on the status of the entry field.

Orange background	The input field is selected
Light orange background	The input field is in edit mode
Pink background	The entered value is incorrect

Selecting parameters

Some parameters require you to select from a number of options in the input field. Fields of this type do not allow you to type in a value.

The selection symbol is displayed in the tooltip: 

Associated selection fields

There are selection fields for various parameters:

- Selection of units
- Changeover between absolute and incremental dimensions

Procedure



1. Keep pressing the <SELECT> key until the required setting or unit is selected.

   The <SELECT> key only works if there are several selection options available.

   - OR -



   Press the <INSERT> key.

   The selection options are displayed in a list.





2. Select the required setting using the <Cursor down> and <Cursor up> keys.

3. If required, enter a value in the associated input field.



4. Press the <INPUT> key to complete the parameter input.

Changing or calculating parameters

If you only want to change individual characters in an input field rather than overwriting the entire entry, switch to insertion mode.

In this mode, you can also enter simple calculation expressions, without having to explicitly call the calculator.

Note

Functions of the calculator

Function calls of the calculator are not available in the parameter screens of the cycles and functions in the "Program" operating area.



Press the <INSERT> key.

The insert mode is activated.





You can navigate within the input field using the <Cursor left> and <Cursor right> keys.



Use the <BACKSPACE> and <DEL> key to delete individual characters.



Enter the value or the calculation.



Close the value entry using the <INPUT> key and the result is transferred into the field.

Accepting parameters

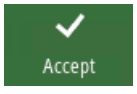
When you have correctly entered all necessary parameters, you can close the window and save your settings.

You cannot accept the parameters if they are incomplete or obviously erroneous. In this case, you can see from the dialog line which parameters are missing or were entered incorrectly.



Press the "OK" softkey.

- OR -



Press the "Accept" softkey.

3.3.9 Pocket calculator

The calculator allows you to calculate values for entry fields. It is possible to choose between a simple standard calculator and the extended view with mathematical functions.

Using the calculator

- You can simply use the calculator at the touch panel.
- Without a touch panel, you can use the calculator using the mouse.

Procedure

1. Position the cursor on the desired entry field.
2. 

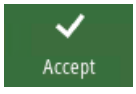
Press the <=> key.
The calculator is displayed.
3. 

Press the <min> key if you would like to work with the standard calculator.
- OR -


Press the <extend> key to switch to the extended view.
4. Input the arithmetic statement.
You can use functions, arithmetic symbols, numbers, and commas.
5. 

Press the equals symbol on the calculator.
- OR -


Press the "Calculate" softkey.
- OR -


Press the <INPUT> key.
The new value is calculated and displayed in the entry field of the calculator.
6. 

Press the "Accept" softkey.
The calculated value is accepted and displayed in the entry field of the window.

See also

Entering or selecting parameters (Page 45)

3.3.10 Pocket calculator functions

The called operations continue to be displayed in the entry field of the calculator until the value is calculated. This allows you to subsequently modify entries and to nest functions.

The following save and delete functions are provided for modifications:

Key	Function
MS	Buffer value (Memory Save)
MR	Retrieve from buffer memory (Memory Recall)
MC	Delete buffer memory contents (Memory Clear)
<--	Delete individual character (Backspace)
CE	Delete expression (Clear Element)
C	Delete all entries (Clear)

Nesting functions

Various possibilities are available for the nesting of functions as follows:

- Position the cursor within the bracket of the function call and supplement the argument with an additional function.
- Highlight the expression which is to be used as an argument in the entry line and then press the desired function key.

Percentage calculation

The calculator supports the calculation of a percentage, as well as changing of a basic value by a percentage. Press the following keys in this regard:

Example: Percentage

4 * 50 % = 2

Example: Change by percentage

4 + 50 % = 6

Calculating trigonometric functions

1. Check whether the angles are specified in radians "RAD" or in degrees "DEG".

RAD

2. Press the "RAD" key to calculate the trigonometric functions in degrees "DEG".
   The designation of the key changes to "DEG".

   - OR -

DEG

Press the "DEG" key to calculate the trigonometric functions in radian.
The designation of the key changes to "RAD".

SIN

3. Press the key for the desired trigonometric function, e.g. "SIN".
4. Enter the numerical value.

...

ATAN

Further mathematical functions

Press the keys in the specified order:

Square number

x^2 S Number

Square root

$\sqrt{x}$ R Number

Exponential function

Base number EXP Exponent

Residue class calculation

Number MOD Divider

Absolute value

ABS Number

Integer component

INT Number

Conversion between millimeters and inches

1. Enter the numerical value.

MM

2. Press the "MM" key to convert inches to millimeters.
The key is highlighted in blue.

- OR -

INCH

Press the "INCH" key to convert millimeters to inches.
The button is highlighted in blue.

=

3. Press the "=" key on the calculator.

The calculated value is displayed in the entry field. The key for the unit is highlighted in gray once again.

3.3.11 Context menu

When you right-click, the context menu opens and provides the following functions:

- Cut
  Cut Ctrl+X
- Copy
  Copy Ctrl+C
- Paste
  Paste Ctrl+V

Program editor

Additional functions are available in the editor

- Undo the last change
  Undo Ctrl+Z
- Redo the changes that were undone
  Redo Ctrl+Y

Up to 50 changes can be undone.

3.3.12 Changing the user interface language

Procedure



1. Select the "Start-up" operating area.



2. Press the "Change language" softkey.
   The "Language selection" window opens. The language set last is selected.
3. Position the cursor on the desired language.



4. Press the "OK" softkey.

   - OR -



   Press the <INPUT> key.

   The user interface changes to the selected language.

Note

Changing the language directly on the input screens

You can switch between the user interface languages available on the controller directly on the user interface by pressing the key combination <CTRL + L>.

3.3.13 Entering Chinese characters

3.3.13.1 Function - input editor

Using the input editor IME (Input Method Editor), you can select Asian characters on classic panels (without touch operation) where you enter the phonetic notation. These characters are transferred into the user interface.

Note

Call the input editor with <Alt + S>

The input editor can only be called there where it is permissible to enter Asian characters.

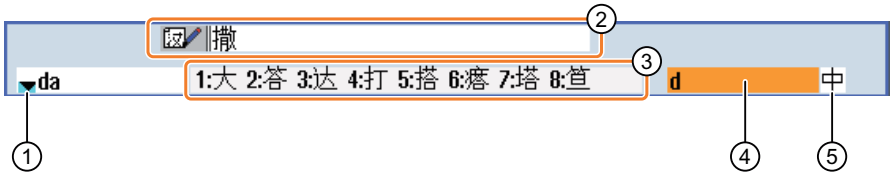
The editor is available for the following Asian languages:

- Simplified Chinese
- Traditional Chinese

Input types

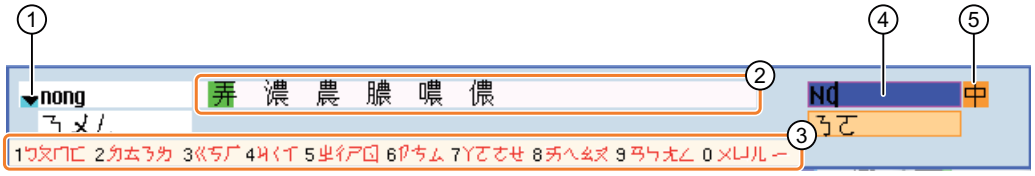
Input type	Description
Pinyin input	Latin letters are combined phonetically to denote the sound of the character. The editor lists all of the characters from the dictionary that can be selected.
Zhuyin input (only traditional Chinese)	Non-Latin letters are combined phonetically to denote the sound of the character. The editor lists all of the characters from the dictionary that can be selected.
Entering Latin letters	The characters that are entered are directly transferred into the input field, from where the editor was called.

Structure of the editor



① Phonetic sound selection from the dictionary
② Learning function of the dictionary
③ Listed characters
④ Phonetic sound input
⑤ Function selection

Figure 3-2 Example: Pinyin input



① Phonetic sound selection from the dictionary
② Listed characters (for the input field)
③ Listed characters (for phonetic sound input)
④ Phonetic sound input
⑤ Function selection

Figure 3-3 Example: Zhuyin input

Functions

中 Pinyin input

A Entering Latin letters

Editing the dictionary

Dictionaries

The simplified Chinese and traditional Chinese dictionaries that are supplied can be expanded:

- If you enter new phonetic notations, the editor creates a new line. The entered phonetic notation is broken down into known phonetic notations. Select the associated character for each component. The compiled characters are displayed in the additional line. Accept the new word into the dictionary and into the input field by pressing the <Input> key.
- Using any Unicode editor, you can enter new phonetic notations into a text file. These phonetic notations are imported into the dictionary the next time that the input editor is started.

3.3.13.2 Entering Asian characters

Precondition

The control has been switched over to Chinese.

Procedure

Editing characters using the Pinyin method



+



1. Open the screen form and position the cursor on the input field.
   Press the <Alt +S> keys.
   The editor is displayed.
2. Enter the desired phonetic notation using Latin letters. Use the upper input field for traditional Chinese.



3. Press the <Cursor down> key to reach the dictionary.
4. Keeping the <Cursor down> key pressed, displays all the entered phonetic notations and the associated selection characters.



5. Press the <BACKSPACE> softkey to delete entered phonetic notations.
6. Press the number key to insert the associated character.
   When a character is selected, the editor records the frequency with which it is selected for a specific phonetic notation and offers this character at the top of the list when the editor is next opened.

Editing characters using the Zhuyin method (only traditional Chinese)



+



1. Open the screen form and position the cursor on the input field.
   Press the <Alt +S> keys.
   The editor is displayed.
2. Enter the desired phonetic notation using the numerical block.
   Each number is assigned a certain number of letters that can be selected by pressing the numeric key one or several times.



3. Press the <Cursor down> key to reach the dictionary.
4. Keeping the <Cursor down> key pressed, displays all the entered phonetic notations and the associated selection characters.



5. Press the <BACKSPACE> softkey to delete entered phonetic notations.





6. To select the associated character, press the <cursor right> or <cursor left> keys.



7. Press the <input> key to enter the character.

3.3.13.3 Editing the dictionary

Learning function of the input editor

Requirement:

The control has been switched over to Chinese.

An unknown phonetic notation has been entered into the input editor.

1. The editor provides a further line in which the combined characters and phonetic notations are displayed.

   The first part of the phonetic notation is displayed in the field for selecting the phonetic notation from the dictionary. Various characters are listed for this particular phonetic notation.
2. Press the number key to insert the associated character into the additional line.

   The next part of the phonetic notation is displayed in the field for selecting the phonetic notation from the dictionary.
3. Repeat step 2 until the complete phonetic notation has been compiled.



   Press the <TAB> key to toggle between the compiled phonetic notation field and the phonetic notation input.



   Compiled characters are deleted using the <BACKSPACE> key.



4. Press the <input> key to transfer the compiled phonetic notation to the dictionary and the input field.

Importing a dictionary

A dictionary can now be generated using any Unicode editor by attaching the corresponding Chinese characters to the pinyin phonetic spelling. If the phonetic spelling contains several Chinese characters, then the line must not contain any additional match. If there are several matches for one phonetic spelling, then these must be specified in the dictionary line by line. Otherwise, several characters can be specified for each line.

The generated file should be saved in the UTF8 format under the name dictchs.txt (simplified Chinese) or dictcht.txt (traditional Chinese).

Line structure:

Pinyin phonetic spelling <TAB> Chinese characters <LF>

OR

Pinyin phonetic spelling <TAB> Chinese character1<TAB> Chinese character2 <TAB> ... <LF>

<TAB> - tab key

<LF> - line break

Store the created dictionary in one of the following paths:

`../user/sinumerik/hmi/ime/`

`../oem/sinumerik/hmi/ime/`

When the Chinese editor is called the next time, it enters the content of the dictionary into the system dictionary.

Example:

ai 哎 哀 唉 埃 挨
caise 彩色
hongse 紅色
huise 灰色
heli 河裏
zuihaowan 最好玩

3.3.14 Entering Korean characters

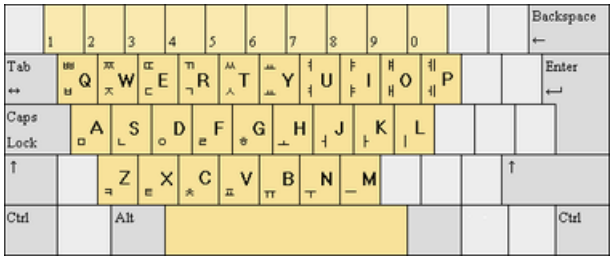
You can enter Korean characters in the input fields on classic panels (without touch operation) using the input editor IME (Input Method Editor).

Note

You require a special keyboard to enter Korean characters. If this is not available, then you can enter the characters using a matrix.

Korean keyboard

To enter Korean characters, you will need a keyboard with the keyboard assignment shown below. In terms of key layout, this keyboard is the equivalent of an English QWERTY keyboard and individual events must be grouped together to form syllables.



Structure of the editor



Functions

Matrix	Editing characters using a matrix
Beolsik 2	Editing characters using the keyboard
한	Entering Korean characters
A	Entering Latin letters

Precondition

The control has been switched over to Korean.

Procedure

Editing characters using the keyboard



+



1. Open the screen form and position the cursor on the input field.
   Press the <Alt +S> keys.
   The editor is displayed.



2. Switch to the "Keyboard - Matrix" selection box.



3. Select the keyboard.



4. Switch to the function selection box.



5. Select Korean character input.

6. Enter the required characters.



7. Press the <input> key to enter the character into the input field.

Editing characters using a matrix



+



1. Open the screen form and position the cursor on the input field.
   Press the <Alt +S> keys.
   The editor is displayed.



2. Switch to the "Keyboard - Matrix" selection box.



3. Select the "matrix".



4. Switch to the function selection box.



5. Select Korean character input.

6. Enter the number of the line in which the required character is located.
   The line is highlighted in color.

7. Enter the number of the column in which the required character is located.
   The character will be briefly highlighted in color and then transferred to the Character field.
   Press the <BACKSPACE> softkey to delete entered phonetic notations.





8. Press the <input> key to enter the character into the input field.

3.3.15 Protection levels

The input and modification of data in the control system is protected by passwords at sensitive places.

Access protection via protection levels

The input or modification of data for the following functions depends on the protection level setting:

- Tool offsets
- Work offsets
- Setting data
- Program creation / program editing

Note

Configuring access levels for softkeys

You have the option of providing softkeys with protection levels or completely hiding them.

Softkeys

As standard, the following softkeys are protected by access levels:

Machine operating area	Access level
	End user (protection level 3)

Parameters operating area	Access level
Tool management lists 	Keyswitch 3 (protection level 4)

Diagnostics operating area	Access level
Logbook	Keyswitch 3 (protection level 4)
Change	End user (protection level 3)
New entry	End user (protection level 3)
Startup complete	Manufacturer (protection level 1)
Machine installed	End user (protection level 3)
Add HW comp.	Service (protection level 2)

Commissioning operating area	Access level
System data	End user (protection level 3)
Setup archive	Keyswitch 3 (protection level 4)
General MD / Control Unit parameter	Keyswitch 3 (protection level 4)
Li-censes	Keyswitch 3 (protection level 4)
Set MD active (cf)	Keyswitch 3 (protection level 4)
Reset (po)	End user (protection level 3)
Change password	End user (protection level 3)
Delete password	End user (protection level 3)

Further information

Additional information on the access levels is provided in the SINUMERIK Operate Commissioning Manual.

3.3.16 Work station safety

In order to secure machines against manipulation and protect people from accidents, proceed as follows when leaving the work station:

1. Set the keyswitch to 0 and then remove it.



2. Press the "Delete password" softkey.
   Access authorization is then initiated.

You can reset the password when you return to the work station.

You can find **more information** about access levels and creating passwords in the SINUMERIK Operate Commissioning Manual.

3.3.17 Cleaning mode

In cleaning mode, you can clean the user interface of the panel without inadvertently initiating touch functions.

When you activate cleaning mode, the system does not respond when you touch the screen. Switching over to another panel and entering data at the keyboard are deactivated. The display is dimmed. The progress bar shows the remaining time in seconds.

Depending on the setting, cleaning mode lasts between 10 seconds and 1 minute. You can work as usual once this time has expired.

Note

Use a suitable cleaning agent to clean the screen.

Procedure



1. Select the "Setup" operating area.



2. Press the "Cleaning mode for panel" softkey.
   The system switches into cleaning mode.

Note

Softkey "Cleaning mode for panel" only functions if the PLC basic program is used.

3.3.18 Display live image from a camera

In SINUMERIK Operate, you can display a live image from a camera.

- The camera image allows you to track remote processes and monitor difficult-to-access areas.
- You can also document and store machine states.
- You can create and configure up to two cameras.



Machine manufacturer
Please observe the information provided by the machine manufacturer.

You configure the cameras in the "Camera configuration" window, which you call with the "Camera" softkey in the "Setup" operating area.

Requirement

- The cameras used meet the required criteria for resolution, frame rate, compression rate and network capability.
- You have configured the cameras used.

Displaying live images from a camera

You have the following options to call up the video stream:

- By clicking the "Camera" softkey in the "Machine" operating area
- Via the "Camera 1" and "Camera 2" widgets in the sidescreen
- Via the "Camera 1" and "Camera 2" apps in the Display Manager

The video stream is displayed in the first-called window.

To switch between the calling windows, the video stream must be restarted.

Note

As long as the "Camera configuration" window is open, the video stream can only be started from this window.

Note

An IPC with a higher performance is required if you wish to use two cameras.

Note

In certain situations, disruptions to the video stream may occur in the Display Manager.

To avoid disruptions, you can reduce the frame rate and resolution.

If these parameters cannot be reduced any further, display the camera images via the "Camera 1" and "Camera 2" widgets in the sidescreen.

For **more information** on the technical requirements of the cameras used, refer to the SINUMERIK Operate online help under "Setup functions and service > Set up camera".

3.3.19 Online help in SINUMERIK Operate

Context-sensitive help is stored in the control system.

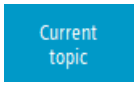
- A brief description is provided for each window and, if required, step-by-step instructions for the operating sequences.
- A detailed help is provided in the editor for every entered G code. You can also display all G functions and take over a selected command directly from the help into the editor.
- A help page with all parameters is provided on the input screen in the cycle programming.
- Lists of the machine data
- Lists of the setting data
- Lists of the drive parameters
- List of all alarms

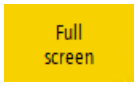
Procedure

Calling context-sensitive help



1. You are in an arbitrary window of an operating area.
2. Press the <HELP> key or on an MF2 keyboard, the <F12> key.
   The help page of the currently selected window is opened in a subscreen.





3. Press the "Full screen" softkey to use the entire user interface to display the help.



Press the "Full screen" softkey again to return to the subscreen.

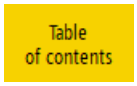


4. If further help is offered for the function or associated topics, position the cursor on the desired link and press the "Follow reference" softkey.
   The selected help page is displayed.



5. Press the "Back to reference" softkey to jump back to the previous help.

Calling a topic in the table of contents



1. Press the "Table of contents" softkey.
   Depending on which technology is set, help is displayed for "Operate Milling", "Operate Turning", "Operate Grinding", or "Operate Universal" - as well as on the topic of "Programming".



2. Select the desired Chapter with the <Cursor down> and <Cursor up> keys.





3. Press the <Cursor right> or <INPUT> key or double-click to open the section.





4. Navigate to the desired topic with the "Cursor down" key.



5. Press the <Follow reference> softkey or the <INPUT> key to display the help page for the selected topic.





6. Press the "Current topic" softkey to return to the original help.

Searching for a topic



1. Press the "Search" softkey.
   The "Search in Help for: " window appears.
2. Activate the "Full text " checkbox to search in all help pages.
   If the checkbox is not activated, a search is performed in the table of contents and in the index.



3. Enter the desired keyword in the "Text" field and press the "OK" softkey.
   If you enter the search term on the operator panel, replace an umlaut (accented character) by an asterisk (*) as dummy.
   All entered terms and sentences are sought with an AND operation. In this way, only documents and entries that satisfy all the search criteria are displayed.



4. Press the "Keyword index" softkey to display the index.

Displaying alarm descriptions and machine data



1. If messages or alarms are active in the "Alarms", "Messages" or "Alarm Log" window, position the cursor at the appropriate display and press <HELP> or key <F12>

   The associated alarm description is displayed.



2. If you are in the "Start-up" operating area in the windows for the display of the machine, setting and drive data, position the cursor on the desired machine data or drive parameter and press the <HELP> key or <F12> key.

   The associated data description is displayed.

Displaying and inserting a G code command in the editor



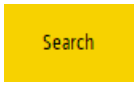
1. A program is opened in the editor.

   Position the cursor on the desired G code command and press the <HELP> or the <F12> key.

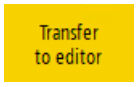
   The associated G code description is displayed.



2. Press the "Display all G functions" softkey.



3. Using the search function, select, for example, the desired G code command.



4. Press the "Transfer to editor" softkey.

   The selected G function is taken into the program at the cursor position.



5. Press the "Exit help" softkey again to close the help.

Multi-touch operation

4

4.1 Touch-sensitive user interface

When using touch panels, wear thin gloves made of cotton or gloves for touch-sensitive glass user interfaces with capacitive touch function.

If you are using somewhat thicker gloves, then exert somewhat more pressure when using the touch panel.

Compatible gloves

You will operate the touch-sensitive glass user interface on the Operator panel optimally with the following gloves.

- Dermatril L
- Camatril Velours type 730
- Uvex Profas Profi ENB 20A
- Camapur Comfort Antistatic type 625
- Carex type 1505 / k (leather)
- Reusable gloves, medium, white, cotton: BM Polyco (RS order number 562-952)

Thicker work gloves

- Thermoplus KCL type 955
- KCL Men at Work type 301
- Camapur Comfort type 619
- Comasec PU (4342)

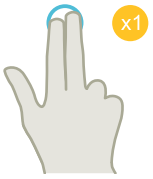
4.2 Finger gestures

Finger gestures



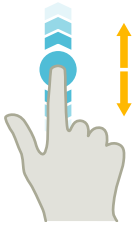
Tap

- Select window
- Select object (e.g. NC set)
- Activate entry field
  - Enter or overwrite value
  - Tap again to change the value



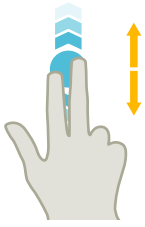
Tap with 2 fingers

- Call the shortcut menu (e.g. copy, paste)



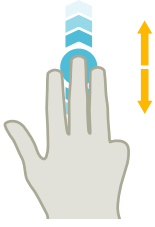
Flick vertically with one finger

- Scroll in lists (e.g. programs, tools, zero points)
- Scroll in files (e.g. NC program)



Flick vertically with two fingers

- Page-scroll in lists (e.g. ZO)
- Page-scroll in files (e.g. NC programs)



Flick vertically with three fingers

- Scroll to the start or end of lists
- Scroll to the start or end of files



Flick horizontally with one finger

- Scroll in lists with many columns



Spread

- Zoom in on graphic contents (e.g. simulation, mold making view)



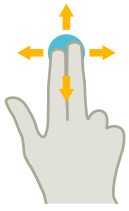
Pinch

- Zoom out from graphic contents (e.g. simulation, mold making view)



Pan with one finger

- Move graphic contents (e.g. simulation, mold making view)
- Move list contents



Pan with two fingers

- Rotate graphic contents (e.g. simulation, mold making view)



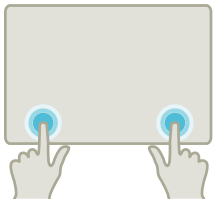
Tap and hold

- Open input fields to change
- Activate or deactivate edit mode (e.g. current block display)



Tap and hold using 2 fingers

- Open cycles line by line to change (without input screen form)



Tap with two index fingers

- Tap with two fingers simultaneously in the lower right- and left-hand corners to open the TCU menu.
  The menu has to be opened for service purposes.

Note

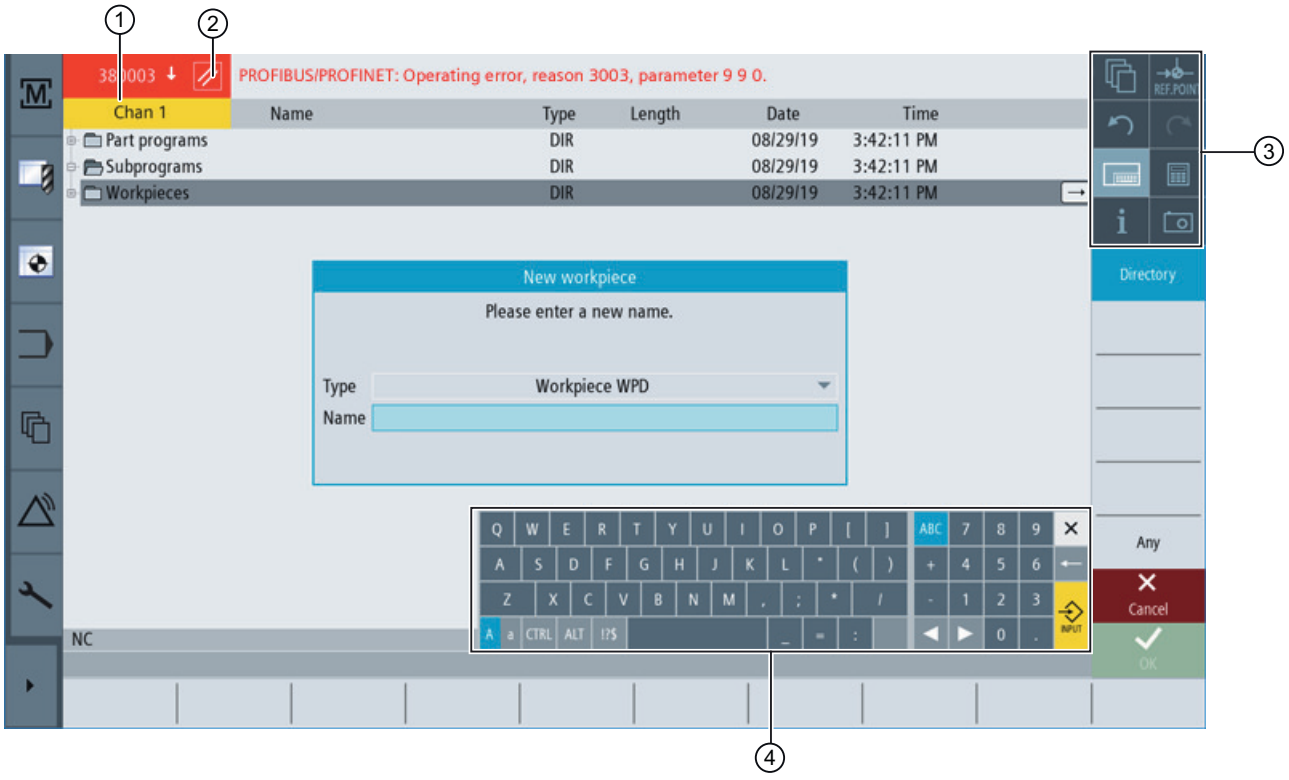
Flicking gestures with several fingers

The gestures only function reliably if you hold your fingers sufficiently far apart. The fingers should be at least 1 cm apart.

4.3 Multitouch user interface

4.3.1 Screen layout

Touch and gesture operator controls for SINUMERIK Operate with the "SINUMERIK Operate Generation 2" user interface.

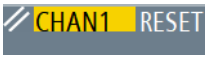


① Changing the channel
② Cancel alarms
③ Function key block
④ Virtual keyboard

4.3.2 Function key block

Operator control	Function
	**Switch operating area** Tap the current operating area, and select the desired operating area from the operating area bar.
	**Switch operating mode** The operating mode is only displayed. To switch the operating mode, tap the operating area and select the operating area from the vertical softkey bar. The selection for the functions available for the operating mode is opened.
	**Close the selection** The selection for the functions available for the operating mode is closed.
	**Switching the handwheel on and off** The handwheel is switched on for axis traversing (HT10). The "Handwheel" operator control has three states: • Gray: Handwheel not available or not configured. • Active (not pressed): Handwheel available and configured. • Shaded (pressed): Handwheel is assigned to the selected axis.
	**Undo** Multiple changes are undone one by one. As soon as a change has been completed in an input field, this function is no longer available.
	**Restoring** Multiple changes are restored one by one. As soon as a change has been completed in an input field, this function is no longer available.
	**Virtual keyboard** Activates the virtual keyboard.
	**Calculator** Displays a calculator.
	**Online help** Opens the online help.
	**Camera** Generates a screenshot.

4.3.3 Further operator touch controls

Operator control	Function
	Advances to the next horizontal softkey bar. When page 2 of the menu is called, the arrow appears on the right.
	Advances to the higher-level menu.
	Advances to the next vertical softkey bar.
	Tapping the Cancel alarm symbol clears all queued cancel alarms.
	If a channel menu has been configured, it is displayed. Tapping the channel display in the status display switches you to the next channel.

4.3.4 Virtual keyboard

If you called the virtual keyboard using the function key block, then you have the option of adapting the key assignment using the shift keys.



① Shift key for uppercase and lowercase letters
② Shift key for letters and special characters
③ Shift key for country-specific keyboard assignment
④ Shift key for full keyboard and numerical key block

Input of Chinese characters in the IME editor

You can enter Chinese characters in the IME editor, even when using the virtual keyboard.

For the input of Chinese characters via the virtual keyboard, change the language of the user interface to Chinese. To display the input field of the IME editor, click on the shift key for country-specific keyboard assignment "CHS".

Hardware keyboard

If a real keyboard is connected, the icon of a minimized keyboard appears in place of the virtual keyboard.

 Use the icon to open the virtual keyboard again.

4.3.5 Special "tilde" character

If the shift key for letters and special characters is pressed, the keyboard assignment changes to the special characters.



① <Tilde>

In the Editor or in alphanumeric input fields, the special character <Tilde> is entered with the <Tilde> key. In numerical input fields, the <Tilde> key changes the sign of a number between plus and minus.

4.4 Expansion with side screen

4.4.1 Overview

Panels in widescreen format provide the possibility of using the extra area to display additional elements. In addition to the SINUMERIK Operate screen, displays and virtual keys are shown to provide faster information and operation.

This sidescreen must be activated. To do this, a navigation bar is displayed.

You can display the following elements above the navigation bar:

- Displaying (widgets)
- Virtual keys (pages)
  - ABC keyboard
  - MCP keys



Machine manufacturer
Refer to the information provided by the machine manufacturer.

NOTICE
No rights management for sidescreen
Sidescreen does not have its own rights management If you program applications for sidescreen, then you are responsible for handling rights management.

Requirements

- A widescreen format multi-touch panel is required to display widgets and pages.
- It is only possible to activate and configure a sidescreen when using the "SINUMERIK Operate Generation 2" user interface.

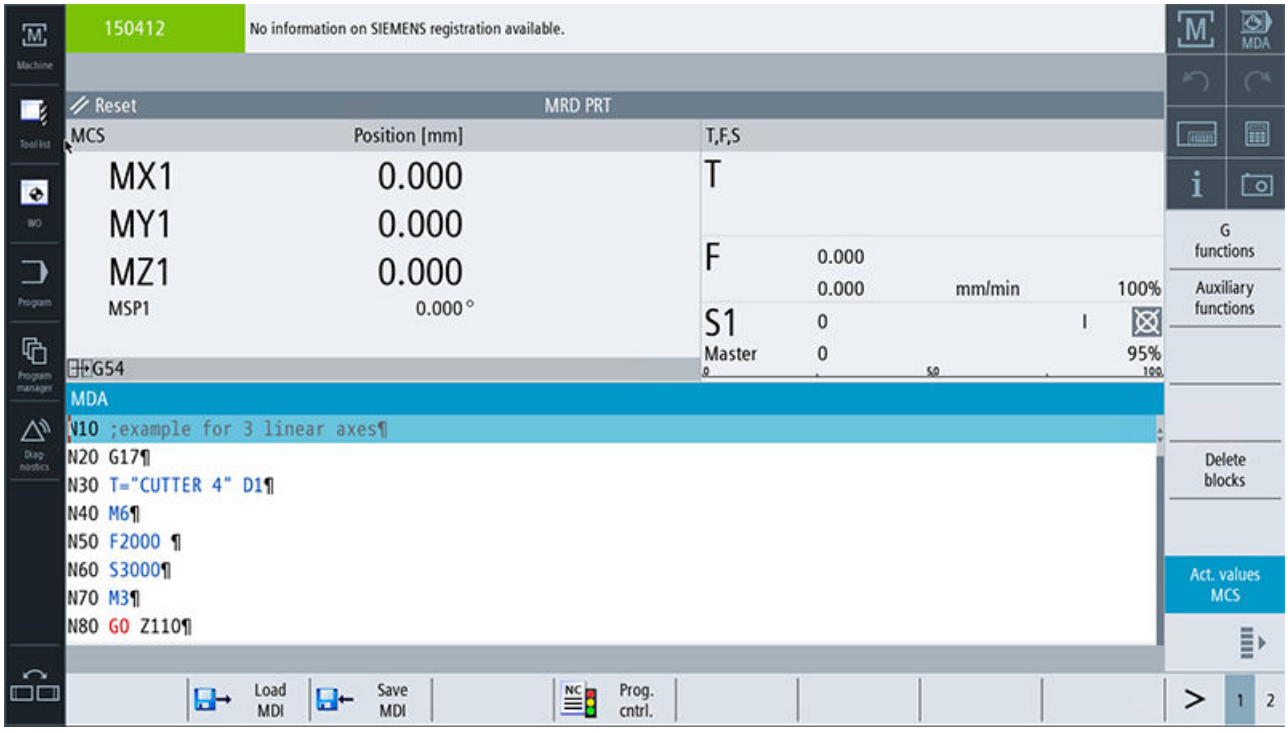
More information

For information on activating the side screen and to configure the virtual keys, refer to the SINUMERIK Operate Commissioning Manual.

4.4.2 Sidescreen with standard windows

When the sidescreen is activated, a navigation bar is shown on the left-hand side of the user interface.

This navigation bar can be used to switch directly to the desired operating area, and to show and hide the sidescreen.



Navigation bar

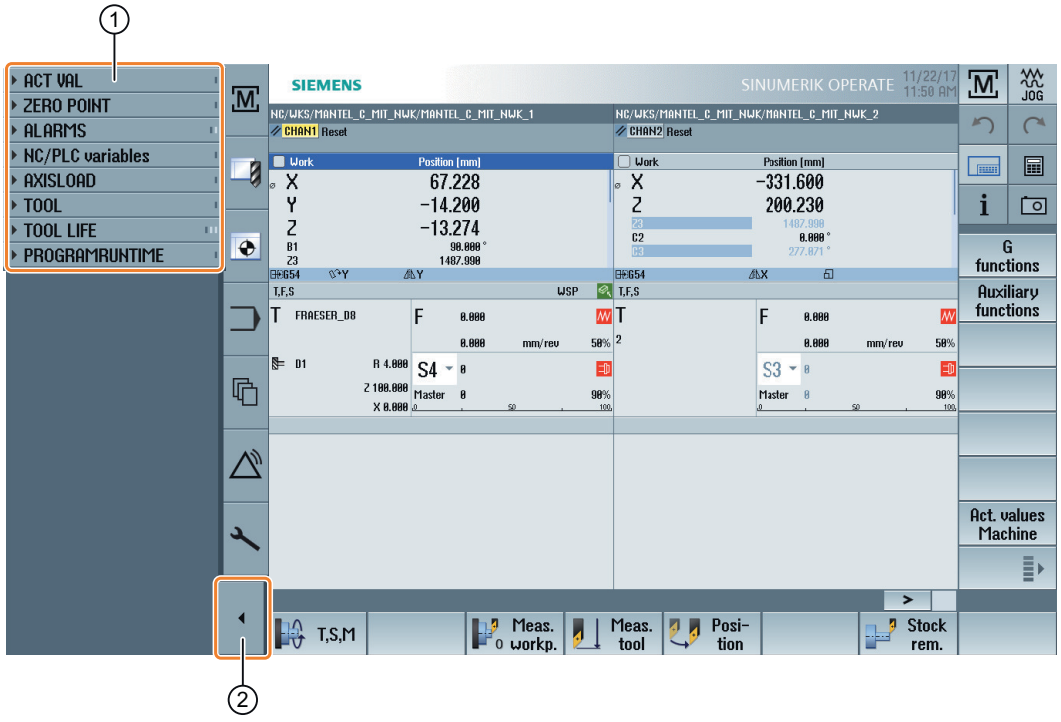
Operator control	Function
	Opens the "Machinery" operating area.
	Opens the tool list in the "Parameter" operating area.
	Opens the "Work offset" window in the "Parameter" operating area.
	Opens the "Program" operating area.
	Opens the "Program manager" operating area.
	Opens the "Diagnostics" operating area.
	Opens the "Commissioning" operating area.

Operator control	Function
	Hides the sidescreen.
	Shows the sidescreen.

4.4.3 Standard widgets

Open sidescreen

- Tap the arrow on the navigation bar to show the sidescreen.
  The standard widgets are displayed in minimized form as the header line.



① Widget header lines
② Arrow key for showing/hiding the sidescreen

Navigating in sidescreen

- To scroll through the list of widgets, swipe vertically with 1 finger.
  - OR -
- To return to the end or to the beginning of the list of widgets, swipe vertically with 3 fingers.

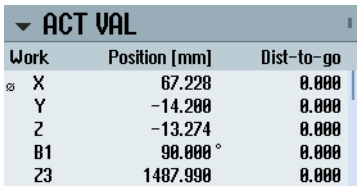
Open widgets

- To open a widget, tap the header line of the widget.

4.4.4 "Actual value" widget

The widget contains the position of the axes in the displayed coordinate system.

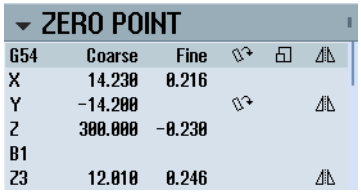
The distance-to-go for the current NC block is displayed while a program is running.



4.4.5 "Zero point" widget

The widget includes values of the active work offset for all configured axes.

The approximate and detailed offset, as well as rotation, scaling and mirroring are displayed for each axis.



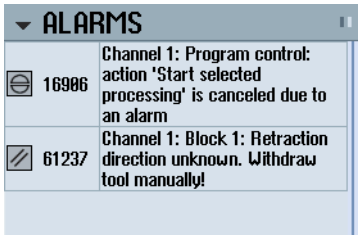
4.4.6 "Alarms" widget

The widget contains all the messages and alarms in the alarm list.

The alarm number and description are displayed for every alarm. An acknowledgment symbol indicates how the alarm is acknowledged or canceled.

Vertical scrolling is possible if multiple alarms are pending.

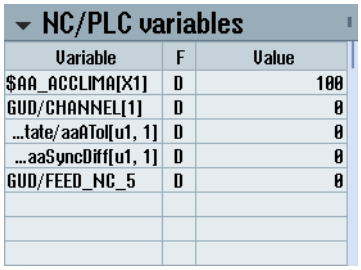
Wipe horizontally to switch between alarms and messages.



4.4.7 "NC/PLC variables" widget

The "NC/PLC variables" widget displays the NC and PLC variables.

The variable name, data type and value are shown for each variable.



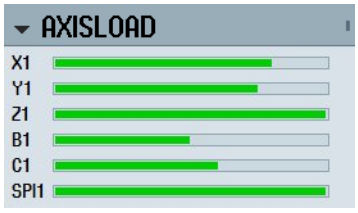
Only those variables that are currently displayed in the "NC/PLC variables" screen in the "Diagnostics" operating area are shown. To update the list in the "NC/PLC variables" widget following a change in the "NC/PLC variables" screen in the "Diagnostics" operating area, collapse and expand the widget again.

Vertical scrolling is possible.

4.4.8 "Axle load" widget

The widget shows the load on all axles in a bar chart.

Up to 6 axes are displayed. Vertical scrolling is possible if multiple axes are present.

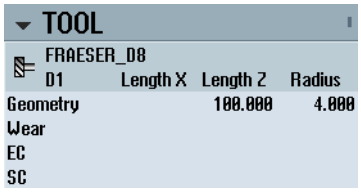


4.4.9 "Tool" widget

The widget contains the geometry and wear data for the active tool.

The following information is additionally displayed depending on the machine configuration:

- EC: Active location-dependent offset - setting up offset
- SC: Active location-dependent offset - additive offset
- TOFF: Programmed tool length offset in WCS coordinates, and programmed tool radius offset
- Override: Value of the overridden movements that were made in the individual tool directions



4.4.10 "Service life" widget

The widget displays the tool monitoring in relation to the following values:

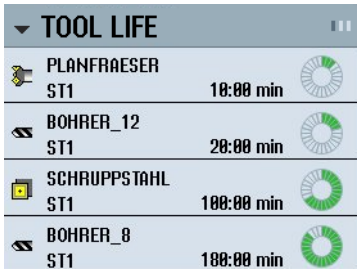
- Operating time of tool (standard time monitoring)
- Finished workpieces (quantity monitoring)
- Tool wear (wear monitoring)

Note

Multiple cutting edges

If a tool has multiple cutting edges, the values of the edge with the lowest residual service life, quantity and wear is displayed.

It possible to alternate between views by scrolling horizontally.



4.4.11 "Program runtime" widget

The widget contains the following data:

- Total runtime of the program
- Time remaining to end of program

This data is estimated for the first program run.

Additionally, progress of the program is visualized in a bar chart as a percentage.



4.4.12 Widget "Camera 1" and "Camera 2"

You can create up to two cameras for tracking remote processes and monitoring difficult-to-access areas.

Widgets "Camera 1" and "Camera 2" are used to display camera images. There is a dedicated widget for each camera.



If the particular camera has been configured, start streaming by opening the widget.

Additional information on activating widgets "Camera 1" and "Camera 2" is provided in the SINUMERIK Operate Commissioning Manual.

4.4.13 Sidescreen with pages for the ABC keyboard and/or machine control panel

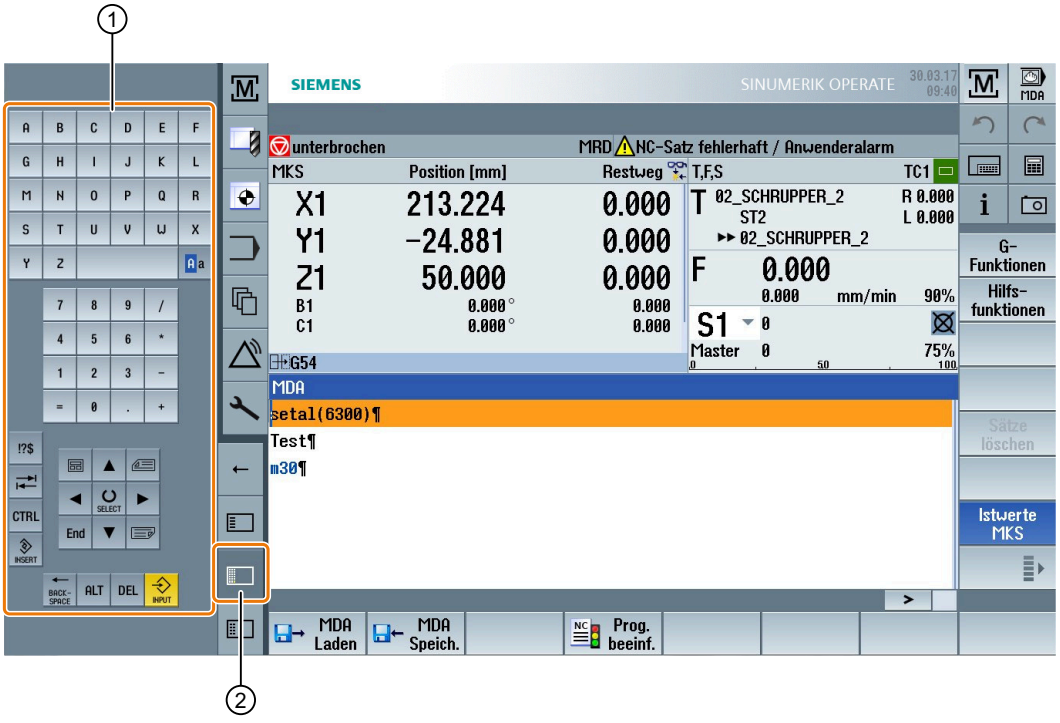
Not only standard widgets but also pages with ABC keyboards and machine control panels can be configured in the sidescreen of a multitouch panel.

Configure ABC keyboard and MCP

If you configured ABC keyboard and MCP keys, then the navigation bar is extended for the sidescreen:

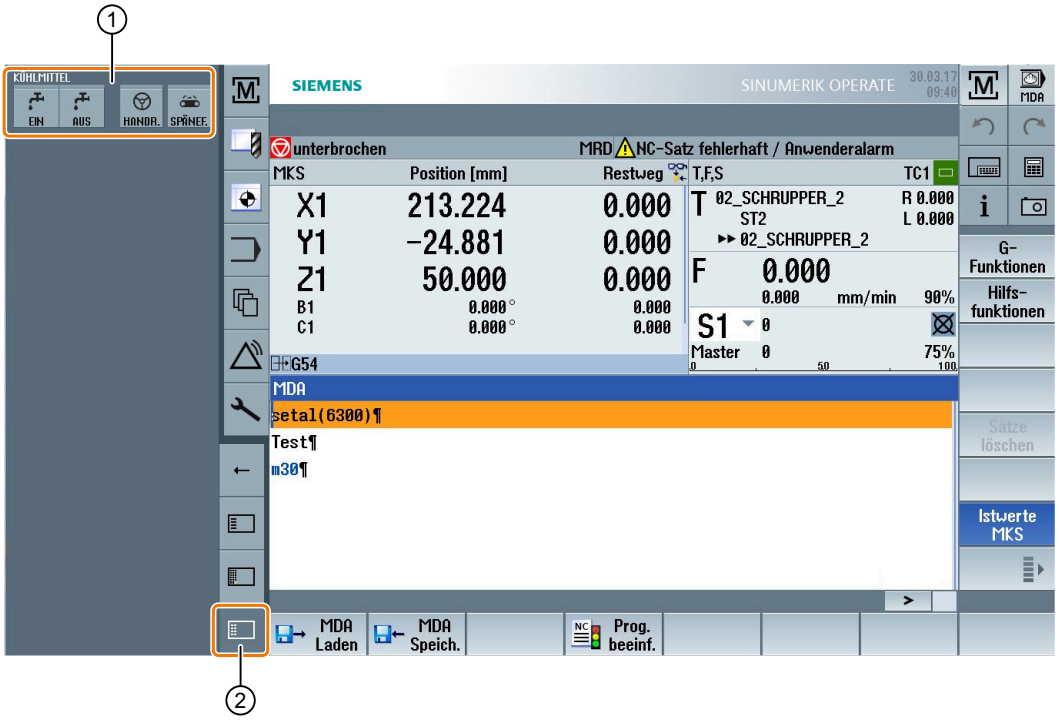
Operator control	Function
	Display of standard widgets in the sidescreen
	Display of an ABC keyboard on the sidescreen
	Display of a machine control panel on the sidescreen

4.4.14 Example 1: ABC keyboard in the sidescreen



① ABC keyboard
② Key to display the keyboard

4.4.15 Example 2: Machine control panel in the sidescreen



① Machine control panel
② Key to display the machine control panel

4.5 SINUMERIK Operate Display Manager

4.5.1 Overview

With a panel with full HD resolution (1920x1080), you have the possibility to work with the Display Manager.

The Display Manager allows you to see a lot of information at a glance.

With the Display Manager, the screen area is divided into several display areas.

In addition to SINUMERIK Operate, widgets, keyboards, a machine control panel and various applications are provided in the various areas.

Display Manager is available for the following configurations:

- SINUMERIK Operate on PC/PCU
- SINUMERIK Operate on NCU ("Embedded HMI") when using an NCU 1740, NCU 1750, NCU 1760 and PPU 1740.



Software option

The option "P81 – SINUMERIK Operate Display Manager" is required for the "SINUMERIK Operate Display Manager" function.

Note

The standard configuration of the Display Manager only supports the landscape orientation of the screen.

Note

The Display Manager does not provide access restrictions for OEM-specific entry points (e.g. menu). The access restrictions must be implemented in the respective OEM application.

NOTICE
No rights management for Display Manager Display Manager does not have its own rights management If you program applications for Display Manager, then you are responsible for handling rights management.

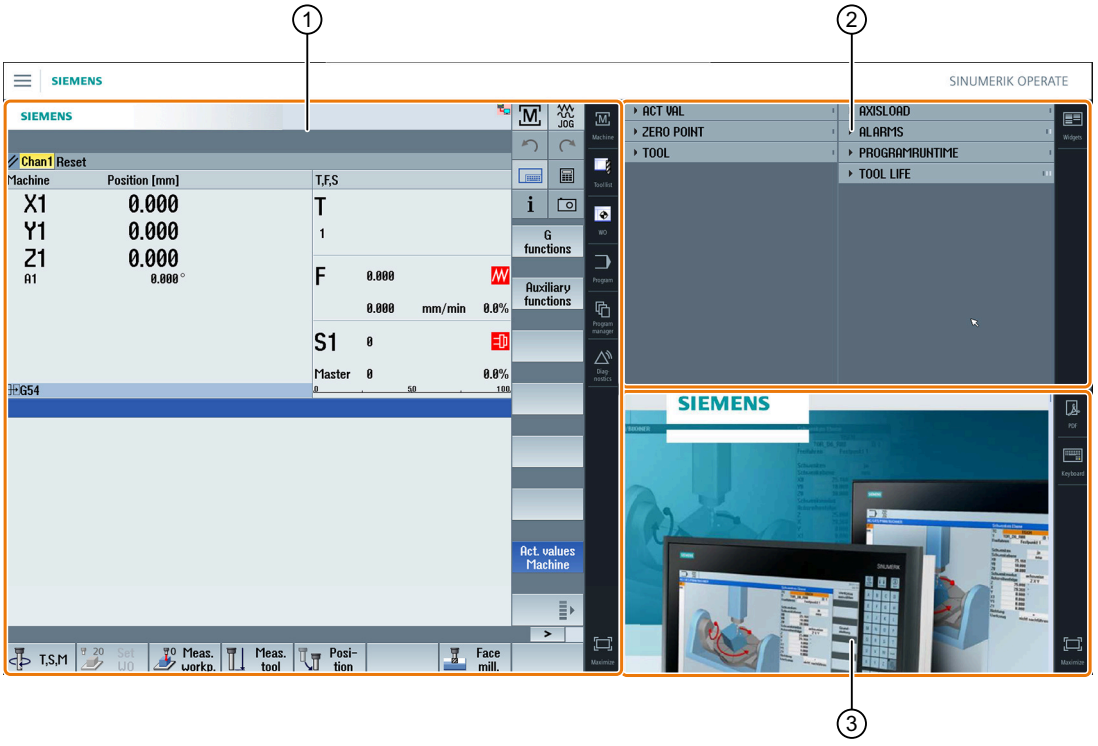
More information

For more information on the activation and configuration of the Display Manager, please refer to the SINUMERIK Operate Commissioning Manual.

You can find additional information about Full HD Panels in the Operator Panel Fronts Equipment Manual: TOP 1500, TOP 1900, TOP 2200.

4.5.2 Screen layout

The standard supply of a SINUMERIK Operate Display Manager offers the option of choosing between 3-display areas and 4-display areas.



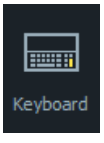
① SINUMERIK Operate with navigation bar for switchover of the operating area
② Display area for standard widgets
③ Display area for applications (e.g. PDF)

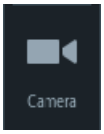
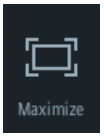
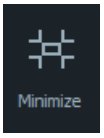
4.5.3 Operator controls

The Display Manager is activated.

Operator control	Function
	**Menu** Tap on the menu to select the desired arrangement of the display areas.
	**3-display areas** • SINUMERIK Operate (with function block) • Widget area • Applications area (PDF, virtual keyboard)

Operator control	Function
	**4-display areas** • SINUMERIK Operate (with function block) • Widget area • Applications area (PDF, virtual keyboard) • Area with virtual keyboard
	**Mirroring display areas** Mirrors the selected arrangement of the display areas.
	**Navigating in SINUMERIK Operate** Tap on the corresponding icon to directly open the desired operating area.
...	
	
	**Widgets** The following widgets are available by default: • Actual values (Page 76) • Zero point (Page 76) • Tool (Page 77) • Axle load (Page 77) • Alarms (Page 76) • Program runtime (Page 78) • Service life (Page 78) • NC/PLC variables (Page 76)

Operator control	Function
	**PDF** Opens the PDF stored here. The following functions are available in the PDF display: • Open (<CTRL> + <O>) • Mark (<CTRL> + <A>) • Copy (<CTRL> + <C>) • Go to (<CTRL> + <G>) • Search (<CTRL> + <F>) • Display and hide bookmarks (<CTRL> + <B>) The pinch/zoom function and finger gestures are supported in the PDF view. Alternatively, use the PDF display via the animated toolbar. The following icons are available in full screen mode: •  Open document •  Mark text in the document •  Copy contents •  Change to entered page •  Search for content •  Rotate view to the left by 90 degrees •  Rotate view to the right by 90 degrees •  Adjust view to window width •  Adjust view to window height •  Enlarge display •  Reduce display •  Show and hide bookmarks You optimize the read view using the following finger gestures: • Double tap: Adapt width: • <CTRL> + double tap: Adapt height: If you change the language when viewing a PDF document, the PDF document is reloaded in the respective language. If no PDF document is available for the language set, the English PDF document is displayed. The position in the PDF document is retained beyond language switchover across sessions if the PDF document contains bookmarks.
 	**Virtual keyboard** Displays a QWERTY keyboard in the display area for applications as well as in the 4th display area below SINUMERIK Operate. If the virtual keyboard is selected while the display area is maximized, the keyboard opens as a pop-up. The keyboard can be moved on the display as required by means of touch operation.

Operator control	Function
	**Camera** Live streaming of the configured camera: • 1 Live streaming camera 1 • 2 Live streaming camera 2 • 1+2 Live streaming camera 1 and camera 2 If a camera has been configured, you can directly view the relevant streaming process. If a camera configuration is changed or an issue with connectivity occurs, reboot the system to activate the streaming process on the camera.
	**Maximizing the display area** Enlarges the area with SINUMERIK Operate and the area for the applications to the full dimensions of the Panel.
	**Minimizing the display area** The area with SINUMERIK Operate and the area for the applications are reduced back to their original size.
	**Machine control panel** Shows a machine control panel. **Note**: Please observe the information provided by the machine manufacturer.

See also

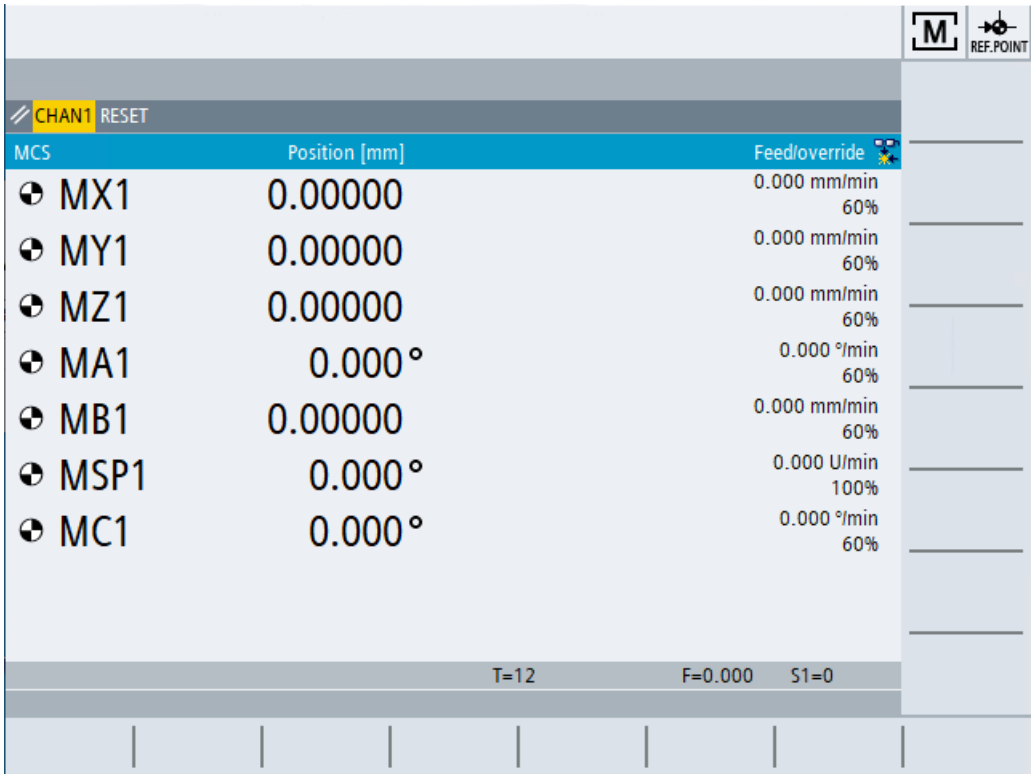
Widget "Camera 1" and "Camera 2" (Page 78)

Setting up the machine

5

5.1 Switching on and switching off

Startup



When the control starts up, the main screen opens according to the operating mode specified by the machine manufacturer. This is usually the main screen for the "REF POINT" function.

Machine manufacturer

Please observe the information provided by the machine manufacturer.

5.2 Approaching a reference point

5.2.1 Referencing axes

Your machine tool can be equipped with an absolute or incremental path measuring system. An axis with incremental path measuring system must be referenced after the controller has been switched on – however, an absolute path measuring system does not have to be referenced.

For the incremental path measuring system, all the machine axes must therefore first approach a reference point, the coordinates of which are known to be relative to the machine zero-point.

Sequence

Prior to the approach, the axes must be in a position from where they can approach the reference point without a collision.

The axes can also all approach the reference point simultaneously, depending on the manufacturer's settings.



Machine manufacturer
Please refer to the machine manufacturer's specifications.

NOTICE
Risk of collision If the axes are not in a collision-free position, you must first traverse them to safe positions in "JOG" or "MDI" mode. You must follow the axis motions directly on the machine! Ignore the actual value display until the axes have been referenced! The software limit switches are not active!

Procedure



1. Press the <JOG> key.



2. Press the <REF. POINT>. key





3. Select the axis to be traversed.





4. Press the <-> or <+> key.
   The selected axis moves to the reference point.
   If you have pressed the wrong direction key, the action is not accepted and the axes do not move.



A symbol is shown next to the axis if it has been referenced.

The axis is referenced as soon as the reference point is reached. The actual value display is set to the reference point value.

From now on, path limits, such as software limit switches, are active.

End the function via the machine control panel by selecting operating mode "AUTO" or "JOG".

5.2.2 User agreement

If you are using Safety Integrated (SI) on your machine, you will need to confirm that the current displayed position of an axis corresponds to its actual position on the machine when you reference an axis. Your confirmation is the requirement for the availability of other Safety Integrated functions.

You can only give your user agreement for an axis after it has approached the reference point.

The displayed axis position always refers to the machine coordinate system (Machine).

Option

User agreement with Safety Integrated is only possible with a software option.

Procedure



1. Select the "Machine" operating area.



2. Press the <REF POINT> key.



3. Select the axis to be traversed.







4. Press the <-> or <+> key.

   The selected axis moves to the reference point and stops. The coordinate of the reference point is displayed.

   The axis is marked with ◔.



5. Press the "User enable" softkey.

   The "User Agreement" window opens.

   It shows a list of all machine axes with their current position and SI position.

6. Position the cursor in the "Acknowledgement" field for the axis in question.



7. Activate the acknowledgement with the <SELECT> key.

   The selected axis is marked with an "x" meaning "safely referenced" in the "Acknowledgement" column.



   By pressing the <SELECT> key again, you deactivate the acknowledgement again.

5.3 Modes of operation

5.3.1 General information

You can work in three different operating modes.

"JOG" mode

"JOG" mode is used for the following preparatory actions:

- Approach reference point, i.e. the machine axis is referenced
- Preparing a machine for executing a program in automatic mode, i.e. measuring tools, measuring the workpiece and, if necessary, defining the work offsets used in the program
- Traverse axes, e.g. during a program interrupt
- Positioning axes

Select "JOG"



Press the <JOG> key.

The following functions are available in "JOG" mode:

- "REF POINT"
- "REPOS"

"REF POINT" function

The "REF POINT" function is used to synchronize the control and the machine. For this purpose, you approach the reference point in "JOG" mode.

Selecting "REF POINT"



Press the <REF POINT> key.

"REPOS" function

The "REPOS" function is used for repositioning to a defined position. After a program interrupt (e.g. to correct tool wear values), move the tool away from the contour in "JOG" mode.

The path differences traversed in "JOG" mode are displayed in the actual value window as the "REPOS" offset.

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

Select "REPOS"



Press the <REPOS> key.

"MDI" mode (Manual Data Input)

In "MDI" mode, you can enter and execute G code commands non-modally to set up the machine or to perform a single action.

Selecting "MDI"



Press the <MDI> key.

The "TEACH IN" function is available in "MDI" mode.

"TEACH IN" function

With the "TEACH IN" function, you can create, edit and execute part programs (main programs and subroutines) for motion sequences or simple workpieces by approaching and saving positions.

Selecting "Teach In"



Press the <TEACH IN> key.

"AUTO" mode

In automatic mode, you can execute a program completely or only partially.

Select "AUTO"



Press the <AUTO> key.

The "Single block" function is available in "AUTO" mode.

"Single block" function

You can execute a program block-by-block with the "Single block" function.

Select "Single block"



Press the <SINGLE BLOCK> key.

5.3.2 Modes groups and channels

Every channel behaves like an independent NC. A maximum of one part program can be processed per channel.

- Control with 1channel
  One mode group exists.
- Control with several channels
  Channels can be grouped to form several "mode groups."

Example

Control with 4 channels, where machining is carried out in 2 channels and 2 other channels are used to control the transport of the new workpieces.

Mode group 1 channel 1 (machining)

Channel 2 (transport)

Mode group 2 channel 3 (machining)

Channel 4 (transport)

Mode groups (MGs)

Technologically-related channels can be combined to form a mode group.

Axes and spindles of the same mode group can be controlled by one or more channels.

An operating mode group is in one of "Automatic", "JOG" or "MDI" operating modes, i.e., several channels of an operating mode group can never assume different operating modes.

5.3.3 Channel switchover

It is possible to switch between channels when several are in use. Since individual channels may be assigned to different mode groups, a channel switchover command is also an implicit mode switchover command.

When a channel menu is available, all of the channels are displayed on softkeys and can be switched over.

Changing the channel



Press the <CHANNEL> key.

The channel changes over to the next channel.

- OR -

If the channel menu is available, a softkey bar is displayed. The active channel is highlighted.

Another channel can be selected by pressing one of the other softkeys.

Further information

For information on configuring the channel menu, refer to the SINUMERIK Operate Commissioning Manual.

5.4 Settings for the machine

5.4.1 Switching over the coordinate system (MCS/WCS)

The coordinates in the actual value display are relative to either the machine coordinate system or the workpiece coordinate system.

By default, the workpiece coordinate system is set as a reference for the actual value display.

The machine coordinate system (MCS), in contrast to the workpiece coordinate system (WCS), does not take into account any zero offsets, tool offsets and coordinate rotation.

Procedure

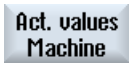


1. Select the "Machine" operating area.

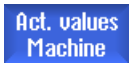




2. Press the <JOG> or <AUTO> key.



3. Press the "Act.vls. MCS" softkey.



The machine coordinate system is selected.
The title of the actual value window changes in the MCS.



Machine manufacturer
The softkey to changeover the coordinate system can be hidden. Please refer to the machine manufacturer's specifications.

5.4.2 Switching the unit of measurement

You can set millimeters or inches as the unit of measurement for the machine. Switching the unit of measurement always applies to the entire machine. All required information is automatically converted to the new unit of measurement, for example:

- Positions
- Tool offsets
- Work offsets

The following conditions must be met before you can switch between units of measurement:

- The corresponding machine data are set.
- All channels are in the reset state.
- The axes are not being traversed via "JOG", "DRF", and the "PLC".
- Constant grinding wheel peripheral speed (GWPS) is not active.



Machine manufacturer
Please observe the information provided by the machine manufacturer.

Further information

Additional information on the inch/metric system of measurement is provided in the Axes and Spindles Function Manual.

Procedure



1. Select the mode <JOG> or <AUTO> in the "Machine" operating area.







2. Press the menu forward key and the "Settings" softkey.
   A new vertical softkey bar appears.





3. Press the "Switch to inch" softkey.
   A prompt asks you whether you really want to switch over the unit of measurement.



4. Press the "OK" softkey.

   The softkey label changes to "Switch to metric".
   The unit of measurement applies to the entire machine.



5. Press the "Switch to metric" softkey to set the unit of measurement of the machine to metric again.

5.4.3 Setting the zero offset

You can enter a new position value in the actual value display for individual axes when a settable work offset is active.

The difference between the position value in the machine coordinate system MCS and the new position value in the workpiece coordinate system WCS is permanently stored in the currently active work offset (e.g. G54).

Note

The result of the work offset varies depending on the channel status.

Relative actual value

Further, you also have the possibility of entering position values in the relative coordinate system.

Note

The new actual value is only displayed. The relative actual value has no effect on the axis positions and the active work offset.

Resetting the relative actual value



Press the "Delete REL" softkey.

The actual values are deleted.

The softkeys to set the zero point in the relative coordinate system are only available if the corresponding machine data is set.



Machine manufacturer
Please refer to the machine manufacturer's instructions.

Requirement

The controller is in the workpiece coordinate system.

The actual value is set in the reset state.

Note

Setting the WO in "interrupted" state

If the channel is in the "interrupted" state, "Set rel." writes the Y axis data with zero bias, and the X axis data is automatically deleted.

Procedure



1. Select the "JOG" mode in the "Machine" operating area.





2. Press the "Set WO" softkey.

- OR -



Press the ">>", "REL act. vals" and "Set REL" softkeys to set position values in the relative coordinate system.

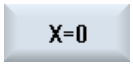






3. Enter the new required position value for X, Y or Z directly in the actual value display (you can toggle between the axes with the cursor keys) and press the "Input" key to confirm the entries.

- OR -

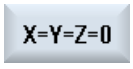


Press softkeys "X=0","Y=0" or "Z=0" to set the relevant position to zero.

...



- OR -



Press softkey "X=Y=Z=0" to set all axis positions to zero simultaneously.

Resetting the actual value



Press the "Delete active WO" softkey.

The offset is deleted permanently.

Note

Irreversible active work offset

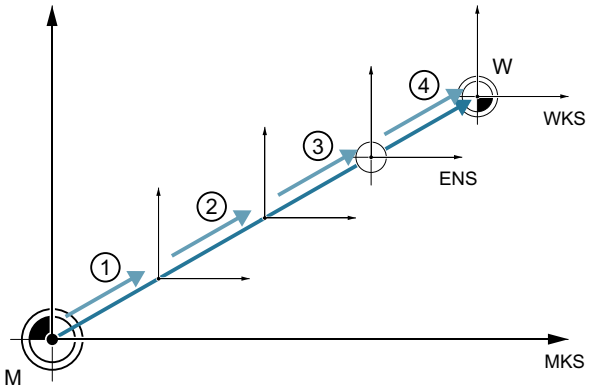
The current active work offset is irreversibly deleted by this action.

5.5 Zero offsets

Following reference point approach, the actual value display for the axis coordinates is based on the machine zero (M) of the machine coordinate system (Machine). The program for machining the workpiece, however, is based on the workpiece zero (W) of the workpiece coordinate system (Work). The machine zero and workpiece zero are not necessarily identical. The distance between the machine zero and the workpiece zero depends on the workpiece type and how it is clamped. This zero offset is taken into account during execution of the program and can be a combination of different offsets.

Following reference point approach, the actual value display for the axis coordinates is based on the machine zero of the machine coordinate system (Machine).

The actual value display of the positions can also refer to the SZS coordinate system (settable zero system). The position of the active tool relative to the workpiece zero is displayed.



① Base offset
② Work offset, coarse
③ Work offset, fine
④ Coordinate transformation

Figure 5-1 Work offsets

When the machine zero is not identical to the workpiece zero, at least one offset (base offset or zero offset) exists in which the position of the workpiece zero is saved.

Base offset

The base offset is a zero offset that is always active. If you have not defined a base offset, its value will be zero. The base offset is specified in the "Zero Offset - Base" window.

Coarse and fine offsets

Every zero offset (G54 to G57, G505 to G599) consists of a coarse offset and a fine offset. You can call the zero offsets from any program (coarse and fine offsets are added together).

You can save the workpiece zero, for example, in the coarse offset, and then store the offset that occurs when a new workpiece is clamped between the old and the new workpiece zero in the fine offset.

Note

Deselect fine offset

You have the option of deselecting the fine offset using machine data MD18600 $MN_MM_FRAME_FINE_TRANS

5.5.1 Display active zero offset

The following zero offsets are displayed in the "Zero Offset - Active" window:

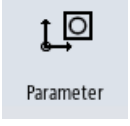
- Zero offsets, for which active offsets are included, or for which values are entered.
- Settable zero offsets
- Total zero offset

This window is generally used only for monitoring.

The availability of the offsets depends on the setting.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

Procedure



1. Select the "Parameter" operating area.



2. Press the "Zero offset" softkey.
   The "Zero Offset - Active" window is opened.



Note

Further details on zero offsets

If you would like to see further details about the specified offsets or if you would like to change values for the rotation, scaling or mirroring, press the "Details" softkey.

5.5.2 Displaying the zero offset "overview"

The active offsets or system offsets are displayed for all axes that have been set up in the "Work offset - overview" window.

In addition to the offset (coarse and fine), the rotation, scaling and mirroring defined using this are also displayed.

This window is generally used only for monitoring.

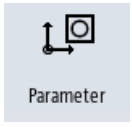
Display of active work offsets

Work offsets	
MCS actual value	Display of the actual value in the Machine Coordinate System.
Kin. trans. workpiece	Displays the additional work offsets programmed with $P_TRAFRAME_P.
Kin. trans. tool	Displays the additional work offsets programmed with $P_TRAFRAME_T.
DRF	Displays the actual DRF offset in the basis coordinate system (BCS) for additive work offset of geometry and special axes.
$AA_OFF override	Displays the actual tool compensation offsets using the online tool length compensation ($AA_TOFF) in the tool coordinate system (TCS).
Basic reference	Displays the additional work offsets programmed with $P_SETFRAME. Access to the system offsets is protected via a keyswitch.
External WO frame	Displays the additional work offsets programmed with $P_EXTFRAME.
Total base WO	Displays all effective basis offsets.
G500	Displays the work offsets activated with G54 - G599. Under certain circumstances, you can change the data using "Set WO", i.e. you can correct a zero point that has been set.
Tool reference	Displays the additional work offsets programmed with $P_TOOLFRAME.
Workpiece reference	Displays the additional work offsets programmed with $P_WPFRAME.
Transformer reference	Displays the additional work offsets programmed with $P_TRAFRAME.
Programmed WO	Displays the additional work offsets programmed with $P_PFRAME.
Cycle reference	Displays the additional work offsets programmed with $P_CYCFRAME.
Total WO	Displays the active work offset, resulting from the total of all work offsets.
TOFF	Displays the tool compensation offsets programmed using TOFFL/TOFF/TOFFR/TOFFLR.
T:	Displays the active tool.
WCS actual value	Displays the actual value in the workpiece coordinate system.

The display of the zero point offsets depends on the settings.

Machine manufacturer
Please observe the machine manufacturer's specifications.

Procedure



1. Select the "Parameter" operating area.



2. Press the "Work offset" and "Overview" softkeys.
   The "Work offsets - Overview" window opens.

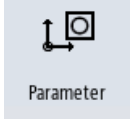


5.5.3 Displaying and editing base zero offset

The defined channel-specific and global base offsets, divided into coarse and fine offsets, are displayed for all set-up axes in the "Zero offset - Base" window.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

Procedure



1. Select the "Parameter" operating area.



2. Press the "Zero offset" softkey.



3. Press the "Base" softkey.
   The "Zero Offset - Base" window is opened.
4. You can edit the values directly in the table.

Note
Activate base offsets

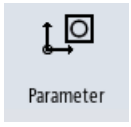
The offsets specified here are immediately active.

5.5.4 Displaying and editing settable zero offset

All settable offsets, divided into coarse and fine offsets, are displayed in the "Work offset - G54...G599" window.

Rotation, scaling and mirroring are displayed.

Procedure



1. Select the "Parameter" operating area.



2. Press the "Work offset" softkey.



3. Press the "G54 ... G599" softkey.
   The "Work offset - G54 ... G599 [mm]" window opens.
   Note
   The labeling of the softkeys for the settable work offsets varies, i.e. the settable work offsets configured on the machine are displayed (examples: G54 ... G57, G54 ... G505, G54 ... G599).
   Please observe the information provided by the machine manufacturer.
4. You can edit the values directly in the table.

Note

Activate settable zero offsets

The settable zero offsets must first be selected in the program before they have an impact.

5.5.5 Displaying and editing details of the zero offsets

For each zero offset, you can display and edit all data for all axes. You can also delete zero offsets.

For every axis, values for the following data will be displayed:

- Coarse and fine offsets
- Rotation
- Scaling
- Mirroring

Machine manufacturer
Please refer to the machine manufacturer's specifications.

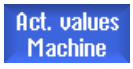
Note

Settings for rotation, scaling and mirroring are specified here and can only be changed here.

Tool details

You can display the following details for the tool and wear data for tools:

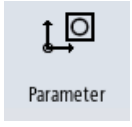
- TC
- Adapter dimension
- Length / length wear
- EC setup correction
- SC sum correction
- Total length
- Radius / radius wear



You can also change the display of the tool correction values between the Machine Coordinate System and the Workpiece Coordinate System.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

Procedure



1. Select the "Parameter" operating area.



2. Press the "Zero offset" softkey.





3. Press the "Active", "Base" or "G54...G599" softkey.
   The corresponding window opens.

4. Place the cursor on the desired zero offset to view its details.



5. Press the "Details" softkey.

   A window opens, depending on the selected zero offset, e.g. "Zero Offset - Details: G54 to G599".

6. You can edit the values directly in the table.

   - OR -



Press the "Clear offset" softkey to reset all entered values.



...



Press the "ZO +" or "ZO -" softkey to select the next or previous offset, respectively, within the selected area ("Active", "Base", "G54 to G599") without first having to switch to the overview window.

If you have reached the end of the range (e.g. G599), you will switch automatically to the beginning of the range (e.g. G54).

These value changes are available in the part program immediately or after "Reset".

Machine manufacturer

Please refer to the machine manufacturer's specifications.

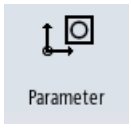


Press the "Back" softkey to close the window.

5.5.6 Deleting a zero offset

You have the option of deleting work offsets. This resets the entered values.

Procedure



1. Select the "Parameter" operating area.



2. Press the "Work offset" softkey.



3. Press the "Overview", "Basis" or "G54...G599" softkey.

...





4. Press the "Details" softkey.

5. Position the cursor on the work offset you would like to delete.



6. Press the "Clear offset" softkey.

   A confirmation prompt is displayed as to whether you really want to delete the work offset.



7. Press the "OK" softkey to confirm that you wish to delete the work offset.

5.6 Monitoring axis and spindle data

5.6.1 Specify working area limitations

Using the "Working area limitation" function you can limit the range within which a tool should traverse in all channel axes. This function allows you to set up protection zones in the working area that are inhibited for tool motion.

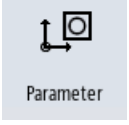
In this way, you are able to restrict the traversing range of the axes in addition to the limit switches.

Requirements

You can only make changes in "AUTO" mode when in the RESET condition. These changes are then immediate.

You can make changes in "JOG" mode at any time. These changes, however, only become active at the start of a new motion.

Procedure



1. Select the "Parameter" operating area.



2. Press the "Setting data" softkey.



The "Working Area Limitation" window appears.

3. Place the cursor in the required field and enter the new values via the numeric keyboard.

   The upper or lower limit of the protection zone changes according to your inputs.

4. Click the "active" checkbox to activate the protection zone.

Note

You will find all of the setting data in the "Start-up" operating area under "Machine data" via the menu forward key.

5.6.2 Editing spindle data

The speed limits set for the spindles that must not be under- or overshot are displayed in the "Spindles" window.

You can limit the spindle speeds in fields "Minimum" and "Maximum" within the limit values defined in the relevant machine data.

Spindle speed limitation at constant cutting rate

In field "Spindle speed limitation at G96", the programmed spindle speed limitation at constant cutting speed is displayed together with the permanently active limitations.

This speed limitation, for example, prevents the spindle from accelerating to the max. spindle speed of the current gear stage (G96) when performing tapping operations or machining very small diameters.

Note

The "Spindle data" softkey only appears if a spindle is configured.

Procedure



1. Select the "Parameter" operating area.





2. Press the "Setting data" and "Spindle data" softkeys.
   The "Spindles" window opens.

3. If you want to change the spindle speed, place the cursor on the "Maximum", "Minimum", or "Spindle speed limitation at G96" and enter a new value.

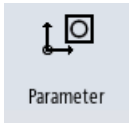
5.7 Displaying setting data lists

You can display lists with configured setting data.



Machine manufacturer
Please refer to the machine manufacturer's specifications.

Procedure



1. Select the "Parameter" operating area.



2. Press the "Setting data" and "Data lists" softkeys.
   The "Setting Data Lists" window opens.





3. Press the "Select data list" softkey and in the "View" list, select the required list with setting data.

5.8 Handwheel assignment

You can traverse the axes in the machine coordinate system (Machine) or in the workpiece coordinate system (Work) via the handwheel.

All axes are provided in the following order for handwheel assignment:

- Geometry axes
  When traversing, the geometry axes take into account the current machine status (e.g. rotations, transformations). All channel machine axes, which are currently assigned to the geometry axis, are in this case simultaneously traversed.
- Channel machine axes
  Channel machine axes are assigned to the particular channel. They can only be traversed individually, i.e. the current machine state has no influence.
  The also applies to channel machine axes, that are declared as geometry axes.



Machine manufacturer

Please observe the information provided by the machine manufacturer.

Procedure



1. Select the "Machine" operating area.





2. Press the <JOG>, <AUTO> or <MDI> key.





3. Press the menu forward key and the "Handwheel" softkey.
   The "Handwheel" window appears.
   A field for axis assignment will be offered for every connected handwheel.
4. Position the cursor in the field next to the handwheel with which you wish to assign the axis (e.g. No. 1).



5. Press the corresponding softkey to select the desired axis (e.g. "X").

   - OR





   Open the "Axis" selection box using the <INSERT> key, navigate to the desired axis, and press the <INPUT> key.
   Selecting an axis also activates the handwheel (e.g., "X" is assigned to handwheel no. 1 and is activated immediately).



6. Press the "Handwheel" softkey again.



- OR -

Press the "Back" softkey.

The "Handwheel" window closes.

Deactivate handwheel

1. Position the cursor on the handwheel whose assignment you wish to cancel (e.g. No. 1).



2. Press the softkey for the assigned axis again (e.g. "X").

- OR -



Open the "Axis" selection box using the <INSERT> key, navigate to the empty field, and press the <INPUT> key.



Clearing an axis selection also clears the handwheel selection (e.g., "X" is cleared for handwheel no. 1 and is no longer active).

5.9 MDA

In "MDI" mode (Manual Data Input mode), you can enter G-code commands or standard cycles block-by-block and immediately execute them for setting up the machine.

You have the option of loading an MDI program or a standard program with the standard cycles directly into the MDI buffer from the program manager; you can subsequently then edit it.



Machine manufacturer

Please observe the information provided by the machine manufacturer.

You can save programs, generated or modified in the MDI working window, in the program manager, e.g. in a directory specifically created for the purpose.

5.9.1 Loading an MDA program from the Program Manager

Procedure



1. Select the "Machine" operating area.



2. Press the <MDA> key.

   The MDI editor opens.



3. Press the "Load MDI" softkey.

   The "Load into MDI" window opens. It shows you a view of the program manager.



4. Position the cursor to the corresponding storage location, press the "Search" softkey and enter the required search term in the search dialog if you wish to search for a specific file.

   Note: The place holders "*" (replaces any character string) and "?" (replaces any character) make it easier for you to perform a search.

5. Select the program that you would like to edit or execute in the MDI window.



6. Press the "OK" softkey.

   The window closes and the program is ready for operation.

5.9.2 Saving an MDA program

Procedure



1. Select the "Machine" operating area.



2. Press the <MDA> key.

   The MDI editor opens.

3. Create the MDI program by entering the G-code commands using the operator's keyboard.



4. Press the "Save MDI" softkey.

   The "Save from MDI: Select storage location" window opens. It shows you a view of the program manager.

5. Select the drive to which you want to save the MDI program you created, and place the cursor on the directory in which the program is to be stored.

   - OR -



   Position the cursor to the required storage location, press the "Search" softkey and enter the required search term in the search dialog if you wish to search for a specific directory or subdirectory.

   Note: The place holders "*" (replaces any character string) and "?" (replaces any character) make it easier for you to perform a search.



6. Press the "OK" softkey.

   When you place the cursor on a folder, a window opens which prompts you to assign a name.

   - OR -

   When you place the cursor on a program, you are asked whether the file should be overwritten.



7. Enter the name for the rendered program and press the "OK" softkey.

   The program will be saved under the specified name in the selected directory.

5.9.3 Editing/executing a MDI program

Procedure



1. Select the "Machine" operating area.



2. Press the <MDA> key.
   The MDI editor opens.
3. Enter the desired G-code commands using the operator's keyboard.
   - OR -
   Enter a standard cycle, e.g. CYCLE62 ().

Editing G-code commands/program blocks

4. Edit G-code commands directly in the "MDI" window.
   - OR -
   Select the required program block (e.g. CYCLE62) and press the <cursor right> key, enter the required value and press "OK".







When editing a cycle, either the help screen or the graphic view can be displayed.



5. Press the <CYCLE START> key.

   The control executes the input blocks.

When executing G-code commands and standard cycles, you have the option of controlling the sequence as follows:

- Executing the program block-by-block
- Testing the program
  Settings under program control
- Setting the test-run feedrate
  Settings under program control

See also

Program control (Page 148)

5.9.4 Deleting an MDA program

Requirement

The MDI editor contains a program that you created in the MDI window or loaded from the program manager.

Procedure



Press the "Delete blocks" softkey.

The program blocks displayed in the program window are deleted.

Working in manual mode

6

6.1 General

Always use "JOG" mode when you want to set up the machine for the execution of a program or to carry out simple traversing movements on the machine:

- Synchronize the measuring system of the controller with the machine (reference point approach)
- Set up the machine, i.e. activate manually-controlled motions on the machine using the keys and handwheels provided on the machine control panel.
- You can activate manually controlled motions on the machine using the keys and handwheels provided on the machine control panel while a part program is interrupted.

6.2 Selecting a tool and spindle

6.2.1 T, S, M windows

For the preparatory actions in manual mode, tool selection and spindle control are both performed centrally in a screen form.

In manual mode, you can select a tool either by its name or its location number. If you enter a number, a search is performed for a name first, followed by a location number. This means that if you enter "5", for example, and no tool with the name "5" exists, the tool is selected from location number "5".

Note

Using the location number, you can thus swing around an empty space into the machining position and then comfortably install a new tool.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

Parameter	Meaning		Unit
T	Input of the tool (name or location number) You can select a tool from the tool list using the "Select tool" softkey.		
D	Cutting edge number of the tool (1 - 9)		
ST	Sister tool number (1 - 99 for sister tool strategy)		
Spindle	Spindle selection, identification with spindle number		
Spindle M function		Spindle off: Spindle is stopped	
		CCW rotation: Spindle rotates counterclockwise	
		CW rotation: Spindle rotates clockwise	
		Spindle positioning: Spindle is moved to the desired position.	
Other M functions	Input of machine functions Refer to the machine manufacturer's table for the correlation between the meaning and number of the function.		
Work offset G	Selection of the work offset (basic reference, G54 - 57) You can select work offsets from the tool list of settable work offsets via the "Work offset" softkey.		
Dimension unit	Selecting the measurement unit The setting made here has an effect on the programming.		inch mm
Machining plane	Selection of the machining plane (G17(XY), G18 (ZX), G19 (YZ))		

Parameter	Meaning	Unit
Gear stage	Specification of the gear stage (auto, I - V)	
Stop position	Entering the spindle position	degrees

Note

Spindle positioning

You can use this function to position the spindle at a specific angle, e.g. during a tool change.

- A stationary spindle is positioned via the shortest possible route.
- A rotating spindle is positioned as it continues to turn in the same direction.

6.2.2 Selecting a tool

Procedure



1. Select the "JOG" operating mode.





2. Press the "T, S, M" softkey.

3. Enter the name or the number of the tool T in the input field.

   - OR -



   Press the "Select tool" softkey.

   The tool selection window opens.

   Place the cursor on the desired tool and press the "OK" softkey.

   The tool is transferred to the "T, S, M... window" and displayed in the field of tool parameter "T".



4. Select tool edge D or enter the number directly in field "D".



5. Select the sister tool ST or enter the number directly into the "ST" field



6. Press the <CYCLE START> key.

   The tool is loaded into the spindle.

6.2.3 Starting and stopping a spindle manually

Procedure



1. Select the "JOG" operating mode.





2. Press the "T, S, M" softkey.

3. Select the desired spindle (e.g. S1) and enter the desired spindle speed (rpm) or the constant cutting velocity (m/min) in the adjacent input field.
   The spindle remains stationary.



4. If the machine has a gearbox for the spindle, set the gear stage (e.g. auto).



5. Select a spindle direction of rotation (clockwise or counterclockwise) in the "Spindle M function" field.



6. Press the <CYCLE START> key.
   The spindle rotates.



7. Select the "Stop" setting in the "Spindle M function" field.



Press the <CYCLE START> key.
The spindle stops.

Note

Changing the spindle speed

If you enter the speed in the "Spindle" field while the spindle is rotating, the new speed is applied.

6.2.4 Position spindle

Procedure



1. Select the "JOG" operating mode.





2. Press the "T, S, M" softkey.



3. Select the "Stop Pos." setting in the "Spindle M function" field.
   The "Stop Pos." entry field appears.
4. Enter the desired spindle stop position.
   The spindle position is specified in degrees.



5. Press the <CYCLE START> key.

   The spindle is moved to the desired position.

Note

You can use this function to position the spindle at a specific angle, e.g. during a tool change.

- A stationary spindle is positioned via the shortest possible route.
- A rotating spindle is positioned as it continues to turn in the same direction.

6.3 Traversing axes

You can traverse the axes in manual mode via the Increment or Axis keys or handwheels.

During a traverse initiated from the keyboard, the selected axis moves at the programmed setup feedrate. During an incremental traverse, the selected axis traverses a specified increment.

Set the default feedrate

Specify the feedrate to be used for axis traversal in the set-up, in the "Settings for Manual Operation" window.

6.3.1 Traverse axes by a defined increment

You can traverse the axes in manual mode via the Increment and Axis keys or handwheels.

Note

Menu for increment selection

If you are using the multi-touch user interface, you can call up the menu for selecting increments for axis movement using the function key block (Page 70).

The menu for selecting increments is always available on the HT 10.
On all other HMI devices, the menu for selecting increments is only available if it is a multi-touch panel, and if bit 17 is set in machine data MD52210.

The machine data MD52210 bit 17 is channel-specific and must be set separately for each NC channel.

Procedure



1. Select the "Machine" operating area.



2. Press the <JOG> key.





3. Press keys 1, 10, etc. up to 10000 in order to move the axis in a defined increment.

   The numbers on the keys indicate the traversing distance in micrometers or microinches.

   Example: Press the "100" button for a desired increment of 100 µm (= 0.1 mm).





4. Select the axis to be traversed.





5. Press the <+> or <-> key.
   Each time you press the key the selected axis is traversed by the defined increment.
   Feedrate and rapid traverse override switches can be operative.

Note

When the controller is switched on, the axes can be traversed right up to the limits of the machine as the reference points have not yet been approached and the axes referenced. Emergency limit switches might be triggered as a result.

The software limit switches and the working area limitation are not yet operative!

The feed enable signal must be set.



Machine manufacturer
Please refer to the machine manufacturer's instructions.

6.3.2 Traversing axes by a variable increment

Procedure



1. Select the "Machine" operating area.



2. Press the <JOG> key.



3. Press the "Settings" softkey.
   The "Settings for Manual Operation" window is opened.
4. Enter the desired value for the "Variable increment" parameter.
   Example: Enter 500 for a desired increment of 500 µm (0.5 mm).



5. Press the <Inc VAR> key.

6. Select the axis to be traversed.





7. Press the <+> or <-> key.

   Each time you press the key the selected axis is traversed by the set increment.

   Feedrate and rapid traverse override switches can be operative.

6.4 Positioning axes

In manual mode, you can traverse individual or several axes to certain positions in order to implement simple machining sequences.

The feedrate / rapid traverse override is active during traversing.

Procedure

1. If required, select a tool.
2. 

Select the "JOG" operating mode.
3. 

Press the "Positions" softkey.
4. Specify the desired value for the feedrate F.

   - OR -

   Press the "Rapid traverse" softkey.

   The rapid traverse is displayed in field "F".
5. Enter the target position or target angle for the axis or axes to be traversed.
6. 

Press the <CYCLE START> key.

   The axis is traversed to the specified target position.

   If target positions were specified for several axes, the axes are traversed simultaneously.

6.5 Manual retraction

In the following cases, the "Retract" function allows drilling tools to be retracted in the tool direction in the JOG mode:

- After interrupting a thread tapping operation (G33/331/G332),
- After interrupting machining operations using drilling tools (tools 200 to 299) as a result of power failure or a RESET at the machine control panel.

The tool and/or the workpiece remain undamaged.

Retraction is especially useful when the coordinate system is swiveled, i.e. the infeed axis is not in the vertical position.

Note

Tapping

In the case of tapping, the form fit between the tap and the workpiece is taken into account and the spindle moved according to the thread.

Use the Z axis as well as the spindle when retracting from threads.

The machine OEM sets up the "Retract" function.



Machine manufacturer

Please observe the information provided by the machine manufacturer.

Procedure

1. The power feed to the machine is interrupted.
   - OR -
   <RESET> interrupts an active part program.
2. After a power supply interruption, switch on the controller.



3. Select the JOG operating mode.



4. Press the Menu forward key.



5. Press the "Retract" softkey.
   The "Retract Tool" window opens.
   The softkey is available only when an active tool and retraction data are present.



6. Select the "WCS" coordinate system on the machine control panel.



7. Use the traversing keys (e.g. Z +) to traverse the tool from the workpiece according to the retraction axis displayed in the "Retract Tool" window.



8. Press the "Retract" softkey again when the tool is at the desired position.

6.6 Default settings for manual mode

Specify the configurations for the manual mode in the "Settings for manual operation" window.

Default settings

Settings	Meaning
Type of feedrate	Here, you select the type of feedrate.
	• G94: Axis feedrate/linear feedrate • G95: Revolutional feedrate
Setup feedrate G94	Enter the desired feedrate in mm/min.
Setup feedrate G95	Enter the desired feedrate in mm/rev.
Variable increment	For variable increments, enter the desired increment when traversing axes.
Spindle speed	Here, enter the desired spindle speed in rpm.

Procedure



1. Select the "Machine" operating area.



2. Press the <JOG> key.





3. Press the menu forward key and the "Settings" softkey.
   The "Settings for manual operation" window is opened.

Machining the workpiece

7

7.1 Starting and stopping machining

During execution of a program, the workpiece is machined in accordance with the programming on the machine. After the program is started in automatic mode, workpiece machining is performed automatically.

Preconditions

The following requirements must be met before executing a program:

- The measuring system of the controller is referenced with the machine.
- The necessary tool offsets and work offsets have been entered.
- The necessary safety interlocks implemented by the machine manufacturer are activated.

General sequence



1. Use the Program manager to select the desired program.





2. Select under "NC", "Local. Drive", "USB" or set-up network drives the desired program.



3. Press the "Select" softkey.
   The program is selected for execution and automatically switched to the "Machine" operating area.



4. Press the <CYCLE START> key.
   The program is started and executed.

Note

Starting the program in any operating area

If the control system is in the "AUTO" mode, you can also start the selected program when you are in any operating area.

Stopping machining



Press the <CYCLE STOP> key.

Machining stops immediately, individual blocks do not finish execution. At the next start, execution is resumed at the same location where it stopped.

Canceling machining



Press the <RESET> key.

Execution of the program is interrupted. On the next start, machining will start from the beginning.



Machine manufacturer

Please observe the information provided by the machine manufacturer.

See also

EXTCALL (Page 361)

7.2 Selecting a program

Procedure



1. Select the "Program Manager" operating area.
   The directory overview is opened.
2. Select the location where the program is archived (e.g. "NC")
3. Place the cursor on the directory containing the program that you want to select.



4. Press the <INPUT> key.
   - OR -



   Press the <Right cursor> key.
   The directory contents are displayed.
5. Place the cursor on the desired program.



6. Press the "Select" softkey.
   When the program has been successfully selected, an automatic change-over to the "Machine" operating area occurs.

7.3 Program running-in

When testing a program, you can select that the system can interrupt the machining of the workpiece after each program block, which triggers a movement or auxiliary function on the machine. In this way, you can control the machining result block-by-block during the initial execution of a program on the machine.

Note

Settings for the automatic mode

Rapid traverse reduction and dry run feed rate are available to run-in or to test a program.

Move by single block

In "Program control" you may select from among several types of block processing:

SB mode	Scope
SB1 Single block, coarse	Machining stops after every machine block (except for cycles).
SB2 Data block	Machining stops after every block, i.e. also for data blocks (except for cycles)
SB3 Single block, fine	Machining stops after every machine block (also in cycles)

Precondition

A program must be selected for execution in "AUTO" or "MDA" mode.

Procedure



1. Press the "Prog. ctrl." softkey and select the desired variant in the "SBL" field.



2. Press the <SINGLE BLOCK> key.



3. Press the <CYCLE START> key.

   Depending on the execution variant, the first block will be executed. Then the machining stops.

   In the channel status line, the text “Stop: Block in single block ended" appears.



4. Press the <CYCLE START> key.

   Depending on the mode, the program will continue executing until the next stop.



5. Press the <SINGLE BLOCK> key again, if the machining is not supposed to run block-by-block.

   The key is deselected again.



   If you now press the <CYCLE START> key again, the program is executed to the end without interruption.

See also

Selecting a program (Page 131)

7.4 Displaying the current program block

7.4.1 Current block display

The window of the current block display shows the program blocks currently being executed.

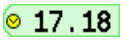
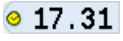
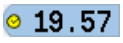
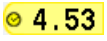
Display of current program

The following information is displayed in the running program:

- The workpiece name or program name is entered in the header line.
- The program block which is just being processed appears colored.

Display of the machining times

If you set that the machining times are to be recorded in the settings for automatic mode, the measured times are shown at the end of the line as follows:

Display	Meaning
Light green background 	Measured machining time of the program block (automatic mode)
Green background 	Measured machining time of the program group (automatic mode)
Light blue background 	Estimated machining time of the program block (simulation)
Blue background 	Estimated machining time of the program group (simulation)
Yellow background 	Wait time (automatic mode or simulation)

Highlighting of selected G code commands or keywords

In the program editor settings, you can specify whether selected G code commands are to be highlighted in color. The following colors are used as standard:

Display	Meaning
Blue font 	D, S, F, T, M and H functions
Red font 	"G0" motion command

Display	Meaning
Green font G1¶	"G1" motion command
Blue-green font G3¶	"G2" or "G3" motion command
Gray font ; Kommentar¶	Comment

Machine manufacturer

You can define further highlight colors in the "sleditorwidget.ini" configuration file.
Please refer to the machine manufacturer's instructions.

Editing a program directly

In the Reset state, you can edit the current program directly.

1. Press the <INSERT> key.
2. Place the cursor at the relevant position and edit the program block.
   Direct editing is only possible for G code blocks in the NC memory, not for external execution.
3. Press the <INSERT> key to exit the program and the edit mode again.

See also

Settings for the automatic mode (Page 187)

7.4.2 Displaying a basic block

If you want precise information about axis positions and important G functions during testing or program execution, you can call up the basic block display. This is how you check, when using cycles, for example, whether the machine is actually traversing.

Positions programmed by means of variables or R parameters are resolved in the basic block display and replaced by the variable value.

You can use the basic block display both in test mode and when machining the workpiece on the machine. All G code commands that initiate a function on the machine are displayed in the "Basic Blocks" window for the currently active program block:

- Absolute axis positions
- G functions for the first G group
- Other modal G functions
- Other programmed addresses
- M functions



Machine manufacturer
Please observe the information provided by the machine manufacturer.

Procedure

1. A program is selected for execution and has been opened in the "Machine" operating area.



2. Press the "Basic blocks" softkey.
   The "Basic Blocks" window opens.



3. Press the <SINGLE BLOCK> key if you wish to execute the program block by block.



4. Press the <CYCLE START> key to start the program execution.
   The axis positions to be approached, modal G functions, etc., are displayed in the "Basic Blocks" window for the currently active program block.



5. Press the "Basic blocks" softkey once again to hide the window again.

7.4.3 Display program level

You can display the current program level during the execution of a large program with several subprograms.

Several program run throughs

If you have programmed several program run throughs, i.e. subprograms are run through several times one after the other by specifying the additional parameter P, then during processing, the program runs still to be executed are displayed in the "Program Levels" window.

Program example

N10 subprogram P25

If, in at least one program level, a program is run through several times, a horizontal scroll bar is displayed that allows the run through counter P to be viewed in the righthand window section. The scroll bar disappears if multiple run-through is no longer applicable.

Display of program level

The following information will be displayed:

- Level number
- Program name
- Block number, or line number
- Remain program run throughs (only for several program run throughs)

Precondition

A program must be selected for execution in "AUTO" mode.

Procedure



Press the "Program levels" softkey.
The "Program levels" window appears.

7.5 Correcting a program

As soon as a syntax error in the part program is detected by the controller, program execution is interrupted and the syntax error is displayed in the alarm line.

Correction options

Depending on the state of the control system, you have various options of correcting the program.

- Stop state
  Only change lines that have not been executed
- Reset status
  Change all lines

Note

The "program correction" function is also available for execute from external; however, when making program changes, the NC channel must be brought into the reset state.

Precondition

A program must be selected for execution in "AUTO" mode.

Procedure

1. The program to be corrected is in the Stop or Reset mode.
2. 

Press the "Prog. corr." softkey.
   The program is opened in the editor.
   The program preprocessing and the current block are displayed. The current block is also updated in the running program, but not the displayed program section, i.e. the current block moves out of the displayed program section.
   If a subprogram is executed, it is not opened automatically.
3. Make the necessary corrections.
4. 

Press the "NC Execute" softkey.
   The system switches back to the "Machine" operating area and selects "AUTO" mode.
5. 

Press the "CYCLE START" key to resume program execution.

Note

When you exit the editor using the "Close" softkey, you return to the "Program manager" operating area.

7.6 Repositioning axes

After a program interruption in the automatic mode (e.g. after a tool breaks), you can move the tool away from the contour in manual mode.

The coordinates of the interrupt position will be saved. The distances traversed in manual mode are displayed in the actual value window. This path difference is called "REPOS offset".

Resuming program execution

You use the "REPOS" function to return the tool to the contour of the workpiece to continue executing the program.

The interrupt position is not passed as it is blocked by the control system.

The feedrate/rapid traverse override is in effect.

NOTICE
Risk of collision When repositioning, the axes move with the programmed feedrate and linear interpolation, i.e. in a straight line from the current position to the interrupt point. Therefore, you must first move the axes to a safe position in order to avoid collisions. If you do not use the "REPOS" function after a program interrupt and then traversing the axes in manual mode, then on changing to automatic mode and starting the machining process, the control automatically traverses the axes in straight lines back to where they were at point of interruption.

Requirement

The following prerequisites must be met when repositioning the axes:

- The program execution was interrupted using <CYCLE STOP>.
- The axes were moved from the interrupt point to another position in manual mode.

Procedure



1. Press the <REPOS> key.





2. Select the axes to be traversed one after the other.





3. Press the <+> or <-> key for the relevant direction.
   The axes are moved to the interrupt position.

7.7 Starting execution at a specific point

7.7.1 Use block search

If you want to perform only a certain section of a program on the machine, then you need not start the program from the beginning. You can start the program from a specified program block.

Applications

- Stopping or interrupting program execution
- Specify a target position, e.g. during remachining

Determining a search target

- User-friendly search target definition (search positions)
  - Direct specification of the search target by positioning the cursor in the selected program (main program)
    Note:
    It must be ensured during a block search that the correct tool is in the working position before commencing execution of the program.
  - Search target via text search
  - The search target is the interruption point (main program and subprogram)
    The function is only available if there is an interruption point. After a program interruption (CYCLE STOP, RESET or Power Off), the controller saves the coordinates of the interruption point.
  - The search target is the higher program level of the interruption point (main program and subprogram)
    A change of levels is only possible if an interruption point located in a sub-program is selected. It is thus possible to switch to the main program level and back to the level of the interruption point.
- Search pointer
  - Direct entry of the program path

    Note

    You can search for a specific point in subprograms with the search pointer if there is no interruption point.

Cascaded search

You can start another search from the "Search target found" state. Each time a search target is found, it is possible to continue cascading arbitrarily.

Note

Another cascaded block search can be started from the stopped program execution only if the search target has been found.

Requirements

- You have selected the desired program.
- The controller is in the reset state.
- The desired search mode is selected.

NOTICE
Risk of collision Pay attention to a collision-free start position and appropriate active tools and other technological values. If necessary, manually approach a collision-free start position. Select the target block considering the selected block search variant.

Toggling between search pointer and search positions



Press the "Search pointer" softkey again to exit the "Search Pointer" window and return to the "Program" window to define search positions.

- OR -



Press the "Back" softkey.

You have now exited the block search function.

Further information

Additional information on the block search function is provided in the Basic Functions Function Manual.

See also

Selecting a program (Page 131)

Block search mode (Page 146)

7.7.2 Continuing program from search target

Press the "CYCLE START" key twice to continue the program from the desired position.

- The first CYCLE START outputs the auxiliary functions collected during the search. The program is then in the Stop state.
- Before the second CYCLE START, you can use the "Overstore" function to create states that are required, but not yet available, for the further program execution.
  If the set position is not to be approached automatically after the program start, you can also traverse the tool manually from the current position to the set position by changing to JOG mode for the REPOS function

7.7.3 Simple search target definition

Requirement

The program is selected and the controller is in Reset mode.

Procedure



1. Press the "Block search" softkey.

2. Place the cursor on a particular program block.

   - OR -



   Press the "Find text" softkey, select the search direction, enter the search text and confirm with "OK".





3. Press the "Start search" softkey.

   The search starts. Your specified search mode will be taken into account.

   The current block will be displayed in the "Program" window as soon as the target is found.



4. If the located target (for example, when searching via text) does not correspond to the program block, press the "Start search" softkey again until you find your target.

   Press the <CYCLE START> key twice.

   Processing is continued from the defined position.

7.7.4 Defining an interruption point as search target

Requirement

A program was selected in "AUTO" mode and interrupted during execution through CYCLE STOP or RESET.

Procedure



1. Press the "Block search" softkey.



2. Press the "Interrupt point" softkey.
   The interruption point is loaded.





3. If the "Higher level" and "Lower level" softkeys are available, use these to change the program level.



4. Press the "Start search" softkey.
   The search starts. Your specified search mode will be taken into account.
   The search screen closes.
   The current block will be displayed in the "Program" window as soon as the target is found.



5. Press the <CYCLE START> key twice.
   The execution will continue from the interruption point.

7.7.5 Entering the search target via search pointer

Enter the program point which you would like to proceed to in the "Search Pointer" window.

Requirement

The program is selected and the controller is in the reset state.

Screen form

Each line represents one program level. The actual number of levels in the program depends on the nesting depth of the program.

Level 1 always corresponds to the main program and all other levels correspond to subprograms.

You must enter the target in the line of the window corresponding to the program level in which the target is located.

For example, if the target is located in the subprogram called directly from the main program, you must enter the target in program level 2.

The target specification must always be unique. This means, for example, that if the subprogram is called in the main program in two different places, you must also specify a target in program level 1 (main program).

Procedure



1. Press the "Block search" softkey.



2. Press the "Search pointer" softkey.
3. Enter the full path of the program as well as the subprograms, if required, in the input fields.



4. Press the "Start search" softkey.

   The search starts. Your specified search mode will be taken into account.

   The Search window closes. The current block will be displayed in the "Program" window as soon as the target is found.



5. Press the <CYCLE START> key twice.

   Processing is continued from the defined location.

Note

Interruption point

You can load the interruption point in search pointer mode.

7.7.6 Parameters for block search in the search pointer

Parameter	Meaning
Number of program level	
Program:	The name of the main program is automatically entered
Ext:	File extension
P:	Number of subprogram repetitions If a subprogram is performed several times, you can enter the number of the pass here at which processing is to be continued
Line:	Is automatically filled for an interruption point

Parameter	Meaning
Type	" " search target is ignored on this level N no. Block number Label Jump label Text string Subprg. Subprogram call Line Line number
Search target	Point in the program at which machining is to start

7.7.7 Block search mode

Set the desired search variant in the "Search Mode" window.

The set mode is retained when the control is shut down. When you activate the "Search" function after restarting the control, the current search mode is displayed in the title row.

Search variants

Block search mode	Meaning
With calculation - without approach	It is used in order to be able to approach a target position in any circumstance (e.g. tool change position). The end position of the target block or the next programmed position is approached using the type of interpolation valid in the target block. Only the axes programmed in the target block are moved. **Note**: If machine data 11450.1=1 is set, the rotary axes of the active swivel data record are pre-positioned after the block search.
With calculation - with approach	It is used to be able to approach the contour in any circumstance. The end position of the block prior to the target block is found with <CYCLE START>. The program runs in the same way as in normal program processing.
With calculation - skip extcall	This is used to speed-up a search with calculation when using EXTCALL programs: EXTCALL programs are not taken into account. **Notice**: Important information, e.g. modal functions, which are located in the EXTCALL program, are not taken into account. In this case, after the search target has been found, the program is not able to be executed. Such information should be programmed in the main program.

Block search mode	Meaning
Without calculation	For a quick search in the main program. Calculations will not be performed during the block search, i.e. the calculation is skipped up to the target block. All settings required for execution have to be programmed from the target block (e.g. feedrate, spindle speed, etc.).
With program test, without approach	Multi-channel block search with calculation (SERUPRO). All blocks are calculated during the block search. Absolutely no axis motion is executed, however, all auxiliary functions are output. The NC starts the selected program in the program test mode. If the NC reaches the specified target block in the actual channel, it stops at the beginning of the target block and deselects program test mode again. After continuing the program with NC start, the auxiliary functions of the target block are output. For single-channel systems, the coordination is supported with events running in parallel, e.g. synchronized actions. **Note** The search speed depends on MD settings.



Machine manufacturer
Please observe the information provided by the machine manufacturer.

More information

More information on configuring the block search function is provided in the SINUMERIK Operate Commissioning Manual.

7.8 Controlling the program run

7.8.1 Program control

You can change the program sequence in the "AUTO" and "MDA" modes.

Abbreviation/program control	**Mode of operation**
PRT No axis motion	The program is started and executed with auxiliary function outputs and dwell times. In this mode, the axes are not traversed. The programmed axis positions and the auxiliary function outputs are controlled this way. **Note**: You can activate program execution without any axis motion using the "Dry run feedrate".
DRY Dry run feedrate	The traversing velocities programmed in conjunction with G1, G2, G3, CIP and CT are replaced by a defined dry run feedrate. The dry run feedrate also applies instead of the programmed revolutional feedrate. **Notice**: Do not machine any workpieces when "Dry run feedrate" is active because the altered feedrates might cause the permissible tool cutting rates to be exceeded and the workpiece or machine tool could be damaged.
RG0 Reduced rapid traverse	In the rapid traverse mode, the traversing speed of the axes is reduced to the percentage value entered in RG0. **Note**: You define the reduced rapid traverse in the settings for automatic operation.
M01 Programmed stop 1	The processing of the program stops at every block in which supplementary function M01 is programmed. In this way you can check the already obtained result during the processing of a workpiece. **Note**: In order to continue executing the program, press the <CYCLE START> key again.
Programmed stop 2 (e.g. M101)	The processing of the program stops at every block in which the "Cycle end" is programmed (e.g. with M101). **Note**: In order to continue executing the program, press the <CYCLE START> key again. Note: The display can be changed. Refer to the machine manufacturer's specifications.
DRF Handwheel offset	While machining in the automatic mode, an additional offset can be entered using the electronic handwheel for the following purposes: • DRF offset in the basis coordinate system (BCS) for additive work offset of geometry and special axes. For example, to correct tool wear within an NC block. • DRF offset in the tool coordinate system (TCS) to enter online tool length compensations (handwheel override in the tool direction). **Note**: The DRF offset in the tool coordinate system (TCS) is only possible when orientation transformation (TRAORI) / kinematic chain are active or for an active tool carrier that can be oriented (TCARR).

Abbreviation/program control	Mode of operation
SB	Individual blocks are configured as follows. • Single block, coarse: The program stops only after blocks which perform a machine function. • Data block: The program stops after each block. • Single block, fine: The program also stops only after blocks which perform a machine function in cycles. Select the desired setting using the <SELECT> key.
SKP	Skip blocks are skipped during machining.
MRD	In the program, the measurement results screen display is activated while machining.
CST Configured stop	Program processing stops at the points you defined as relevant to stop before the program started. These may be, for example, especially critical points, at which you wan to check the correctness of the sequence or exclude collisions. The following NC function transitions can be activated as default setting in window "Program control" as stop relevant: • Transition G0-G1 • Transition G1-G0 • Transition G0-G0 • Set to 0% for stop override (use of the power ride) • WAIT marker (for several channels): – "WAITM" - wait until mark is reached in all specified channels, – "WAITMC" like WAITM, but without exact stop, – "WAITE" - wait for program end in the specified channels. Program processing runs in all channels up to the synchronization point. Subsequently, further machining is continued with NC start. You can also define NC functions (auxiliary functions, cycle calls, T-preselection) as NC function transitions. **Note**: The option to stop can be disabled at special locations. For example, this can be the case in Siemens machining cycles or manufacturer cycles. Refer to the information provided by the machine manufacturer.

Activating program control

You can control the program sequence however you wish by selecting and clearing the relevant checkboxes.

Display / response of active program controls

If program control is activated, the abbreviation of the corresponding function appears in the status display as feedback response.

Procedure



1. Select the "Machine" operating area.





2. Press the <AUTO> or <MDI> key.



3. Press the "Prog. ctrl." softkey.
   The "Program Control" window opens.

7.8.2 Use Powerride for program control

"Powerride" is an additional rotary/push button on the machine control panel. The Powerride combines the override value and the NC start and facilitates program start-up and single block machining.



Machine manufacturer
Please observe the information provided by the machine manufacturer.

By using the Powerride, you can automate rotation of the Override back to 0% for the single block and also for the configured stop.

The Override value can be reduced to 0% using the PLC software.

To use the Powerride, select the "Set to 0% on Stop Override" check box in the "Configured Stop Override" window.

Requirement

- The machine is in reset mode and in the "Automatic" operating mode.
- "Configured stop" is activated.

Procedure

1. Press the Powerride switch.
   The control stops at the first block end for which the configured stop is configured.
   The PLC sets the Override to 0%.
2. Press the Powerride switch.
   The new block is activated.
3. Slowly turn up the Override switch.
   The machine begins to move.

7.8.3 Skip blocks

You can skip program blocks that are not to be executed every time the program runs.

The skip blocks are identified by placing a "/" (forward slash) or "/x" (x = number of skip level) character in front of the block number. You have the option of hiding several block sequences.

The statements in the skipped blocks are not executed. The program continues with the next block, which is not skipped.

The number of skip levels that can be used depends on a machine datum.



Machine manufacturer
Please observe the information provided by the machine manufacturer.

Skip levels, activate

Select the corresponding checkbox to activate the desired skip level.

Note

The "Program Control - Skip Blocks" window is only available when more than one skip level is set up.

7.9 Overstore

With overstore, you have the option of executing technological parameters (for example, auxiliary functions, axis feed, spindle speed, programmable instructions, etc.) before the program is actually started. The program instructions act as if they are located in a normal part program. These program instructions are, however, only valid for one program run. The part program is not permanently changed. When next started, the program will be executed as originally programmed.

After a block search, the machine can be brought into another state with overstore (e.g. M function, tool, feed, speed, axis positions, etc.), in which the normal part program can be successfully continued.

Requirement

The program to be corrected is in the Stop or Reset mode.

Procedure



1. Select the "Machine" operating area in the "AUTO" mode.





2. Press the "Overstore" softkey.
   The "Overstore" window opens.
3. Enter the required data and NC block.



4. Press the <CYCLE START> key.
   The blocks you have entered are stored. You can observe execution in the "Overstore" window.
   After the entered blocks have been executed, you can append blocks again.
   You cannot change the operating mode while you are in overstore mode.



5. Press the "Back" softkey.
   The "Overstore" window closes.



6. Press the <CYCLE START> key again.
   The program selected before overstoring continues to run.

Note

Block-by-block execution

The <SINGLE BLOCK> key is also active in the overstore mode. If several blocks are entered in the overstore buffer, then these are executed block-by-block after each NC start

Deleting blocks



Press the "Delete blocks" softkey to delete program blocks you have entered.

7.10 Editing a program

With the editor, you are able to render, supplement, or change part programs.

Note

Maximum block length

The maximum block length is 512 characters.

Calling the editor

- The editor is started via the "Program correction" softkey in the "Machine" operating area. You can directly change the program by pressing the <INSERT> key.
- The editor is called via the "Open" softkey as well as with the <INPUT> or <Cursor right> key in the "Program manager" operating area.
- The editor opens in the "Program" operating area with the last executed part program, if this was not explicitly exited via the "Close" softkey.

  Note

  - Please note that the changes to programs saved in the NC memory take immediate effect.
  - If you are editing on a local drive or external drives, you can also exit the editor without saving, depending on the setting. Programs in the NC memory are always automatically saved.
  - Exit the program correction mode using the "Close" softkey to return to the "Program manager" operating area.

See also

Editor settings (Page 162)

Opening and closing a program (Page 329)

Correcting a program (Page 138)

7.10.1 Searching in programs

You can use the search function to quickly arrive at points where you would like to make changes, e.g. in very large programs.

Various search options are available that enable selective searching.

Search options

- Whole words
  Activate this option and enter a search term if you want to search for texts/terms that are present as words in precisely this form.
  If, for example, you enter the search term "Finishing tool", only single "Finishing tool" terms are displayed. Word combinations such as "Finishing tool_10" are not found.
- Exact expression
  Activate this option if you wish to search for terms with characters, which can also be used as place holders for other characters, e.g. "?" and "*".

Note

Search with place holders

When searching for specific program locations, you have the option of using place holders:

- "*": Replaces any character string
- "?": Replaces any character

Precondition

The desired program is opened in the editor.

Procedure



1. Press the "Search" softkey.
   A new vertical softkey bar appears.

   The "Search" window opens at the same time.
2. Enter the desired search term in the "Text" field.
3. Select "Whole words" if you want to search for whole words only.
   - OR -
   Activate the "Exact expression" checkbox if, for example, you want to search for place holders ("*", "?") in program lines.



4. Position the cursor in the "Direction" field and choose the search direction (forward, backward) with the <SELECT> key.



5. Press the "OK" softkey to start the search.

   If the text you are searching for is found, the corresponding line is highlighted.



6. Press the "Continue search" softkey if the text located during the search does not correspond to the point you are looking for.

- OR -



Press the "Cancel" softkey when you want to cancel the search.

Further search options

Softkey	Function
Go to start	The cursor is set to the first character in the program.
Go to end	The cursor is set to the last character in the program.

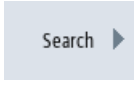
7.10.2 Replacing program text

You can find and replace text in one step.

Precondition

The desired program is opened in the editor.

Procedure



1. Press the "Search" softkey.
   A new vertical softkey bar appears.



2. Press the "Find and replace" softkey.
   The "Find and Replace" window appears.
3. In the "Text" field, enter the term you are looking for and in the "Replace with" field, enter the text you would like to insert automatically during the search.



4. Position the cursor in the "Direction" field and choose the search direction (forward, backward) with the <SELECT> key.



5. Press the "OK" softkey to start the search.
   If the text you are searching for is found, the corresponding line is highlighted.



6. Press the "Replace" softkey to replace the text.

- OR -



Press the "Replace all" softkey to replace all text in the file that corresponds to the search term.

- OR -



Press the "Continue search" softkey if the text located during the search should not be replaced.

- OR -



Press the "Cancel" softkey when you want to cancel the search.

Note

Replacing texts

- Read-only lines (;*RO*)
  If hits are found, the texts are not replaced.
- Contour lines (;*GP*)
  If hits are found, the texts are replaced as long as the lines are not read-only.
- Hidden lines (;*HD*)
  If hidden lines are displayed in the editor and hits are found, the texts are replaced as long as the lines are not read-only. Hidden lines that are not displayed, are not replaced.

See also

Editor settings (Page 162)

7.10.3 Copying/pasting/deleting program blocks

In the editor, you edit both basic G code as well as program steps such as cycles, blocks and subprogram calls.

Inserting program blocks

The editor responds depending on what type of program block you insert.

- If you insert a G code, then the program block is directly inserted where the write mark is located.
- If you insert a program step, then the program block is always inserted at the next block, independent of the position of the write mark within the actual line. This is necessary as a cycle call always requires its own line.
  This behavior is in all applications, irrespective of whether the program step is inserted with a screen form using "Accept" or "Insert" is used as editor function.

Note

Cutout program step and reinsert

- If you cut out a program step at a specific location and you then directly reinsert it again, the sequence changes.
- Press the shortcut (key combination) <CTRL> + <Z> to undo what you have cut out.

Precondition

The program is opened in the editor.

Procedure



1. Press the "Mark" softkey.

   - OR -



   Press the <SELECT> key.

2. Select the desired program blocks with the cursor or mouse.



3. Press the "Copy" softkey in order to copy the selection to the buffer memory.



4. Place the cursor on the desired insertion point in the program and press the "Paste" softkey.

   The content of the buffer memory is pasted.

   - OR -



   Press the "Cut" softkey to delete the selected program blocks and to copy them into the buffer memory.

   Note: When editing a program, you cannot copy or cut more than 1024 lines. While a program that is not on the NC is opened (progress display less than 100%), you cannot copy or cut more than 10 lines or insert more than 1024 characters.

Numbering the program blocks

If you have selected the "Automatic numbering" option for the editor, then the newly added program blocks are allocated a block number (N number).

The following rules apply:

- When creating a new program, the first line is allocated the "first block number".
- If, up until now, the program had no N number, then the program block inserted is allocated the starting block number defined in the "First block number" input field.
- If N numbers already exist before and after the insertion point of a new program block, then the N number before the insertion point is incremented by 1.
- If there are no N numbers before or after the insertion point, then the maximum N number in the program is increased by the "increment" defined in the settings.

Note:

After exiting the program, you have the option of renumbering the program blocks.

Note

The buffer memory contents are retained even after the editor is closed, enabling you to paste the contents in another program.

Note

Copy/cut current line

To copy and cut the current line where the cursor is positioned, it is not necessary to mark or select it. You have the option of making the "Cut" softkey only operable for marked program sections via editor settings.

See also

Opening additional programs (Page 161)

Editor settings (Page 162)

7.10.4 Renumbering a program

You can modify the block numbering of programs opened in the editor at a later point in time.

Precondition

The program is opened in the editor.

Procedure



1. Press the ">>" softkey.
   A new vertical softkey bar appears.



2. Press the "Renumber" softkey.
   The "Renumbering" window appears.
3. Enter the values for the first block number and the increment to be used for numbering.



4. Press the "OK" softkey.
   The program is renumbered.

Note

- If you only want to renumber a section, before the function call, select the program blocks whose block numbering you want to edit.
- When you enter a value of "0" for the increment size, then all of the existing block numbers are deleted from the program and/or from the selected range.

7.10.5 Creating a program block

In order to structure programs to achieve a higher degree of transparency, you have the option of combining several G-code blocks to form program blocks.

Program blocks can be created in two stages. This means that you can form additional blocks within a block (nesting).

You then have the option of opening and closing these blocks depending on your requirement.

Settings for a program block

Display	Meaning
Text	• Block designation
Spindle	• Selecting the spindle Defines at which spindle a program block is to be executed.
Addit. run-in code	• Yes For the case that the block is not executed, as the specified spindle should not be processed, then it is possible to temporarily activate a so-called "Addit. run-in code". • No
Automat. Retraction	• Yes Block start and block end are moved to the tool change point, i.e. the tool is brought into a safe range. • No

Procedure



1. Select the "Program Manager" operating area.





2. Select the storage location and create a program or open a program.
   The program editor opens.

3. Select the required program blocks that you wish to combine to form a block.



4. Press the "Build block" softkey.
   The "Build Block" window opens.



5. Enter a designation for the block, assign the spindle, if required, select the additional run-in code and the automatic retraction and then press the "OK" softkey.

Opening and closing blocks



6. Press the ">>" and "View" softkeys.





7. Press the "Open blocks" softkey if you wish to display the program with all blocks.



8. Press the "Close blocks" softkey if you wish to display the program again in a structured form.

Remove block

9. Open the block.
10. Position the cursor at the end of the block.



11. Press the "Remove block" softkey.

Note

You can also open and close blocks with the mouse or using the cursor keys:

- <Cursor right> opens the block where the cursor is positioned
- <Cursor left> closes the block if the cursor is positioned at the beginning or end of the block
- <ALT> and <Cursor left> closes the block if the cursor is located within the block

Note

DEF statements in program blocks or block generation in the DEF part of a part program / cycle are not permitted.

7.10.6 Opening additional programs

You have the option of viewing and editing several programs simultaneously in the editor.

For instance, you can copy program blocks or machining steps of a program and paste them into another program.

Opening several programs

You have the option of opening up to ten program blocks.



1. In the program manager, select the programs that you wish to open and view in the multiple editor and then press on the "Open" softkey.
   The editor is opened and the first two programs are displayed.



2. Press the <NEXT WINDOW> key to change to the next opened program.



3. Press the "Close" softkey to close the actual program.

Note

Pasting program blocks

JobShop machining steps cannot be copied into a G code program.

Precondition

You have opened a program in the editor.

Procedure



1. Press the ">>" and "Open additional program" softkeys.



   The "Select Additional Program" window is opened.

2. Select the program or programs that you wish to display in addition to the already opened program.



3. Press the "OK" softkey.

   The editor opens and displays both programs next to each another.

See also

Copying/pasting/deleting program blocks (Page 157)

7.10.7 Editor settings

Enter the default settings in the "Settings" window that are to take effect automatically when the editor is opened.

Defaults

Setting	Meaning
Number automatically	• Yes: A new block number will automatically be assigned after every line change. In this case, the specifications provided under "First block number" and "Increment" are applicable. • No: No automatic numbering
First block number	Specifies the starting block number of a newly created program. The field is only visible when "Yes" is selected under "Number automatically".
Increment	Defines the increment used for the block numbers. The field is only visible when "Yes" is selected under "Number automatically".
Show hidden lines	• Yes: Hidden lines marked with "*HD" (hidden) will be displayed. • No: Lines marked with ";*HD*" will not be displayed. **Note**: Only visible program lines are taken into account with the "Search" and "Search and Replace" functions.
Display block end as symbol	The "LF" (line feed) symbol ¶ is displayed at the block end.
Line break	• Yes: Long lines are broken and wrapped around. • No: If the program includes long lines, then a horizontal scrollbar is displayed. You can move the section of the screen horizontally to the end of the line.
Line break also in cycle calls	• Yes: If the line of a cycle call becomes too long, then it is displayed over several lines. • No: The cycle call is truncated. The field is only visible if "Yes" is entered under "Line break".
Visible programs	• 1 - 10 Select how many programs can be displayed next to one another in the editor. • Auto Specifies that the number of programs entered in a job list or up to ten selected programs will be displayed next to each other.
Width per program w. focus	Here, you enter the width of the program that has the input focus in the editor as a percentage of the window width.
Save automatically	• Yes: The changes are saved automatically when you change to another operating area. • No: You are prompted to save when changing to another operating area. Save or reject the changes with the "Yes" and "No" softkeys. **Note**: Only for local and external drives.
Only cut after marking	• Yes: Parts of programs can only be cutout when program lines have been selected, i.e. the "Cutout" softkey only then is active. • No: The program line, in which the cursor is positioned, can be cut out without having to select it.

Setting	Meaning
Determine machining times	Defines which program runtimes are determined in the simulation or in automatic mode: • Off Program runtimes are not determined. • Block-by-block: The runtimes are determined for each program block. • Non-modal: The runtimes are determined at the NC block level. **Note**: You also have the option of displaying the cumulative times for blocks. Please observe the information provided by the machine manufacturer. After the simulation or after executing the program, the required machining times are displayed in the editor.
Save machining times	Specifies how the machining times determined are processed. • Yes A subdirectory with the name "GEN_DATA.WPD" is created in the directory of the part program. There, the machining times determined are saved in an ini file together with the name of the program. The machining times are displayed again when the program or job list are reloaded. • No The machining times that have been determined are only displayed in the editor.
Record tools	Defines whether the tool data is recorded. • Yes Recording takes place during processing. The data is stored in a TTD file (Tool Time Data). The TTD file is located in the directory of the associated part program. • No The tool data is not recorded.
Display cycles as machining step	• Yes: The cycle calls in the G code programs are displayed as plain text. • No: The cycle calls in the G code programs are displayed in the NC syntax.

Setting	Meaning
Highlight selected G code commands	Defines the display of G code commands. • No All G code commands are displayed in the standard color. • Yes Selected G code commands or keywords are highlighted in color. Define the rules for the color assignment in the sleditorwidget.ini configuration file. **Note:** Please observe the information provided by the machine manufacturer. **Note** This setting also has an effect on the current block display.
Font size	Defines the font size for the editor and the display of the program sequence. • auto If you open a second program, then the smaller font size is automatically used. • normal (16) - character height in pixels Standard font size that is displayed with the appropriate screen resolution. • small (14) - character height in pixels More content is displayed in the editor. **Note** This setting also has an effect on the current block display.

Note

All entries that you make here are effective immediately.

Requirement

You have opened a program in the editor.

Procedure



1. Select the "Program" operating area.



2. Press the "Edit" softkey.





3. Press the ">>" and "Settings" softkeys.
   The "Settings" window opens.

4. Make the required changes.



5. Press the "Del. machining times" softkey if you wish to delete the machining times.

   The machining times that have been determined are deleted from the editor as well as from the actual block display. If the machining times are saved to an ini file, then this file is also deleted.



6. Press the "OK" softkey to confirm the settings.

See also

Replacing program text (Page 156)

7.11 Display and edit user variables

The defined user data may be displayed in lists.

User variables

The following variables can be defined:

- Global arithmetic parameters (RG)
- Channel-specific arithmetic parameters (R parameters)
- Global user data (GUD) is valid in all programs
- Local user variables (LUD) are valid in the program where they have been defined.
- Program-global user variables (PUD) are valid in the program in which they have been defined, as well as in all of the subprograms called by this program

Channel-specific user data can be defined with a different value for each channel.

Entering and displaying parameter values

Up to 15 positions (including decimal places) are evaluated. If you enter a number with more than 15 places, it will be written in exponential notation (15 places + EXXX).

LUD or PUD

Only local or program-global user data can be displayed at one time.

Whether the user data are available as LUD or PUD depends on the current control configuration.



Machine manufacturer
Please observe the information provided by the machine manufacturer.

Note

Reading and writing variables protected

Reading and writing of user data are protected via a keyswitch and protection levels.

Comments

You have the option of entering a comment for global and channel-specific arithmetic parameters.

Searching for user data

You may search for user data within the lists using any character string.

Further information

Additional information on the user variables is provided in the Programming Manual NC Programming.

7.11.1 Global R parameters

Global R parameters are arithmetic parameters, which exist in the control itself, and can be read or written to by all channels.

You use global R parameters to exchange information between channels, or if global settings are to be evaluated for all channels.

These values are retained after the controller is switched off.

Comments

You can save comments in the "Global R parameters with comments" window.

These comments can be edited. You have the option of either individually deleting these comments, or using the delete function.

These comments are retained after the control is switched off.

Number of global R parameters

The number of global R parameters is defined in a machine data element.

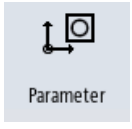
Range: RG[0]– RG[999] (dependent on the machine data).

There are no gaps in the numbering within the range.



Machine manufacturer
Please observe the information provided by the machine manufacturer.

Procedure



1. Select the "Parameter" operating area.



2. Press the "User variable" softkey.



3. Press the "Global R parameters" softkey.
   The "Global R parameters" window opens.

Display comments



1. Press the ">>" and "Display comments" softkeys.
   The "Global R parameters with comments " window opens.





2. Press the "Display comments" softkey once again to return to the "Global R parameters" window.

Deleting R parameters and comments



1. Press the ">>" and "Delete" softkeys.
   The "Delete global R parameters" window opens.



2. In fields "from global R parameters" and "to global R parameters", select the global R parameters whose values you wish to delete.
   - OR -



   Press the "Delete all" softkey.

3. Activate the checkbox "also delete comments" if the associated comments should also be automatically deleted.



4. Press the "OK" softkey.

- A value of 0 is assigned to the selected global R parameters – or to all global R parameters.
- The selected comments are also deleted.

7.11.2 R parameters

R parameters (arithmetic parameters) are channel-specific variables that you can use within a G code program. G code programs can read and write R parameters.

These values are retained after the controller is switched off.

Comments

You can save comments in the "R parameters with comments" window.

These comments can be edited. You have the option of either individually deleting these comments, or using the delete function.

These comments are retained after the control is switched off.

Number of channel-specific R parameters

The number of channel-specific R parameters is defined in a machine data element.

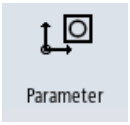
Range: R0-R999 (dependent on machine data).

There are no gaps in the numbering within the range.



Machine manufacturer
Please observe the information provided by the machine manufacturer.

Procedure



1. Select the "Parameter" operating area.



2. Press the "User variable" softkey.



3. Press the "R variables" softkey.
   The "R parameters" window appears.

Display comments



1. Press the ">>" and "Display comments" softkeys.
   The "R parameters with comments " window opens.





2. Press the "Display comments" softkey once again to return to the "R parameters" window.

Delete R variables



1. Press the ">>" and "Delete" softkeys.
   The "Delete R parameters" window appears.



2. In fields "from R parameters" and "to R parameters", select the R parameters whose values you wish to delete.
   - OR -



   Press the "Delete all" softkey.

3. Activate the checkbox "also delete comments" if the associated comments should also be automatically deleted.



4. Press the "OK" softkey.

- A value of 0 is assigned to the selected R parameters or to all R parameters.
- The selected comments are also deleted.

7.11.3 Displaying global user data (GUD)

Global user variables

Global GUDs are NC global user data (**G**lobal **U**ser **D**ata) that remains available after switching the machine off.

GUDs apply in all programs.

Definition

A GUD variable is defined with the following:

- Keyword DEF
- Range of validity NCK
- Data type (INT, REAL, ...)
- Variable names
- Value assignment (optional)

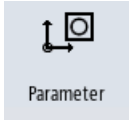
Example

DEF NCK INT ZAEHLER1 = 10

GUDs are defined in files with the ending DEF. The following file names are reserved for this purpose:

File name	Meaning
MGUD.DEF	Definitions for global machine manufacturer data
UGUD.DEF	Definitions for global user data
GUD4.DEF	User-definable data
GUD8.DEF, GUD9.DEF	User-definable data

Procedure



1. Select the "Parameter" operating area.



2. Press the "User variable" softkey.



3. Press the "Global GUD" softkeys.

   The "Global User Variables" window is displayed. A list of the defined UGUD variables will be displayed.

   - OR -



Press the "GUD selection" softkey and the "SGUD" to "GUD6" softkeys if you wish to display SGUD, MGUD, UGUD as well as GUD4 to GUD 6 of the global user variables.

- OR -



Press the "GUD selection" and ">>" softkeys as well as the "GUD7" to "GUD9" softkeys if you want to display GUD 7 to GUD 9 of the global user variables.

Note

After each start-up, a list with the defined UGUD variables is displayed in the "Global User Variables" window.

7.11.4 Displaying channel GUDs

Channel-specific user variables

Like the GUDs, channel-specific user variables are applicable in all programs for each channel. However, unlike GUDs, they have specific values.

Definition

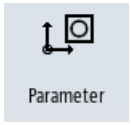
A channel-specific GUD variable is defined with the following:

- Keyword DEF
- Range of validity CHAN
- Data type
- Variable names
- Value assignment (optional)

Example

DEF CHAN REAL X_POS = 100.5

Procedure



1. Select the "Parameter" operating area.



2. Press the "User variable" softkey.





3. Press the "Channel GUD" and "GUD selection" softkeys.

A new vertical softkey bar appears.





4. Press the "SGUD" ... "GUD6" softkeys if you want to display the SGUD, MGUD, UGUD as well as GUD4 to GUD 6 of the channel-specific user variables.

- OR -



Press the "Continue" softkey and the "GUD7" ... "GUD9" softkeys if you want to display GUD 7 and GUD 9 of the channel-specific user variables.



7.11.5 Displaying local user data (LUD)

Local user variables

LUDs are only valid in the program or subprogram in which they were defined.

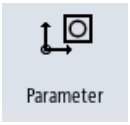
The controller displays the LUDs after the start of program processing. The display is available until the end of program processing.

Definition

A local user variable is defined with the following:

- Keyword DEF
- Data type
- Variable names
- Value assignment (optional)

Procedure



1. Select the "Parameter" operating area.



2. Press the "User variable" softkey.



3. Press the "Local LUD" softkey.

7.11.6 Displaying program user data (PUD)

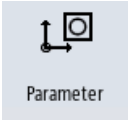
Program-global user variables

PUDs are global part program variables (**P**rogram **U**ser **D**ata). PUDs are valid in all main programs and subprograms, where they can also be written and read.



Machine manufacturer
Please refer to the machine manufacturer's specifications.

Procedure



1. Select the "Parameter" operating area.



2. Press the "User variable" softkey.

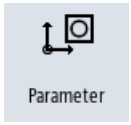


3. Press the "Program PUD" softkey.

7.11.7 Searching for user variables

You can search for R parameters and user variables.

Procedure



1. Select the "Parameter" operating area.



2. Press the "User variable" softkey.





3. Press the "R parameters", "Global GUD", "Channel GUD", "Local GUD" or "Program PUD" softkeys to select the list in which you would like to search for user variables.



4. Press the "Search" softkey.
   The "Search for R Parameters" or "Search for User Variables" window opens.



5. Enter the desired search term and press "OK".

   The cursor is automatically positioned on the R parameters or user variables you are searching for, if they exist.

By editing a DEF/MAC file, you can alter or delete existing definition/macro files or add new ones.

Procedure



1. Select the "Start-up" operating area.



2. Press the "System data" softkey.

3. In the data tree, select the "NC data" folder and then open the "Definitions" folder.
4. Select the file you want to edit.
5. Double-click the file.

   - OR -



   Press the "Open" softkey.

   - OR -



   Press the <INPUT> key.

   - OR -



   Press the <Cursor right> key.

   The selected file is opened in the editor and can be edited there.

6. Define the desired user variable.



7. Press the "Exit" softkey to close the editor.

Activating user variables



1. Press the "Activate" softkey.

   A prompt is displayed.
2. Select whether the current values in the definition files should be retained
   - OR -
   Select whether the current values in the definition files should be deleted. This will overwrite the definition files with the initial values.



3. Press the "OK" softkey to continue the process.

7.12 Display G and auxiliary functions

7.12.1 Selected G functions

16 selected G groups are displayed in the "G Function" window.

Within a G group, the G function currently active in the controller is displayed.

Some G codes (e.g. G17, G18, G19) are immediately active after switching the machine control on.

Which G codes are always active depends on the settings.



Machine manufacturer
Please observe the information provided by the machine manufacturer.

G groups displayed by default

Group	Meaning
G group 1	Modally active motion commands (e.g. G0, G1, G2, G3)
G group 2	Non-modally active motion commands, dwell time (e.g. G4, G74, G75)
G group 3	Programmable offsets, working area limitations and pole programming (e.g. TRANS, ROT, G25, G110)
G group 6	Plane selection (e.g. G17, G18)
G group 7	Tool radius compensation (e.g. G40, G42)
G group 8	Settable work offset (e.g. G54, G57, G500)
G group 9	Offset suppression (e.g. SUPA, G53)
G group 10	Exact stop - continuous-path mode (e.g. G60, G641)
G group 13	Workpiece dimensions, inch/metric (e.g. G70, G700)
G group 14	Workpiece dimensioning absolute/incremental (G90)
G group 15	Feed type (e.g. G93, G961, G972)
G group 16	Feedrate override at inside and outside curvature (e.g. CFC)
G group 21	Acceleration profile (e.g. SOFT, DRIVE)
G group 22	Tool offset types (e.g. CUT2D, CUT2DF)
G group 29	Radius/diameter programming (e.g. DIAMOF, DIAMCYCOF)
G group 30	Compressor on/off (e.g. COMPOF)

G groups displayed by default (ISO code)

Group	Meaning
G group 1	Modally active motion commands (e.g. G0, G1, G2, G3)
G group 2	Non-modally active motion commands, dwell time (e.g. G4, G74, G75)
G group 3	Programmable offsets, working area limitations and pole programming (e.g. TRANS, ROT, G25, G110)
G group 6	Plane selection (e.g. G17, G18)

Group	Meaning
G group 7	Tool radius compensation (e.g. G40, G42)
G group 8	Settable work offset (e.g. G54, G57, G500)
G group 9	Offset suppression (e.g. SUPA, G53)
G group 10	Exact stop - continuous-path mode (e.g. G60, G641)
G group 13	Workpiece dimensions, inch/metric (e.g. G70, G700)
G group 14	Workpiece dimensioning absolute/incremental (G90)
G group 15	Feed type (e.g. G93, G961, G972)
G group 16	Feedrate override at inside and outside curvature (e.g. CFC)
G group 21	Acceleration profile (e.g. SOFT, DRIVE)
G group 22	Tool offset types (e.g. CUT2D, CUT2DF)
G group 29	Radius/diameter programming (e.g. DIAMOF, DIAMCYCOF)
G group 30	Compressor on/off (e.g. COMPOF)

Procedure



1. Select the "Machine" operating area.



2. Press the <JOG>, <MDI> or <AUTO> key.

...





3. Press the "G functions" softkey.
   The "G Functions" window is opened.



4. Press the "G functions" softkey again to hide the window.

The G groups selection displayed in the "G Functions" window may differ.



Machine manufacturer
Please observe the information provided by the machine manufacturer.

Further information

Additional information on configuring the displayed G groups is provided in the SINUMERIK Operate Commissioning Manual.

7.12.2 All G functions

All G groups and their group numbers are listed in the "G Functions" window.

Within a G group, only the G function currently active in the controller is displayed.

Additional information in the footer

The following additional information is displayed in the footer:

- Actual transformations

Display	Meaning
TRANSMIT	Polar transformation active
TRACYL	Cylinder surface transformation active
TRAORI	Orientation transformation active
TRAANG	Inclined axis transformation active
TRACON	Cascaded transformation active For TRACON, two transformations (TRAANG and TRACYL or TRAANG and TRANSMIT) are activated in succession.

- Current work offsets
- Spindle speed
- Path feedrate
- Active tool

7.12.3 G functions for mold making

In the window "G functions", important information for machining free-form surfaces can be displayed using the "High Speed Settings" function (CYCLE832).

High-speed cutting information

In addition to the information that is provided in the "All G functions" window, the following programmed values of the following specific information is also displayed:

- CTOL
- OTOL
- CTOLG0
- OTOLG0

The tolerances for G0 are only displayed if they are active.

Particularly important G groups are highlighted.

You have the option to configure which G functions are highlighted.

More information

More information on the contour / orientation tolerance is provided in the Basic Functions Function Manual.

Procedure



1. Select the "Machine" operating area





2. Press the <JOG>, <MDI> or <AUTO> key.





3. Press the ">>" and "All G functions" softkeys.
   The "G Functions" window is opened.

7.12.4 Auxiliary functions

Auxiliary functions include M and H functions preprogrammed by the machine manufacturer, which transfer parameters to the PLC to trigger reactions defined by the manufacturer.

Displayed auxiliary functions

Up to five current M functions and three H functions are displayed in the "Auxiliary Functions" window.

Procedure



1. Select the "Machine" operating area.



2. Press the <JOG>, <MDA> or <AUTO> key.

...





3. Press the "H functions" softkey.
   The "Auxiliary Functions" window opens.



4. Press the "H functions" softkey again to hide the window again.

7.13 Displaying superimpositions

In window "Overrides", override motion can be displayed using the online tool length compensation and DRF offsets.

Input field	Meaning
Tool	Actual tool compensation offset using the online tool length compensation ($AA_TOFF)
Min	Lower limit of the tool compensation offset $AA_TOFF that can be entered using setting data SD42972 $SC_TOFF_LIMIT_MINUS
Max	Upper limit of the tool compensation offset $AA_TOFF that can be entered using setting data SD42970 $SC_TOFF_LIMIT
DRF	DRF offset in the basis coordinate system (BCS) for additive work offset of geometry and special axes

The selection of values displayed in the "Override" window may differ.



Machine manufacturer
Please observe the information provided by the machine manufacturer.

Procedure



1. Select the "Machine" operating area.



2. Press the <AUTO>, <MDI> or <JOG> key.

...





3. Press the ">>" and "Override" softkeys.
   The "Override" window opens.





4. Enter the required new minimum and maximum values for override and press the <INPUT> key to confirm your entries.
   Note:
   You can only change the override values in "JOG" mode.



5. Press the "Override" softkey again to hide the window.

7.14 Displaying status of synchronized actions

You can display status information for diagnosing synchronized actions in the "Synchronized Actions" window.

You get a list with all currently active synchronized actions.

In this list, the synchronized action programming is displayed in the same form as in the part program.

Synchronized actions

Status of synchronized actions

You can see the status of the synchronized actions in the "Status" column.

- Waiting
- Active
- Blocked
- Completed

Non-modal synchronized actions can only be identified by their status display. They are only displayed during execution.

Synchronization types

Synchronization types	Meaning
ID=n	Modal synchronized actions in the automatic mode up to the end of program, local to program; n = 1... 254
IDS=n	Static synchronized actions, modally effective in every operating type, also beyond the end of program; n = 1... 254
Without ID/IDS	Non-modal synchronized actions in the automatic mode

Note

Numbers from the number range 1 to 254 can only be assigned once, irrespective of the identification number.

Display of synchronized actions

Using softkeys, you have the option of restricting the display to activated synchronized actions.

Further information: Synchronized Actions Function Manual

Procedure



1. Select the "Machine" operating area.





2. Press the <AUTO>, <MDA> or <JOG> key.





3. Press the menu forward key and the "Synchron." softkey.
   The "Synchronized Actions" window appears.
   You obtain a display of all activated synchronized actions.



4. Press the "ID" softkey if you wish to hide the modal synchronized actions in the automatic mode.

   - AND / OR -



   Press the "IDS" softkey if you wish to hide static synchronized actions.

   - AND / OR -



   Press the "Blockwise" softkey if you wish to hide the non-modal synchronized actions in the automatic mode.



5. Press the "ID", "IDS" or "Blockwise" softkeys to re-display the corresponding synchronized actions.

...



7.15 Displaying the program runtime and counting workpieces

To gain an overview of the program runtime and the number of machined workpieces, open the "Times, Counter" window.



Machine manufacturer
Please observe the information provided by the machine manufacturer.

Displayed times

- Program
  Pressing the softkey the first time shows how long the program has already been running. At every further start of the program, the time required to run the entire program the first time is displayed.
  If the program or the feedrate is changed, the new program runtime is corrected after the first run.
- Program remainder
  Here you can see how long the current program still has to run. In addition, you can follow how much of the current program has been completed in percent on a progress bar.
  The first program execution differs in the calculation of the additional program executions. When a program is executed for the first time, the progress is estimated based on the program size and the actual program offset. The larger the program and the more linear that it is executed, the more precise the first estimate. This estimate is very inaccurate as a result of the system for programs with steps and/or subprograms.
  For each additional program execution, the measured overall program execution time is used as basis for the program progress display.
- Influencing the time measurement
  The time measurement is started with the start of the program and ends with the end of the program (M30) or with an agreed M function.
  When the program is running, the time measurement is interrupted with CYCLE STOP and continued with CYCLE START.
  The time measurement starts at the beginning with RESET and subsequent CYCLE START.
  The time measurement stops with CYCLE STOP or a feedrate override = 0.

Counting workpieces

You can also display program repetitions and the number of completed workpieces. For the worpiece count, enter the actual and planned workpiece numbers.

Workpiece count

Completed workpieces can be counted via the end of program command (M30) or an M command.

Procedure



1. Select the "Machine" operating area.



2. Press the <AUTO> key.



3. Press the "Times, Counter" softkey.
   The "Times, Counter" window opens.



4. Select "Yes" under "Count workpieces" if you want to count completed workpieces.

5. Enter the number of workpieces needed in the "Desired workpieces" field.
   The number of workpieces already finished is displayed in "Actual workpieces". You can correct this value when required.
   After the defined number of workpieces is reached, the current workpieces display is automatically reset to zero.

7.16 Settings for the automatic mode

Before machining a workpiece, you can test the program in order to identify programming errors at an early stage. Use the dry run feedrate for this purpose.

You have the option of additionally limiting the traversing speed so that when running-in a new program with rapid traverse, no undesirably high traversing speeds occur.

Dry run feedrate

If you selected "DRY run feedrate" under program control, then the value entered in "Dry run freedrate DRY" replaces the programmed feedrate when executing/machining.

Reduced rapid traverse

If you selected "RG0 reduced rapid traverse" under program control, then rapid traverse is reduced to the percentage value entered in "reduced rapid traverse RG0".

Displaying measurement results

Using an MMC command, measurement results can be displayed in a part program.

The following settings are possible:

- When it reaches the command, the control automatically jumps to the "Machine" operating area and the window with the measurement results is displayed
- The window with the measurement results is opened by pressing the "Measurement result" softkey

Recording machining times

To provide support when creating and optimizing a program, you have the option of displaying the machining times.

You define whether the time is determined while the workpiece is being machined (i.e. if the function is energized).

- Off
  Machining times are not determined when machining a workpiece. No machining times are determined.
- Non-modal
  The machining times are determined for each traversing block of a main program.
  Note: You also have the option of displaying the cumulative times for blocks.
  Please observe the information provided by the machine manufacturer.
- Block-by-block
  Machining times are determined for all blocks.

Note

Utilization of resources

The more machining times are displayed, the more resources are utilized.

More machining times are determined and saved with the non-modal setting than with the block-by-block setting.

Note

Please observe the information provided by the machine manufacturer.

Save machining times

You define how the machining times determined are processed.

- Yes
  A subdirectory with the name "GEN_DATA.WPD" is created in the directory of the part program. The machining times determined are saved in an ini file in the subdirectory, together with the name of the program.
- No
  The machining times that have been determined are only displayed in the program block display.

Record tools

You define whether the tool data is recorded.

- Yes
  Recording takes place during processing. The data is stored in a TTD file (Tool Time Data). The TTD file is located in the directory of the associated part program.
- No
  The tool data is not recorded.

Procedure



1. Select the "Machine" operating area.



2. Press the <AUTO> key.





3. Press the menu forward key and the "Settings" softkey.
   The "Settings for Automatic Operation" window opens.
4. In "DRY run feedrate," enter the desired dry run speed.
5. Enter the desired percentage in the "Reduced rapid traverse RG0" field.
   RG0 has no effect if you do not change the specified amount of 100 %.



6. Select the required entry in the "Display measurement result" field:
   - "Automatic"
     The measurement result window opens automatically.
   - "manual"
     The measurement result window is to be opened by pressing the "Measurement result" softkey.



7. Select the desired entry in the fields "Record machining time", "Save machining times" and "Record tools".

Further information

Additional information on the display of measurement result display is provided in the Measuring Cycles Programming Manual.

Note

You have the option of changing the feed velocity during operation.

See also

Current block display (Page 41)

7.17 Working with DXF files

7.17.1 Overview

The "DXF-Reader" function allows you to open files created in SINUMERIK Operate directly in a CAD system and accept and store contours directly in G-code.

The DXF file can be displayed in the Program Manager.



Software option
You require the "DXF-Reader" software option in order to use this function.



Machine manufacturer
Please observe the information provided by the machine manufacturer.

The DXF reader reads the following elements:

- "POINT"
- "LINE"
- "CIRCLE"
- "ARC"
- "TRACE"
- "SOLID"
- "TEXT"
- "SHAPE"
- "BLOCK"
- "ENDBLK"
- "INSERT"
- "ATTDEF"
- "ATTRIB"
- "POLYLINE"
- "VERTEX"
- "SEQEND"
- "3DLINE"
- "3DFACE"
- "DIMENSION"
- "LWPOLYLINE"
- "ELLIPSE"

- "LTYPE"
- "LAYER"
- "STYLE"
- "VIEW"
- "UCS"
- "VPORT"
- "APPID"
- "DIMSTYLE"
- "HEADER ($INSUNITS, $MEASUREMENT)"
- "TABLES"
- "BLOCKS"
- "ENTITIES"

7.17.2 Displaying CAD drawings

7.17.2.1 Open a DXF file

Procedure



1. Select the "Program Manager" operating area.

2. Choose the desired storage location and position the cursor on the DFX file that you want to display.



3. Press the "Open" softkey.

   The selected CAD drawing will be displayed with all its layers, i.e. with all graphic levels.



4. Press the "Close" softkey to close the CAD drawing and to return to the Program Manager.

7.17.2.2 Cleaning a DXF file

All contained layers are shown when a DXF file is opened.

Layers that do not contain any contour- or position-relevant data can be shown or hidden.

Requirement

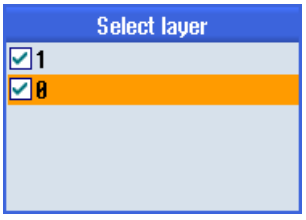
The DXF file is open in the Program Manager or in the editor.

Procedure





1. Press the "Clean" and "Layer selection" softkeys if you want to hide specific layers.
   The "Layer Selection" window opens.





2. Deactivate the required layers and press the "OK" softkey.

- OR -



Press the "Clean automat." softkey to hide all non-relevant layers.



3. Press the "Clean automat." softkey to redisplay the layers.

7.17.2.3 Enlarging or reducing the CAD drawing

Requirement

The DXF file is opened in the Program Manager.

Procedure





1. Press the "Details" and "Zoom +" softkeys if you wish to enlarge the size of the segment.

- OR -





2. Press the "Details" and "Zoom -" softkeys if you wish to reduce the size of the segment.

- OR -



3. Press the "Details" and "Auto zoom" softkeys if you wish to automatically adapt the segment to the size of the window.



- OR -



4. Press the "Details" and "Zoom elem. selection" softkeys if you want to automatically zoom elements that are in a selection set.

7.17.2.4 Changing the section

If you want to move or change the size of a section of the drawing, for example, to view details or redisplay the complete drawing later, use the magnifying glass.
You can use the magnifying glass to determine the section and then change its size.

Requirement

The DXF file is opened in the Program Manager or in the editor.

Procedure



1. Press the "Details" and "Magnifying glass" softkeys.
   A magnifying glass in the shape of a rectangular frame appears.





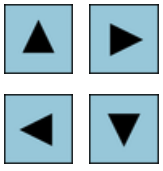
2. Press the <+> key to enlarge the frame.

- OR -



Press the <-> key to reduce the frame.

- OR -



Press a cursor key to move the frame up, down, left or right.



3. Press the "OK" softkey to accept the section.

7.17.2.5 Rotating the view

You can change the orientation of the drawing.

Requirement

The DXF file is open in the Program Manager or in the editor.

Procedure





1. Press the "Details" and "Rotate figure" softkeys.



2. Press the "Arrow right", "Arrow left", "Arrow up", "Arrow down", "Arrow clockwise" or "Arrow counter-clockwise" softkey to change the position of the drawing.

...



7.17.2.6 Displaying/editing information for the geometric data

Precondition

The DXF file is opened in the Program Manager or in the editor.

Procedure





1. Press the "Details" and "Geometry info" softkeys.
   The cursor takes the form of a question mark.



2. Position the cursor on the element for which you want to display its geometric data and press the "Element Info" softkey.
   If, for example, you have selected a straight line, the following window opens "Straight line on layer: ...". You are shown the coordinates corresponding to the actual zero point in the selected layer: Start point for X and Y, end point for X and Y as well as the length.

4. If you are currently in the editor, press the "Element edit" softkey.
   The coordinate values can be edited.





3. Press the "Back" softkey to close the display window.

Note

Editing a geometric element

You can use this function to make smaller changes to the geometry, e.g. for missing intersections.

You should make larger changes in the input screen of the editor.

You cannot undo any changes that you make with "Element Edit".

7.17.3 Importing and editing a DXF file

7.17.3.1 General procedure

- Creating/opening a G code program
- Call the "Contour" cycle and create a "New contour"
- Import the DXF file
- Select the contour in the DXF file or CAD drawing and click "OK" to accept the cycle
- Add program record with "Accept" to the G code program

7.17.3.2 Setting the tolerance

To allow even inaccurately created drawings to be used, i.e. to compensate for gaps in the geometry, you can enter a snap radius in millimeters. This relates elements.

Note

Large snap radius

The larger that the snap radius is set, the larger the number of available following elements.

Procedure







1. The DXF file is opened in the editor.
2. Press the "Details" and "Snap radius" softkeys.
   The "Input" window appears.
3. Enter the desired value and press the "OK" softkey.

7.17.3.3 Assigning the machining plane

You can select the machining plane in which the contour created with the DXF reader should be located.

Procedure





1. The DXF file is opened in the editor.
2. Press the "Select plane" softkey.
   The "Select Plane" window opens.
3. Select the desired plane and press the "OK" softkey.

7.17.3.4 Selecting the machining range / deleting the range and element

You can select ranges in the DXF file and therefore reduce the elements. After accepting the 2nd position, only the contents of the selected rectangle are displayed. Contours are cut to the rectangle.

Requirement

The DXF file is open in the editor.

Procedure

Select the machining range from the DXF file





1. Press the "Reduce" and "Select range" softkeys if you want to select specific ranges of the DXF file.
   An orange rectangle is displayed.





2. Press the "Range +" softkey to enlarge the section or press the "Range -" softkey to reduce the section.





3. Press the "Arrow right", "Arrow left", "Arrow up" or "Arrow down" softkey to move the selection tool.





4. Press the "OK" softkey.
   The machining section is displayed.
   Use the "Cancel" softkey to return to the previous window.



5. Press the "Deselect range" softkey to undo the selection of the machining range.
   The DXF fie is reset to the original display.

Delete selected ranges and elements of the DXF file



6. Press the "Reduce" softkey.

Delete range



7. Press the "Range delete" softkey.
   A blue rectangle is displayed.



8. Press the "Range +" softkey to enlarge the section or press the "Range -" softkey to reduce the section.





9. Press the "Arrow right", "Arrow left", "Arrow up" or "Arrow down" softkey to move the selection tool.



- OR -

Delete element



10. Press the "Element delete" softkey, and using the selection tool, select the element that you wish to delete.
11. Press "OK".

7.17.3.5 Saving the DXF file

You can save DXF files that you have reduced and edited.

Requirement

The DXF file is open in the editor.

Procedure



1. Reduce file according to your requirements and/or select the working areas.



- OR -





2. Press the "Back" and ">>" softkeys.





3. Press the "Save DXF" softkey.



4. Enter the required name in the "Save DXF Data" window and press "OK".
   The "Save As" window opens.
5. Select the required storage location.



6. If required, press the "New directory" softkey, enter the required name in the "New Directory" window and press the "OK" softkey to create a directory.



7. Press the "OK" softkey.

7.17.3.6 Specifying a reference point

Because the zero point of the DXF file normally differs from the zero point of the CAD drawing, specify a reference point.

Procedure

1. The DXF file is opened in the editor.



2. Press the ">>" and "Specify reference point" softkeys.





3. Press the "Element start" softkey to place the zero point at the start of the selected element.

   - OR -



   Press the "Element center" softkey to place the zero point at the center of the selected element.

   - OR -



   Press the "Element end" softkey to place the zero point at the end of the selected element.

   - OR -



   Press the "Arc center" softkey to place the zero point at the center of an arc.

   - OR -



   Press the "Cursor" softkey to define the zero point at any cursor position.

- OR -



Press the "Free input" softkey to open the "Reference Point Input" window and enter the values for the positions (X, Y) there.

7.17.3.7 Accepting contours

1. The part program to be processed has been created and you are in the editor.



2. Press the "Contour" softkey.



3. Press the "New contour" softkey.

Select contour

The start and end point are specified for the contour line.

The start point and the direction are selected on a selected element. Beginning at the start point, the automatic contour line takes all subsequent elements of a contour. The contour line ends as soon as there are no subsequent elements – or intersections with other elements of the contour occur.

Note

If a contour includes more elements than can be processed, you will be offered the option of transferring the contour to the program as pure G code.

This contour then can no longer be edited in the editor.



With the "Undo" softkey, you can undo your contour selection back to a specific point.

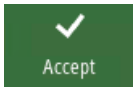
Procedure

Opening a DXF file

1. Enter the desired name in the "New Contour" window.



2. Press the "From DXF file" and "Accept" softkeys.
   The "Open DXF File" window opens.



3. Select a storage location and place the cursor on the relevant DXF file.



You can, for example, use the search function to search directly for a DXF file in comprehensive folders and directories.



4. Press the "OK" softkey.
   The CAD drawing opens and can be edited for contour selection.
   The cursor takes the form of a cross.

Specifying a reference point

5. If required, specify a zero point.

Contour line





6. Press the ">>" and "Automatic" softkeys if you want to accept the largest possible number of contour elements.
   This makes it fast to accept contours that consist of many individual elements.

   - OR -



   Press "Only to 1st cut" if you do not want to accept the complete contour elements at once.

   The contour will be followed to the first cut of the contour element.

Defining the start point



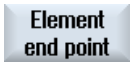
7. Press the "Select element" softkey to select the desired element.



8. Press the "Accept element" softkey.



9. Press the "Element start point" softkey to place the contour start at the start point of the element.
   - OR -



   Press the "Element end point" softkey to place the contour start at the end point of the element.
   - OR -



   Press the "Element center" softkey to place the contour start at the center of the element.
   - OR -



   Press the "Cursor" softkey to define the start of the element with the cursor at any position.



9. Press the "OK" softkey to confirm your selection.



10. Press the "Accept element" softkey to accept the offered elements.
    The softkey can be operated while elements are still available to be accepted.

Specifying the end point





11. Press the ">>" and "Specify end point" softkeys if you do not want to accept the end point of the selected element.



12. Press the "Current position" softkey if you want to set the currently selected position as end point.

    - OR -



Press the "Element center" softkey to place the contour end at the center of the element.

- OR -



Press the "Element end" softkey to place the contour end at the end of the element.

- OR -



Press the "Cursor" softkey to define the start of the element with the cursor at any position.

Transferring the contour to the cycle and to the program



Press the "OK" softkey.

The selected contour is transferred to the contour input screen of the editor.



Press the "Accept contour" softkey.

The program block is transferred to the program.

Operation with mouse and keyboard

In addition to operation using softkeys, you can also operate the functions with the keyboard and with the mouse.

7.18 Mold making view

For large mold making programs such as those provided by CAD/CAM systems, you have the option to display the machining paths by using a fast view. This provides you with a fast overview of the program, and you have the possibility of correcting it.



Machine manufacturer
The mold making view has possibly been hidden.
Please observe the information provided by the machine manufacturer.

Checking the program

You can check the following:

- Does the programmed workpiece have the correct shape?
- Are there large traversing errors?
- Which program block hasn't been correctly programmed?
- How is the approach and retraction realized?

NC blocks that can be interpreted

The following NC blocks are supported for the mold making view:

- Types
  - Lines
    G0, G1 with X Y Z
  - Circles
    G2, G3 with center point I, J, K or radius CR, depending on the working plane G17, G18, G19, CIP with circular point I1, J1, K1 or radius CR
  - Absolute data AC and incremental data IC are possible
  - For G2, G3 and different radii at the start and end, an Archimedes spiral is used
- Orientation
  - Rotary axis programming with ORIAXES or ORIVECT using ABC for G0, G1, G2, G3, CIP, POLY
  - Orientation vector programming with ORIVECT using A3, B3, C3 for G0, G1, G2, G3, CIP
  - Rotary axes can be specified using DC
- G codes
  - Working planes (for circle definition G2, G3): G17 G18 G19
  - Incremental or absolute data: G90 G91

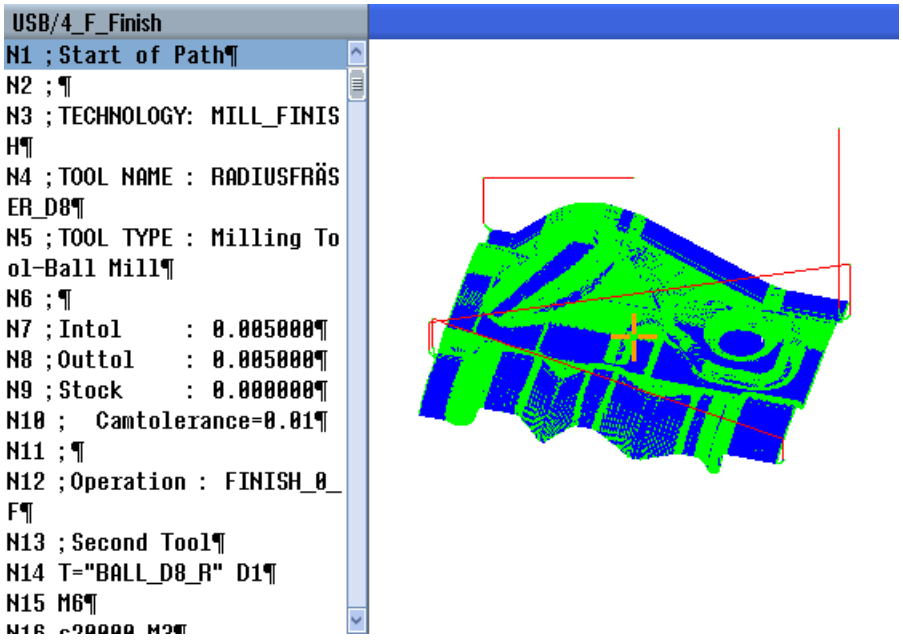
The following NC blocks are **not** supported for the mold making view:

- Helix programming
- Rational polynomials
- Other G codes or language commands

All NC blocks that cannot be interpreted are simply overread.

Simultaneous view of the program and mold making view

You have the option of displaying the mold making view next to the program blocks in the editor.



You can navigate back and forth between the NC blocks listed on the left and the associated points in the mold making view.

- On the left in the editor, if you place the cursor on an NC block with position data, then this NC block is marked in the graphic view.
- If you select a point on the right in the mold making view using the mouse, then conversely you mark the corresponding NC block on the left-hand side of the editor. This is how you jump directly to a position in the program in order to edit a program block for example.

Switch between the program window and the mold making view



Press the <NEXT WINDOW> key if you wish to toggle between the program window and the mold making view.

Changing and adapting the mold making view

Like simulation and simultaneous recording, you have the option of changing and adapting the mold making view in order to achieve the optimum view.

- Increasing or reducing the size of the graphic
- Moving the graphic
- Rotating the graphic
- Changing the section

7.18.1 Starting the mold making view

Procedure



1. Select the "Program manager" operating area.

2. Select the program that you would like to display in the mold making view.
3. Press the "Open" softkey.
   The program is opened in the editor.



4. Press the ">>" and "Mold making view" softkeys.
   The editor splits up into two areas.







The G code blocks are displayed in the left half of the editor.

The workpiece is displayed in the mold making view on the right-hand side of the editor. All of the points and paths programmed in the part program are represented.

7.18.2 Adapting the mold making view

You can adapt the graphic in various ways to better assess the workpiece in the mold making view.

Preconditions

- The required program is opened in the mold making view.
- The "Graphic" softkey is active.

Procedure



1. Press the softkey "Hide G1/G2/G3" if you want to conceal the machining paths.

- OR -



2. Press the softkey "Hide G0" if you want to deactivate the approach and retraction paths.

- OR -



Press softkey "Hide points" to conceal all the points in the graphic.

Note:

You have the option of simultaneously hiding G1/G2/G3 and G0 lines.

In this case softkey "Hide points" is deactivated.

- OR -



Press the softkeys ">>" and "Vectors" to display all orientation vectors.

Note:

This softkey can only be operated if vectors are programmed.



- OR -



Press the softkeys ">>" and "Surface" to calculate the surface area of the workpiece.



- OR -



Press the softkeys ">>" and "Curvature".

The "Curvature" input window opens.



Enter the desired minimum and maximum value and press "OK" to confirm the entry and to highlight the curvature changes in color.



7.18.3 Specifically jump to the program block

If you notice anything peculiar in the graphic or identify an error, then from this location, you can directly jump to the program block involved where you can edit the program.

Requirements

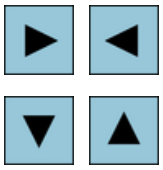
- The required program is opened in the mold making view.
- The "Graphic" softkey is active.

Procedure



1. Press the ">>" and "Select point" softkeys.
   Cross-hairs for selecting a point are shown in the diagram.





2. Using the cursor keys, move the cross-hairs to the desired position in the graphic.



3. Press the "Select NC block" softkey.
   The cursor jumps to the associated program block in the editor.

7.18.4 Searching for program blocks

Using the "Search" function, you can go specifically to program blocks where you can edit programs. You can find and replace text in one step.

Precondition

- The required program is opened in the mold making view.
- The "NC blocks" softkey is active.

Procedure



1. Press the "Search" softkey.
   A new vertical softkey bar appears.

7.18.5 Changing the view

7.18.5.1 Enlarging or reducing the graphical representation

Precondition

- The mold making view has been started.
- The "Graphic" softkey is active.

Procedure



1. Press the <+> and <-> keys if you wish to enlarge or reduce the graphic display.
   The graphic display enlarged or reduced from the center.

...



- OR -





Press the "Details" and "Zoom +" softkeys if you wish to increase the size of the segment.

- OR -





Press the "Details" and "Zoom -" softkeys if you wish to decrease the size of the segment.

- OR -





Press the "Details" and "Auto zoom" softkeys if you wish to automatically adapt the segment to the size of the window.
The automatic scaling function "Fit to size" takes account of the largest expansion of the workpiece in the individual axes.

Note

Selected section

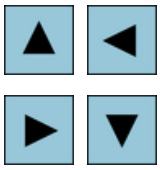
The selected sections and size changes are kept as long as the program is selected.

7.18.5.2 Moving and rotating the graphic

Precondition

- The mold making view has been started.
- The "Graphic" softkey is active.

Procedure

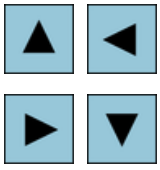


1. Press one of the cursor keys to move the mold making view up, down, left or right.

- OR -



With the <SHIFT> key pressed, rotate the mold building view in the required direction using the cursor keys.



Note

Working with a mouse

Using the mouse, you have the option of rotating and shifting the mold making view.

- To do this, move the graphic with the left-hand mouse key pressed in order to reposition the mold making view.
- To do this, move the graphic with the right-hand mouse key pressed in order to rotate the mold making view.

7.18.5.3 Modifying the viewport

If you want to look at the details, you can shift and change the size of the mold building view section using a magnifying glass.

Using the magnifying glass, you can define your own segment and then increase or decrease its size.

Precondition

- The mold making view has been started.
- The "Graphic" softkey is active.

Procedure



1. Press the "Details" softkey.



2. Press the "Zoom" softkey.
   A magnifying glass in the shape of a rectangular frame appears.



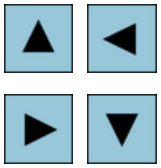
3. Press the "Magnify +" or <+> softkey to enlarge the frame.

- OR -

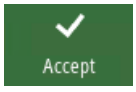


Press the "Magnify -" or <-> softkey to reduce the frame.

- OR -



Press one of the cursor keys to move the frame up, down, left or right.



4. Press the "Accept" softkey to accept the section.

Simulating machining

8

8.1 Overview

During simulation, the current program is calculated in its entirety and the result is displayed in graphical form. The result of programming is verified without traversing the machine axes. Incorrectly programmed machining steps are detected at an early stage and incorrect machining on the workpiece is prevented.

Graphical display

The simulation represented on the screen uses the correct workpiece and tool proportions.

For simulation at milling machines, the workpiece is located, fixed in space. Only the tool moves, independent of the machine type.

Defining a blank

The blank dimensions that are entered in the program editor are used for the workpiece.

The blank is clamped with reference to the coordinate system, which is valid at the time the blank was defined. This means that before defining the blank in G code programs, the required output conditions must be established, e.g. by selecting a suitable work offset.

Programming a blank (example)

```
G54 G17 G90
CYCLE800(0,"TABLE", 100000,57,0,0,0,0,0,0,0,0,0,-1,100,1)
WORKPIECE(,,,"Box",112,0,-50,-80,00,155,100)
T="NC-SPOTDRILL_D16
```

Note

Blank offset for a changed work offset

The blank is always created in the work offset which is presently active.

If you select another work offset, then the coordinate system is converted, however, the display of the blank is not changed.

Note

Blank clamping

If your machine has various blank clamping options, enter the required clamping in the program header or in the blank screen.

Note also the machine manufacturer's instructions in this regard.

Display of the traversing paths

The traversing paths are displayed in color. Rapid traverse is red and the feedrate is green.

Depth display

The depth infeed is color-coded. The depth display indicates the depth level at which machining is currently taking place. "The deeper, the darker" applies for the depth display.

MCS references

The simulation is implemented as workpiece simulation. This means that it is not assumed that the work offset has already been precisely scratched or is known.

In spite of this, there are unavoidable MCS references in the programming, such as the tool change point in the machine coordinate system, the retraction position for swiveling, and the table components of swivel kinematics. Depending on the current work offset - in the worst case - these machine references can mean that collisions are shown in the simulation that would not occur for a realistic work offset - or vice versa, collisions are not shown, which could occur for a realistic work offset.

Programmable frames

All frames and work offsets are taken into account in the simulation.

Note

Manually swiveled axes

Note that swivel movement in simulation and during simultaneous recording is also displayed when the axes are swiveled manually at the start.

Simulation display

You can choose one of the following types of display:

- Material removal simulation
  During simulation or simultaneous recording you can follow stock removal from the defined blank.
- Path display
  You have the option of including the display of the path. The programmed tool path is displayed.

  Note

  Tool display in the simulation and for simultaneous recording

  In order that workpiece simulation is also possible for tools that have either not been measured or have been incompletely entered, certain assumptions are made regarding the tool geometry.

  For instance, the length of a miller or drill is set to a value proportional to the tool radius so that cutting can be simulated.

Note

Thread turns not displayed

For thread and drill thread milling, the thread turns are not displayed in the simulation and for simultaneous recording.

Display variants

You can choose between three variants of graphical display:

- Simulation before machining of the workpiece
  Before machining the workpiece on the machine, you can perform a quick run-through in order to graphically display how the program will be executed.
- Simultaneous recording before machining of the workpiece
  Before machining the workpiece on the machine, you can graphically display how the program will be executed during the program test and dry run feedrate. The machine axes do not move if you have selected "no axis motions".
- Simultaneous recording during machining of the workpiece
  You can follow machining of the workpiece on the screen while the program is being executed on the machine.

Views

The following views are available for all three variants:

- Top view
- 3D view
- Side views

Status display

The current axis coordinates, the override, the current tool with cutting edge, the current program block, the feedrate and the machining time are displayed.

In all views, a clock is displayed during graphical processing. The machining time is displayed in hours, minutes and seconds. It is approximately equal to the time that the program requires for processing including the tool change.



Software options

You require the "3D simulation of the finished part" option for the 3D view.

You require the "Simultaneous recording (real-time simulation)" option for the "Simultaneous recording" function.

Determining the program runtime

The program runtime is determined when executing the simulation. The program runtime is temporarily displayed in the editor at the end of the program.

Model quality

The processing speed and total duration of a program in simulation partly depends on the quality and complexity of the model used. By reducing the optical quality, a shorter simulation time can be achieved under certain circumstances. You can set the model quality for simulation according to your requirements in the interface.

The following table shows which model quality options are available depending on the SINUMERIK ONE version:

SINUMERIK ONE version	Available options
SINUMERIK Operate on NCU 1740	• Low • High
SINUMERIK Operate on NCU 1750	• Low • High
SINUMERIK Operate on NCU 1760	• Low • High • Very high
SINUMERIK Operate on IPC	• Low • High • Very high

Properties of simultaneous recording and simulation

Traversing paths

For the simulation, the displayed traversing paths are saved in a ring buffer. If this buffer is full, then the oldest traversing path is deleted with each new traversing path.

Optimum display

If simultaneous machining is stopped or has been completed, the display is again converted into a high-resolution image. In some cases, this is not possible. In this case, the following message is output: "High-resolution image cannot be generated".

Working zone limitation

No working area limits and software limit switches are effective in the tool simulation.

Start position for simulation and simultaneous recording

During simulation, the start position is converted via the work offset to the workpiece coordinate system.

The simultaneous recording starts at the position at which the machine is currently located.

Restrictions

- TRAORI: 5-axis motion is linearly interpolated. More complex motion cannot be displayed.
- Referencing: G74 from a program run does not function.
- Alarm 15110 "REORG block not possible" is not displayed.
- Compile cycles are only partly supported.

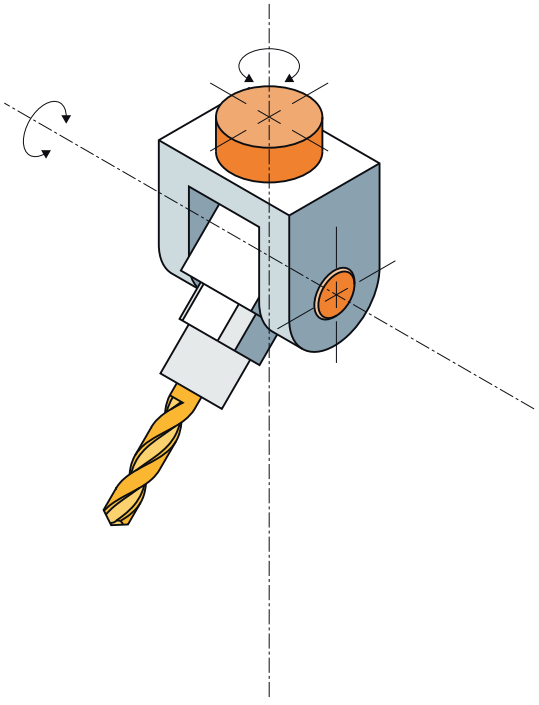
- No PLC support.
- Axis containers are not supported.

Supplementary conditions

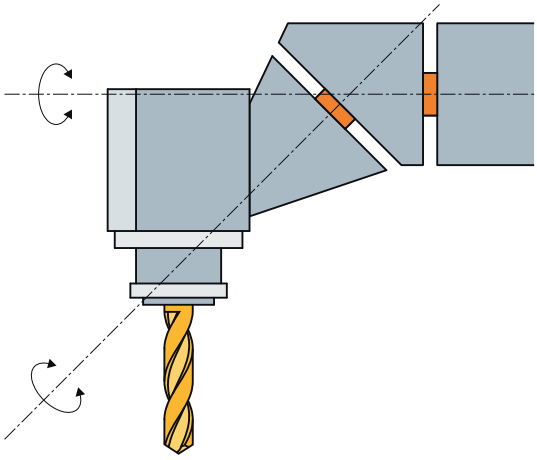
- All of the existing data records (tool carrier / TRAORI, TRACYL) are evaluated and must be correctly commissioned for correct simulation.
- The machine kinematics for TRAFOOF are not taken into consideration.
- Transformations with swiveled linear axis (TRAORI 64 - 69) as well as OEM transformations (TRAORI 4096 - 4098) are not supported.
- Changes to the tool carrier or transformation data only become effective after Power On.
- Transformation change and swivel data record change are supported. However, a real kinematic change is not supported, where a swivel head is physically changed.
- The simulation of mold making programs with extremely short block change times can take longer than machining, as the computation time distribution for this application is dimensioned in favor of the machining and to the detriment of simulation.

Examples

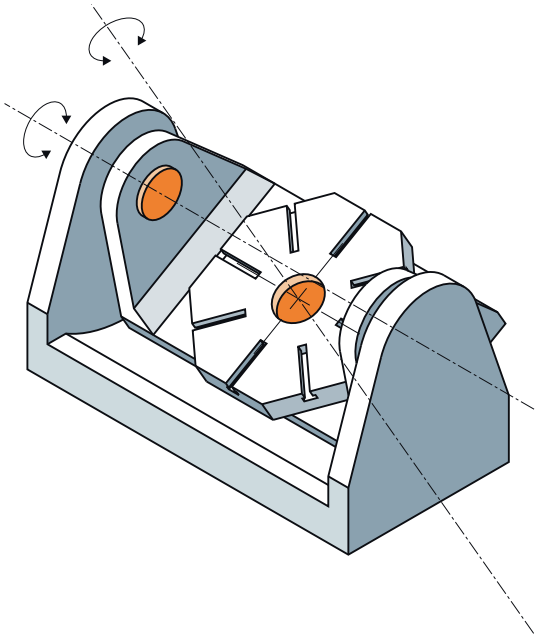
Several examples for machine types that are supported:



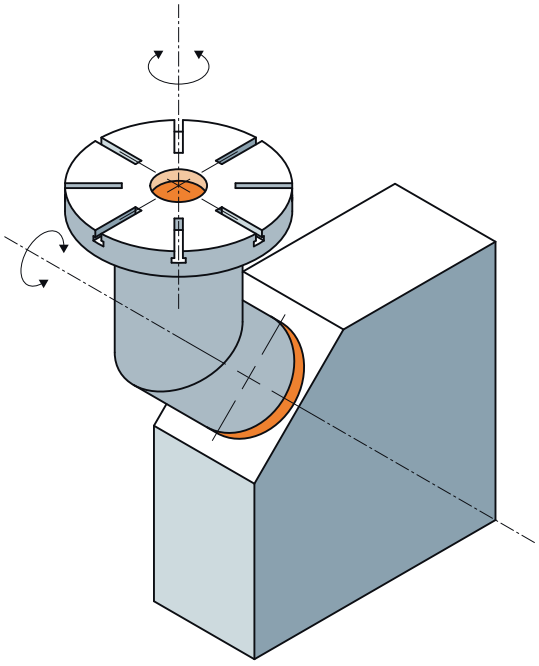
Swivel head 90°/90°



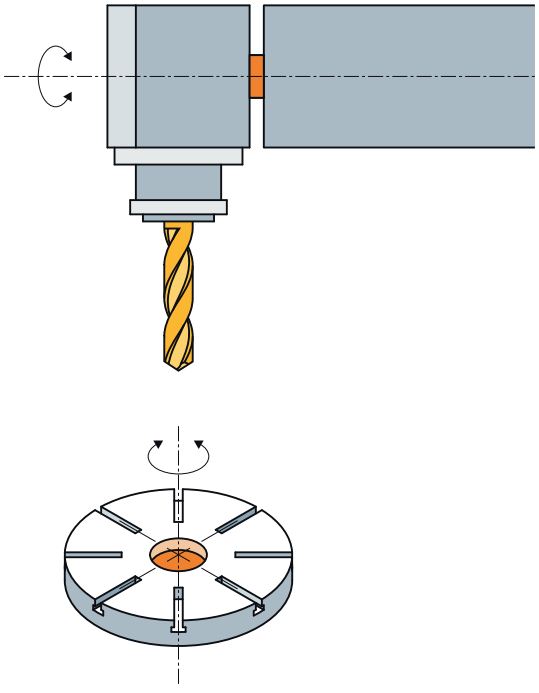
Swivel head 90°/45°



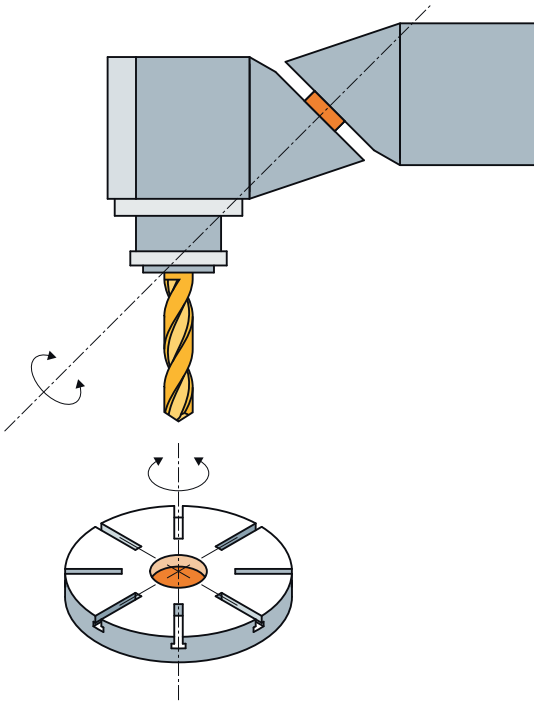
Swivel table 90°/90°



Swivel table 90°/45°



Swivel combination 90°/90°



Swivel combination 45°/90°

8.2 Simulation prior to machining of the workpiece

Before machining the workpiece on the machine, you have the option of performing a quick run-through in order to graphically display how the program will be executed. This provides a simple way of checking the result of the programming.

Feedrate override

The rotary switch (override) on the control panel only influences the functions of the "Machine" operating area.

Press the "Program control" softkey to change the simulation feedrate. You can select the simulation feedrate in the range of 0 - 120%.

See also

Changing the feedrate (Page 228)

Simulating the program block by block (Page 229)

Procedure



1. Select the "Program Manager" operating area.

2. Select the storage location and position the cursor on the program to be simulated.



3. Press the <INPUT> or <Cursor right> key.



- OR -

Double-click the program.

The selected program is opened in the "Program" operating area.



4. Press the "Simulation" softkey.

The program execution is displayed graphically on the screen. The machine axes do not move.





5. Press the "Stop" softkey if you wish to stop the simulation.

- OR -



Press the "Reset" softkey to cancel the simulation.



6. Press the "Start" softkey to restart or continue the simulation.

Note

Operating area switchover

The simulation is exited if you switch into another operating area. If you restart the simulation, then this starts again at the beginning of the program.

8.3 Simultaneous recording prior to machining of the workpiece

Before machining the workpiece on the machine, you can graphically display the execution of the program on the screen to monitor the result of programming.



Software option

You require the option "Simultaneous recording (real-time simulation)" for simultaneous recording.

You can replace the programmed feedrate with a dry run feedrate to influence the speed of execution and select the program test to disable axis motion.

If you would like to view the current program blocks again instead of the graphical display, you can switch to the program view.

Procedure

1. Load a program in the "AUTO" mode.



2. Press the "Prog. ctrl." softkey and activate the checkboxes "PRT no axis movement" and "DRY run feedrate".

   The program is executed without axis movement. The programmed feedrate is replaced by a dry run feedrate.



3. Press the "Sim. rec." softkey.



4. Press the <CYCLE START> key.

   The program execution is displayed graphically on the screen.



5. Press the "Sim. rec." softkey again to stop the recording.

8.4 Simultaneous recording during machining of the workpiece

If the view of the work space is blocked by coolant, for example, while the workpiece is being machined, you can also track the program execution on the screen.



Software option

You require the option "Simultaneous recording (real-time simulation)" for simultaneous recording.

Procedure

1. Load a program in the "AUTO" mode.



2. Press the "Sim. rec." softkey.



3. Press the <CYCLE START> key.

   The machining of the workpiece is started and graphically displayed on the screen.



4. Press the "Sim. rec." softkey again to stop the recording.

Note

- If you switch on simultaneous recording after the blank information has already been processed in the program, only traversing paths and tool are displayed.
- If you switch off simultaneous recording during machining and then switch on the function again at a later time, the traversing paths generated in the meantime will not be displayed.

8.5 Setting the model quality

Precondition

Simulation is started.

Procedure





1. Press the ">>" and "Model quality" softkeys.
   The "Model quality" windows opens and displays the available values.
2. Select the desired model quality.
3. Confirm your selection with "OK".



For complex workpieces, switching the model quality can take some time. In this case, a progress bar will be displayed.

8.6 Different views of the workpiece

8.6.1 Overview

In the graphical display, you can choose between different views so that you constantly have the best view of the current workpiece machining, or in order to display details or the overall view of the finished workpiece.

The following views are available:

- Top view
- 3D view (with option)
- Side views
- Machine space (with "collision avoidance" option)

8.6.2 Plan view

Display as a top view



1. Simultaneous recording or the simulation is started.
2. Press the "Top view" softkey.
   The workpiece is shown from above in the top view.

Changing the display

You can increase or decrease the size of the simulation graphic and move it, as well as change the segment.

See also

Enlarging or reducing the graphical representation (Page 230)

Panning a graphical representation (Page 231)

Modifying the viewport (Page 232)

8.6.3 3D view

Displaying the 3D view

1. Simultaneous recording or the simulation is started.
2. Press the "Other views" and "3D view" softkeys.







Software option

You require the option "3D simulation (finished part)" for the simulation.

Changing the display

You can increase or decrease the size of the simulation graphic, move it, turn it, or change the segment.

Displaying and moving cutting planes

You can display and move cutting planes X, Y, and Z.

See also

Enlarging or reducing the graphical representation (Page 230)

Panning a graphical representation (Page 231)

Rotating the graphical representation (Page 231)

Modifying the viewport (Page 232)

Defining cutting planes (Page 232)

8.6.4 Side view

Displaying further side views

1. Simultaneous recording or the simulation is started.
2. Press the "Other views" softkey.



3. Press the "From front" softkey if you wish to view the workpiece from the front.



- OR -



Press the "From rear" softkey if you wish to view the workpiece from the rear.

- OR -



Press the "From left" softkey if you wish to view the workpiece from the left.

- OR -



Press the "From right" softkey if you wish to view the workpiece from the right.

Changing the display

You can increase or decrease the size of the simulation graphic and move it, as well as change the segment.

8.7 Editing the simulation display

8.7.1 Blank display

You have the option of replacing the blank defined in the program or to define a blank for programs in which a blank definition cannot be inserted.

Note

The unmachined part can only be entered if simulation or simultaneous recording is in the reset state.

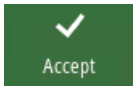
Procedure



1. The simulation or the simultaneous recording is started.
2. Press the ">>" and "Blank" softkeys.
   The "Blank Input" windows opens and displays the pre-assigned values.



3. Enter the desired values for the dimensions.



4. Press the "Accept" softkey to confirm your entries. The newly defined workpiece is displayed.

8.7.2 Showing and hiding the tool path

The path display follows the programmed tool path of the selected program. The path is continuously updated as a function of the tool movement. The tool paths can be shown or hidden as required.

Procedure



1. The simulation or the simultaneous recording is started.
2. Press the ">>" softkey.
   The tool paths are displayed in the active view.



3. Press the softkey to hide the tool paths.
   The tool paths are still generated in the background and can be shown again by pressing the softkey again.



4. Press the "'Delete tool path" softkey.
   All of the tool paths recorded up until now are deleted.

8.8 Program control during simulation

8.8.1 Changing the feedrate

You have the capability of changing the feedrate at any time during the simulation.

You track the changes in the status bar.

Note

Use the rotary switch (override) at the control panel if you are working with the "Simultaneous recording" function.

Procedure

1. Simulation is started.
2. 

Press the "Program control" softkey.
3. 



Press the "Override +" or "Override -" softkey to increase or decrease the feedrate by 5 %, respectively.

   - OR -

Press the "100% override" softkey to set the feedrate to 100%.

   - OR -

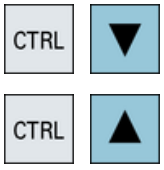
Press the "Override Max" softkey to set the feedrate to the maximum value.

   The simulation runs at the highest possible speed.

   The originally set override value is re-activated by pressing the "Override Max" softkey again.

   
Press the "<<" softkey to return to the main screen and perform the simulation with changed feedrate.

Toggling between "Override +" and "Override -"



Simultaneously press the <Ctrl> and <cursor down> or <cursor up> keys to toggle between the "Override +" and "Override -" softkeys.

Selecting the maximum feedrate

CTRL M Press the <Ctrl> and <M> keys simultaneously to select the maximum feedrate of 120%.

8.8.2 Simulating the program block by block

You have the capability of controlling the program execution during the simulation, i.e. to execute a program, e.g. program block by program block.

Procedure

1. Simulation is started.
2. Press the "Program control" and "Single block" softkeys.





3. Press the "Back" and "Start SBL" softkeys.
   The pending program block is simulated and then stops.







4. Press "Start SBL" as many times as you want to simulate a single program block.
5. Press the "Program control" and the "Single block" softkeys to exist the single block mode.





Switching a single block on and off

CTRL S Press the <CTRL> and <S> keys simultaneously to enable and disable the single block mode.

8.9 Modifying and adapting the simulation graphics

8.9.1 Enlarging or reducing the graphical representation

Precondition

The simulation or the simultaneous recording is started.

Procedure



1. Press the <+> and <-> keys if you wish to enlarge or reduce the graphic display.
   The graphic display enlarged or reduced from the center.

...



- OR -





Press the "Details" and "Zoom +" softkeys if you wish to increase the size of the segment.

- OR -





Press the "Details" and "Zoom -" softkeys if you wish to decrease the size of the segment.

- OR -





Press the "Details" and "Auto zoom" softkeys if you wish to automatically adapt the segment to the size of the window.
The automatic scaling function "Fit to size" takes account of the largest expansion of the workpiece in the individual axes.

Note

Selected section

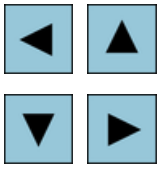
The selected sections and size changes are kept as long as the program is selected.

8.9.2 Panning a graphical representation

Precondition

The simulation or the simultaneous recording is started.

Procedure



1. Press a cursor key if you wish to move the graphic up, down, left, or right.

8.9.3 Rotating the graphical representation

In the 3D view you can rotate the position of the workpiece to view it from all sides.

Requirement

The simulation or simultaneous recording has been started and the 3D view is selected.

Procedure



1. Press the "Details" softkey.



2. Press the "Rotate view" softkey.



3. Press the "Arrow right", "Arrow left", "Arrow up", "Arrow down", "Arrow clockwise" and "Arrow counterclockwise" softkeys to change the position of the workpiece.

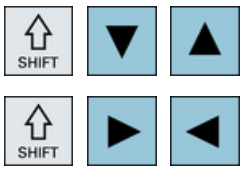
...



...



- OR -



Keep the <Shift> key pressed and then turn the workpiece in the desired direction using the appropriate cursor keys.

8.9.4 Modifying the viewport

If you would like to move, enlarge or decrease the size of the segment of the graphical display, e.g. to view details or display the complete workpiece, use the magnifying glass.

Using the magnifying glass, you can define your own section and then enlarge or reduce its size.

Precondition

The simulation or the simultaneous recording is started.

Procedure



1. Press the "Details" softkey.



2. Press the "Magnifying glass" softkey.
   A magnifying glass in the shape of a rectangular frame appears.



3. Press the "Magnify +" or <+> softkey to enlarge the frame.



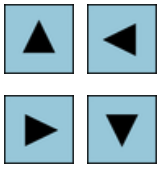
- OR -



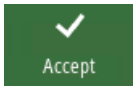
Press the "Magnify -" or <-> softkey to reduce the frame.



- OR -



Press one of the cursor keys to move the frame up, down, left or right.



4. Press the "Accept" softkey to accept the selected section.

8.9.5 Defining cutting planes

In the 3D view, you have the option of "cutting" the workpiece and therefore displaying certain views in order to show hidden contours.

Precondition

The simulation or the simultaneous recording is started.

Procedure



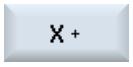
1. Press the "Details" softkey.



2. Press the "Cut" softkey.



The workpiece is displayed in the cut state.



3. Press the corresponding softkey to shift the cutting plane in the required direction.

...



8.10 Displaying simulation alarms

Alarms might occur during simulation. If an alarm occurs during a simulation run, a window opens in the operating window to display it.

The alarm overview contains the following information:

- Date and time
- Deletion criterion
  Specifies with which softkey the alarm is acknowledged
- Alarm number
- Alarm text

Precondition

Simulation is running and an alarm is active.

Procedure





1. Press the "Program control" and "Alarm" softkeys.
   The "Simulation Alarms" window is opened and a list of all pending alarms is displayed.



Press the "Acknowledge alarm" softkey to reset the simulation alarms indicated by the Reset or Cancel symbol.

The simulation can be continued.

- OR -



Press the "Simulation Power On" softkey to reset a simulation alarm indicated by the Power On symbol.

Creating a G code program

9

9.1 Graphical programming

Functions

The following functionality is available:

- Technology-oriented program step selection (cycles) using softkeys
- Input windows for parameter assignment with animated help screens
- Context-sensitive online help for every input window
- Support with contour input (geometry processor)

Call and return conditions

- The G functions active before the cycle call and the programmable frame remain active beyond the cycle.
- The starting position must be approached in the higher-level program before the cycle is called. The coordinates are programmed in a clockwise coordinate system.

9.2 Program views

You can display a G code program in various ways.

- Program view
- Parameter screen, either with help screen or graphic view

Note

Help screens / animations

Please note that not all the conceivable kinematics can be displayed in help screens and animations of the cyclic support.

Program view

The program view in the editor provides an overview of the individual machining steps of a program.

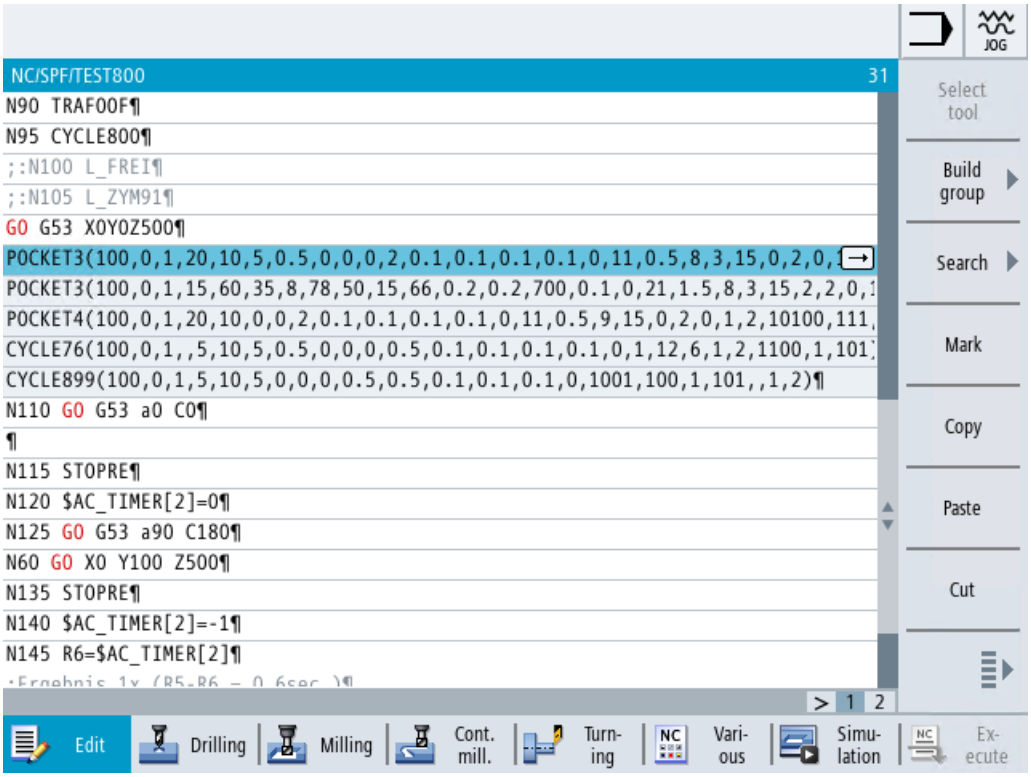
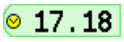
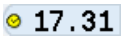
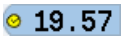
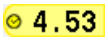


Figure 9-1 Program view of a G code program

Note

In the program editor settings, you define as to whether cycle calls are to be displayed as plain text or in NC syntax. You can also configure the recording of the machining times

Display of the machining times

Display	Meaning
Light green background 	Measured machining time of the program block (automatic mode)
Green background 	Measured machining time of the program group (automatic mode)
Light blue background 	Estimated machining time of the program block (simulation)
Blue background 	Estimated machining time of the program group (simulation)
Yellow background 	Wait time (automatic mode or simulation)

Highlighting of selected G code commands or keywords

In the program editor settings, you can specify whether selected G code commands are to be highlighted in color. The following colors are used as standard:

Display	Meaning
Blue font 	D, S, F, T, M and H functions
Red font 	"G0" motion command
Green font 	"G1" motion command
Blue-green font 	"G2" or "G3" motion command
Gray font 	Comment

Machine manufacturer

You can define further highlight colors in the "sleditorwidget.ini" configuration file. Please refer to the machine manufacturer's instructions.

Synchronization of programs on multi-channel machines

Special commands (e.g. GET and RELEASE) are used on multi-channel machines to synchronize the programs. These commands are marked with a clock symbol.

If the programs of several channels are displayed, the associated commands are displayed in one line.

Display	Meaning
	Synchronization command





In the program view, you can move between the program blocks using the <Cursor up> and <Cursor down> keys.

Parameter screen with help display



Press the <Cursor right> key to open a selected program block or cycle in the program view.

The associated parameter screen with help screen is then displayed.

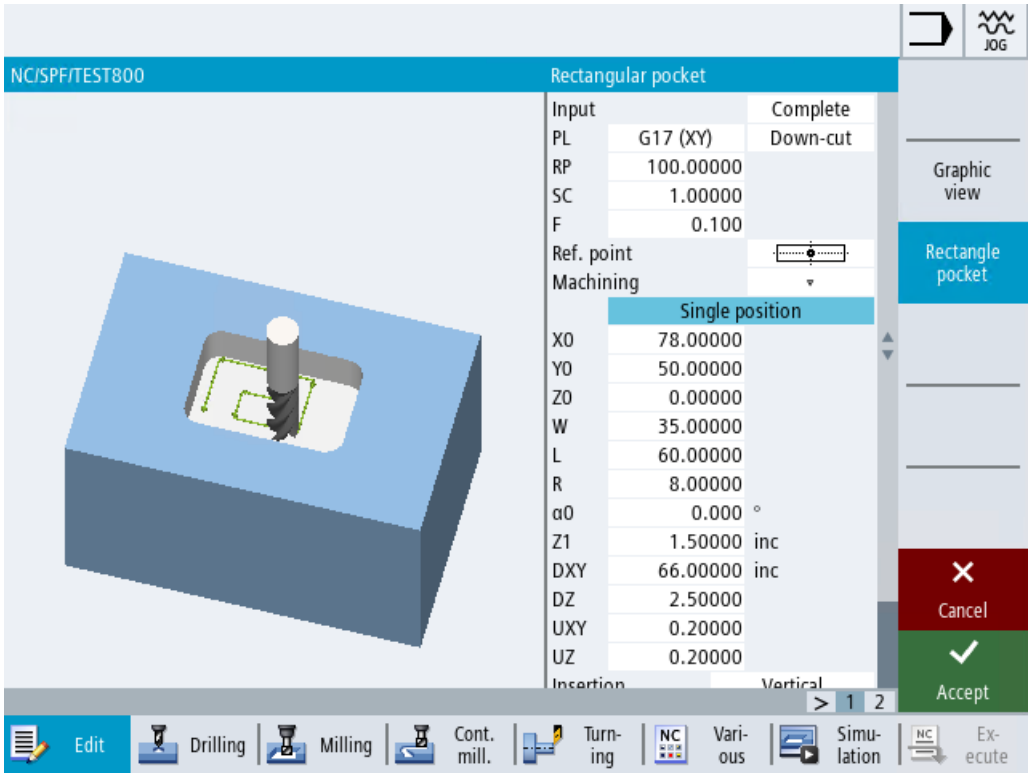


Figure 9-2 Parameter screen with help display

The animated help displays are always displayed with the correct orientation to the selected coordinate system. The parameters are dynamically displayed in the graphic. The selected parameter is displayed highlighted in the graphic.

The colored symbols

Red arrow = tool traverses in rapid traverse

Green arrow = tool traverses with the machining feedrate

Parameter screen with graphic view

Graphic view	Press the "Graphic view" softkey to toggle between the help screen and the graphic view.

Note

Switching between the help screen and the graphic view

The key combination <CTRL> + <G> is also available for the switchover between the help screen and the graphic view.

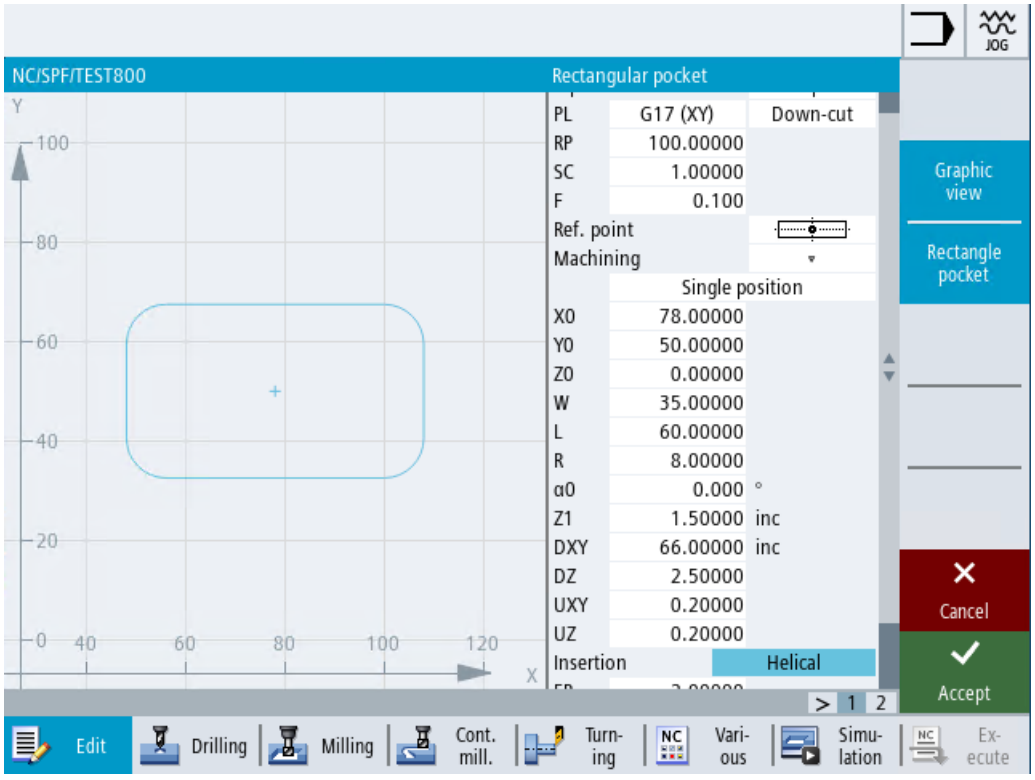


Figure 9-3 Parameter screen with a graphical view of a G code program block

9.3 Program structure

G_code programs can always be freely programmed. The most important commands that are included in the rule:

- Set a machining plane
- Call a tool (T and D)
- Call a work offset
- Technology values such as feedrate (F), feedrate type (G94, G95,....), speed and direction of rotation of the spindle (S and M)
- Positions and calls, technology functions (cycles)
- End of program

For G code programs, before calling cycles, a tool must be selected and the required technology values F, S programmed.

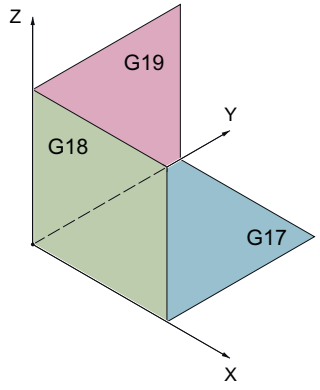
A blank can be specified for simulation.

9.4 Fundamentals

9.4.1 Machining planes

A plane is defined by means of two coordinate axes. The third coordinate axis (tool axis) is perpendicular to this plane and determines the infeed direction of the tool (e.g. for 2½-D machining).

When programming, it is necessary to specify the working plane so that the control system can calculate the tool offset values correctly. The plane is also relevant to certain types of circular programming and polar coordinates.



Working planes

Working planes are defined as follows:

Plane		Tool axis
X/Y	G17	Z
Z/X	G18	Y
Y/Z	G19	X

9.4.2 Current planes in cycles and input screens

Each input screen has a selection box for the planes, if the planes have not been specified by NC machine data.

- Empty (for compatibility reasons to screen forms without plane)
- G17 (XY)
- G18 (ZX)
- G19 (YZ)

There are parameters in the cycle screens whose names depend on this plane setting. These are usually parameters that refer to positions of the axes, such as reference point of a position pattern in the plane or depth specification when drilling in the tool axis.

For G17, reference points in the plane are called X0 Y0, for G18 they are called Z0 X0 - and for G19, they are called Y0 Z0. The depth specification in the tool axis for G17 is called Z1, for G18, Y1 and for G19, X1.

If the entry field remains empty, the parameters, the help screens and the broken-line graphics are displayed in the default plane (can be set via machine data):

- Turning: G18 (ZX)
- Milling: G17 (XY)

The plane is transferred to the cycles as new parameter. The plane is output in the cycle, i.e. the cycle runs in the entered plane. It is also possible to leave the plane fields empty and thus create a plane-independent program.

The entered plane only applies for this cycle (not modal)! At the end of the cycle, the plane from the main program applies again. In this way, a new cycle can be inserted in a program without having to change the plane for the remaining program.

9.4.3 Programming a tool (T)

Calling a tool

1. You are in the part program.



2. Press the "Select tool" softkey.

   The "Tool selection" window is opened.



3. Position the cursor on the desired tool and press the "To program" softkey.

   The selected tool is loaded into the G code editor. Text such as the following is displayed at the current cursor position in the G code editor:
   T="ROUGHINGTOOL100"

   - OR -



4. Press the "Tool list" and "New tool" softkeys.





5. Then select the required tool using the softkeys on the vertical softkey bar, parameterize it and then press the softkey "To program".

   The selected tool is loaded into the G code editor.

6. Then program the tool change (M6), the spindle direction (M3/M4), the spindle speed (S...), the feedrate (F), the feedrate type (G94, G95,...), the coolant (M7/M8) and, if required, further tool-specific functions.

9.5 Generating a G code program

Create a separate program for each new workpiece that you would like to produce. The program contains the individual machining steps that must be performed to produce the workpiece.

Part programs in the G code can be created under the "Workpiece" folder or under the "Part programs" folder.

Procedure



1. Select the "Program Manager" operating area.



2. Select the required archiving location.

Creating a new part program



3. Position the cursor on the folder "Part programs" and press the "New" softkey.



The "New G Code Program" window opens.



4. Enter the required name and press the "OK" softkey.

   The name can contain up to 28 characters (name + dot + 3-character extension). You can use any letters (except accented), digits or the underscore symbol (_).

   The program type (MPF) is set by default.

   The project is created and opened in the Editor.

Creating a new part program for a workpiece



5. Position the cursor on the folder "Workpieces" and press the "New" softkey.



The "New G Code Program" window opens.



6. Select the file type (MPF or SPF), enter the desired name of the program and press the "OK" softkey.

   The project is created and opened in the Editor.

7. Enter the desired G code commands.

9.6 Blank input

9.6.1 Overview

Function

The blank is used for the simulation and the simultaneous recording. A useful simulation can only be achieved with a blank that is as close as possible to the real blank.

Create a separate program for each new workpiece that you would like to produce.
The program contains the individual machining steps that are performed to produce the workpiece.

For the blank of the workpiece, define the shape (cuboid, tube, cylinder, polygon or centered cuboid) and your dimensions.

Manually reclamping the blank

If the blank is to be manually reclamped from the main spindle to the counterspindle for example, then delete the blank.

Example

- Blank, main spindle, cylinder
- Machining
- M0 ; Manually reclamping the blank
- Blank, main spindle, delete
- Blank, counterspindle, cylinder
- Machining

The blank entry always refers to the work offset currently effective at the position in the program.

Note

Swiveling

For programs that use "Swiveling", a 0 swivel must first be made and then the blank defined.

9.6.2 Calling the input screen

Procedure



1. Select the "Program" operating area.





2. Press the "Misc." and "Blank" softkeys.
   The "Blank Input" window opens.

9.6.3 Blank input

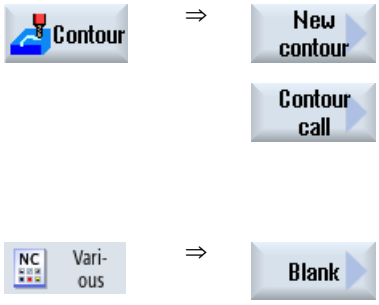
Parameter	Description	Unit
Data for	Selection of the spindle for blank • Main spindle • Counterspindle **Note:** If the machine does not have a counterspindle, then the entry field "Data for" is not applicable.	
Clamping	Selecting the clamping location of the blank • Table All clampings are mounted on a table **Note**: No turning cycles can be used in the program with the "Table" selection. • C1... All clampings are mounted on a rotary axis Note: please refer to the machine manufacturer's specifications.	
Blank	Selecting the blank • Cuboid • Tube • Cylinder • Polygon • Centered cuboid • Delete	
X0	1. Rectangular point X - (only for cuboid)	
Y0	1. Rectangular point Y - (only for cuboid)	
X1	2. Rectangular point X (abs) or 2nd rectangular point X referred to X0 (inc) - (only for cuboid)	
Y1	2. Rectangular point Y (abs) or 2nd rectangular point Y referred to Y0 (inc) - (only for cuboid)	

Parameter	Description	Unit
ZA	Initial dimension	
ZI	Final dimension (abs) or final dimension in relation to ZA (inc)	
ZB	Machining dimension (abs) or machining dimension in relation to ZA (inc)	
XA	Outside diameter – (only for tube and cylinder)	mm
XI	Inside diameter (abs) or wall thickness (inc) – (only for tube)	mm
N	Number of edges – (only for polygon)	
SW or L	Width across flats or edge length – (only for polygon)	
W	Width of the blank - (only for centered cuboid)	mm
L	Length of the blank - (only for centered cuboid)	mm

9.7 Selection of the cycles via softkey

Overview of machining steps

The following softkeys are available to insert machining steps.



Multi-channel view

10

10.1 Multi-channel view

The multi-channel view allows you to simultaneously view several channels in the following operating areas:

- "Machine" operating area
- "Program" operating area

10.2 Multi-channel view in the "Machine" operating area

With a multi-channel machine, you have the option of simultaneously monitoring and influencing the execution of several programs.



Machine manufacturer

Please observe the information provided by the machine manufacturer.

Displaying the channels in the "Machine" operating area

In the "Machine" operating area, you can display 2 - 4 channels simultaneously.

Using the appropriate settings, you can define the sequence in which channels are displayed. Here, you can also select if you wish to hide a channel.

Note

The "REF POINT" function is shown only in the single-channel view.

Multi-channel view

2 - 4 channels are simultaneously displayed in channel columns on the user interface.

- Two windows are displayed one above the other for each channel.
- The actual value display is always in the upper window.
- The same window is displayed for both channels in the lower window.
- You can select the display in the lower window using the vertical softkey bar.
  The following exceptions apply when making a selection using the vertical softkeys:
  - The "Actual values MCS" softkey switches over the coordinate systems of both channels.
  - The "Zoom actual value" and "All G functions" softkeys switch into the single-channel view.

Single-channel view

If you only wish to monitor one channel for your multi-channel machine, then you can set a permanent single-channel view.

Horizontal softkeys

- Block search
  When selecting the block search, the multi-channel view is kept. The block display is displayed as search window.
- Program control
  The "Program Control" window is displayed for the channels configured in the multi-channel view. The data entered here applies for these channels together.
- If you press an additional horizontal softkey in the "Machine" operating area (e.g. "Overstore", "Synchronized actions"), then you change into a temporary single-channel view. If you close the window again, then you return to the multi-channel view.

Switching between single- and multi-channel view



Press the <MACHINE> key in order to briefly switch between the single- and multi-channel view in the machine area.



Press the <NEXT WINDOW> key in order to switch between the upper and lower window within a channel column.

Editing a program in the block display



You can perform simple editing operations as usual with the <INSERT> key in the actual block display.

If there is not sufficient space, you switch over into the single-channel view.

Running-in a program

You select individual channels to run-in the program at the machine.

Requirement

- Several channels have been set-up.
- The setting "2 channels", "3 channels" or "4 channels" is selected.

Displaying/hiding a multi-channel view



1. Select the "Machine" operating area



2. Select the "JOG", "MDA" or "AUTO" mode.

...





3. Press the menu forward key and the "Settings" softkey.





4. Press the "Multi-channel view" softkey.

5. In the window "Settings for Multi-Channel View" in the selection box "View", select the required entry (e.g. "2 channels") and define the channels as well as the sequence in which they are to be displayed.

   In the basic screen for the "AUTO", "MDA" and JOG" operating modes, the upper window of the left-hand and right-hand channel columns are occupied by the actual value window.



6. Press the "T,F,S" softkey if you wish to view the "T,F,S" window.

   The "T,F,S" window is displayed in the lower window of the left-hand and right-hand channel column.

   Note:

   The "T,F,S" softkey is present only for smaller operator panels, i.e. up to OP012.

10.3 Setting the multi-channel view

Setting	Meaning
View	Here, you specify how many channels are displayed. • 1 channel • 2 channels • 3 channels • 4 channels
Channel selection and sequence (for "2 - 4 channels" view)	You specify which channels in which sequence are displayed in the multi-channel view.
Visible (for "2 - 4 channels" view)	Here, you specify which channels are displayed in the multi-channel view. You can quickly hide channels from the view.

Example

Your machine has 6 channels.

You configure channels 1 - 4 for the multi-channel view and define the display sequence (e.g. 1,3,4,2).

In the multi-channel view, for a channel switchover, you can only switch between the channels configured for the multi-channel view; all others are not taken into consideration. Using the <CHANNEL> key, advance the channel in the "Machine" operating area - you obtain the following views: Channels "1" and "3", channels "3" and "4", channels "4" and "2". Channels "5" and "6" are not displayed in the multi-channel view.

In the single-channel view, toggle between all of the channels (1...6) without taking into account the configured sequence for the multi-channel view.

Using the channel menu, you can always select all channels, also those not configured for multi-channel view. If you switch to another channel, which is not configured for the multi-channel view, then the system automatically switches into the single-channel view. There is no automatic switchback into the multi-channel view, even if a channel is again selected, which has been configured for multi-channel view.

Procedure



1. Select the "Machine" operating area.



2. Select the "JOG", "MDA" or "AUTO" mode.





3. Press the menu forward key and the "Settings" softkey.





4. Press the "Multi-channel view" softkey.
   The "Settings for Multi-Channel View" window is opened.
5. Set the multi-channel or single-channel view and define which channels are to be seen in the "Machine" operating area - and in the editor - in which sequence.

Collision avoidance

11

Collision avoidance allows you to avoid collisions and damage while machining a workpiece or creating programs.



Software option

You require the "Collision Avoidance ECO (machine)" software option in order to use this function for geometrically primitive protection area elements.



Software option

You require the "Collision Avoidance (machine, working area)" software option in order to use this function additionally for protection area elements in the STL and NPP data formats.



Software option

You require the "Collision Avoidance ADVANCED (machine, workpiece)" software option in order to use this function additionally for the autonomous realization of collision avoidance application.



Machine manufacturer

Please observe the information provided by the machine manufacturer.

Collision avoidance is based on a machine model. The kinematics of the machine are described as a kinematic chain. For machine parts to be protected, protection areas are attached to these chains. The geometry of the protection areas is defined using protection area elements. The control then knows how they move in the machine coordinate system depending on the position of the machine axes. You then subsequently define the collision pairs, i.e. two protection areas, which are monitored with respect to one another.

The "Collision avoidance" function regularly calculates the clearance from these protection areas. When two protection areas approach one another and a specific safety clearance is reached, an alarm is displayed and the program is stopped before the corresponding traversing block and/or the traversing motion is stopped.

Note

Referenced axes

The positions of the axes in the machine area must be known so that the protection areas can be monitored. For this reason, collision avoidance is only active after the referencing.

NOTICE
No complete machine protection Incomplete models (e.g. machine parts, workpieces that have not been modeled or new objects in the working area) as well as inaccuracies in values or dimenstions can cause collisions.

More information

More information about the kinematic chain and collision avoidance is provided in:

- Basic Functions Function Manual
- Monitoring and Compensating Function Manual

11.1 Activate collision avoidance

Precondition

- Collision avoidance is setup and an active machine model is available.
- The setting "Collision avoidance" has been selected for the AUTO operating mode or for the JOG and MDA operating modes.

Procedure



1. Select the "Machine" operating area.



2. Press the <AUTO> key.



3. Press the "Sim. rec." softkey.



4. Press the "Other views" and "Machine space" softkeys.



During simultaneous recording, an active machine model is displayed.

11.2 Set collision avoidance

Using "Settings", you have the option of separately activating or deactivating the collision monitoring for the Machine operating area (operating modes, AUTO, JOG and MDI) separately for the machine and tools.

Using machine data, you define from which protection level the collision avoidance for the machine or the tool can be activated or deactivated in the operating modes JOG/MDI or AUTO.



Machine manufacturer

Please refer to the machine manufacturer's instructions.

Setting	Effect
JOG/MDI operating mode Collision avoidance	They switch the collision avoidance for the JOG/MDI operating modes on or off.
AUTO mode Collision avoidance	They switch the collision avoidance for the AUTO operating mode on or off depending on machine data $MN_JOG_MODE_MASK **Note**: Please refer to the machine manufacturer's instructions.
JOG/MDI Machine	If the collision monitoring for the JOG/MDI operating modes is activated, then as a minimum, the machine protection areas are monitored. The parameter cannot be changed.
AUTO Machine	If the collision monitoring for the AUTO operating mode is activated, then as a minimum, the machine protection areas are monitored. The parameter cannot be changed.
JOG/MDI Tools	They switch the collision avoidance of the tool protection areas for the operating modes JOG/MDI on or off.
AUTO Tools	They switch the collision avoidance of the tool protection areas for the operating mode AUTO on or off.

Procedure



1. Select the "Machine" operating area.



2. Select the "JOG", "MDI" or "AUTO" mode.





3. Press the menu forward key and the "Settings" softkey.





4. Press the "Collision avoidance" softkey.
   The "Collision Avoidance" window opens.



5. In the "Collision avoidance" line for the required operating modes (e.g. for JOG/MDI), select the entry "On" to activate the collision avoidance or "Off" to deactivate collision avoidance.
6. Deactivate the "Tools" checkbox if you only want to monitor the machine protection areas.

See also

Actual value window (Page 37)

Tool management 12

12.1 Lists for the tool management

All tools and also all magazine locations that have been created or configured in the NC are displayed in the lists in the Tool area.

All lists display the same tools in the same order. When switching between the lists, the cursor remains on the same tool in the same screen segment.

The lists have different parameters and softkey assignments. Switching between lists is a specific change from one topic to the next.

- **Tool list**
  All parameters and functions required to create and set up tools are displayed.
- **Tool wear**
  All parameters and functions that are required during operation, e.g. wear and monitoring functions, are listed here.
- **Magazine**
  You will find the magazine and magazine location-related parameters and functions for the tools/magazine locations here.
- **Tool data OEM**
  This list can be freely defined by the OEM.

Sorting the lists

You can change the sorting within the lists according to:

- the magazine
- the name (tool identifier, alphabetic)
- the tool type
- the T number (tool identifier, numerical)
- the D number

Filtering the lists

You can filter the lists according to the following criteria:

- only display the first cutting edge
- only tools that are ready to use
- only tools that have reached the pre-alarm limit
- only locked tools
- only tools with active code

Search functions

You have the option of searching through the lists according to the following objects:

- Tool
- Magazine location
- Empty location

Representation in the tool management user interface

In some columns of the tool management, values with different meanings are displayed alternately. The column heading always refers to the line with the active cursor. An example of this are the tool monitoring types. Here, depending on the selected monitoring type, the values for tool life monitoring or quantity monitoring are displayed in the same column. Depending on the line where the cursor is positioned, the heading is "Tool life" or "Quantity".

If no monitoring type has been selected in the line, then no heading is displayed.

This allows the large number of parameters and data to be displayed in a particularly compact manner.

12.2 Magazine management

Depending on the configuration, the tool lists support a magazine management.

Magazine management functions

- Press the "Magazine" horizontal softkey to obtain a list that displays tools with magazine-related data.
- The Magazine / Magazine location column is displayed in the lists.
- In the default setting, the lists are displayed sorted according to magazine location.
- The magazine selected via the cursor is displayed in the title line of each list.
- The "Magazine selection" vertical softkey is displayed in the tool list.
- You can load and unload tools to and from a magazine via the tool list.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

12.3 Tool types

A number of tool types are available when you create a new tool. The tool type determines which geometry data is required and how it will be computed.

In the vertical softkey bar you switch between the following tool groups:

- Favorites
- Milling cutter
- Drill
- Special tool

Tool types

The following tools are listed in the "New Tool - Favorites" window:

Type	Identifier	Tool position
120	End mill	
140	Facing tool	
200	Twist drill	
220	Centering tool	
240	Tap	
710	3D probe	
711	Edge probe	
110	Ballhead cutter cylindrical	
111	Ballhead cutter tapered	
121	End mill corner rounding	
155	Bevel cutter	
156	Bevel cutter corner rounding	
157	Tapered die milling tool	

The following tools are listed in the "New Tool - Milling Cutter " window:

Type	Identifier	Tool position
100	Milling cutter	
110	Ballhead cutter cylindrical	
111	Ballhead cutter tapered	
120	End mill	
121	End mill corner rounding	
130	Angle head cutter	
131	Angle head corner rounding	
140	Facing tool	
145	Thread cutter	
150	Side milling cutter	
151	Saw	
155	Bevel cutter	
156	Bevel cutter corner rounding	
157	Tapered die milling tool	
160	Drill and thread milling cutter	

The following tools are listed in the "New Tool - Drills" window:

Type	Identifier	Tool position
200	Twist drill	
205	Solid drill	
210	Boring bar	
220	Centering tool	

Type	Identifier	Tool position
230	Countersink	
231	Counterbore	
240	Tap	
241	Fine thread tap	
242	Whitworth tap	
250	Reamer	

The following tools are listed in the "New Tool - Special Tools " window:

Type	Identifier	Tool position
700	Grooving saw	
710	3D probe	
711	Edge probe	
712	Mono probe	
713	L probe	
714	Star probe	
725	Calibrating tool	
730	Stop	
900	Auxiliary tools	

12.4 Tool dimensioning

This section provides an overview of the dimensioning of tools.

Tool types

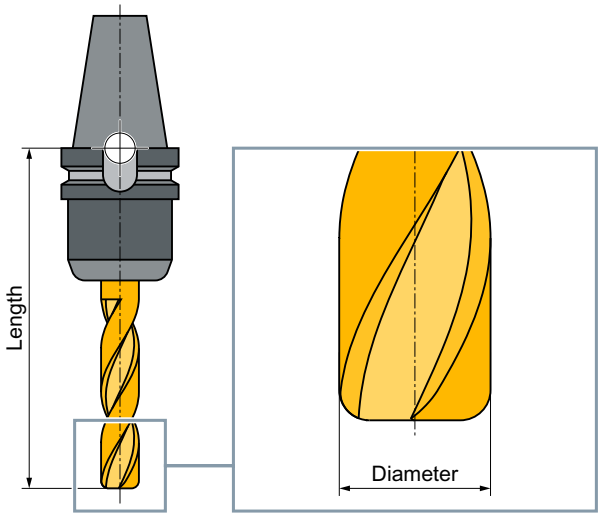


Figure 12-1 End mill (Type 120)

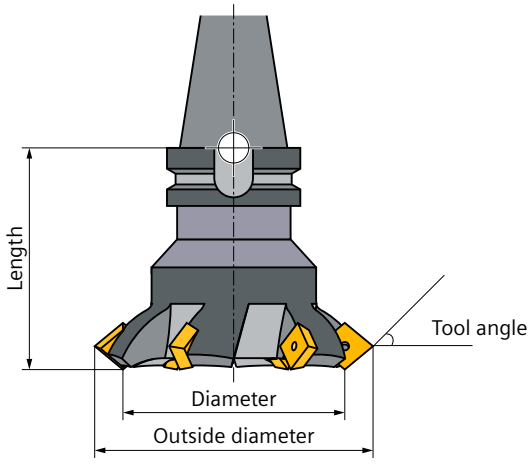


Figure 12-2 Face mill (Type 140)

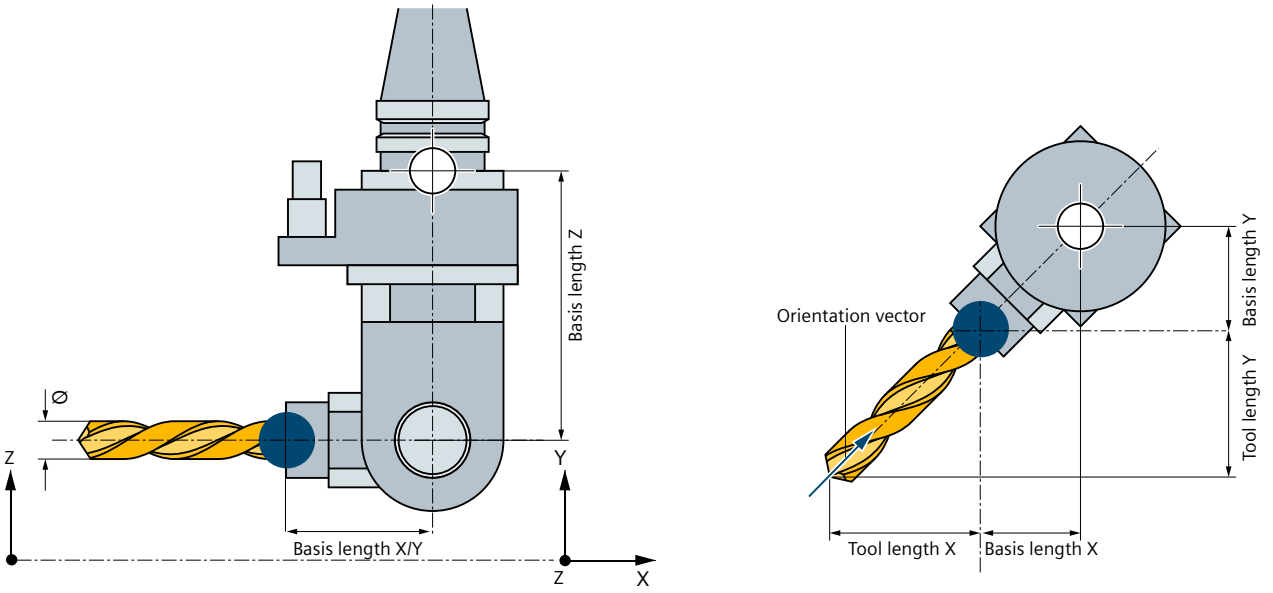


Figure 12-3 Angle head cutter (Type 130)

The following example explains the relevant parameters for the angle head cutter:

- Vector = (x,y,z)
- Basis length Z, e.g. $TC_DP21-23[n] = (100,100,200)
- Tool length, e.g. $TC_DP3-5[n] = (150,150,0)
- Orientation vector, e.g. $TC_DPV/N3-5[n] = (1,1,0)* relevant for orientation functions and PMM

Note

To automatically create tool protection areas (collision avoidance), the following applies:

- The basis length is only taken into account as offset without a protection area ("flying milling cutter")
- The flying milling cutter is emulated as protection area or cylinder, which is obtained in the spanned area of the tool length (L1, L2, L3) with orientation vector ($TC_DPV/N3-5).

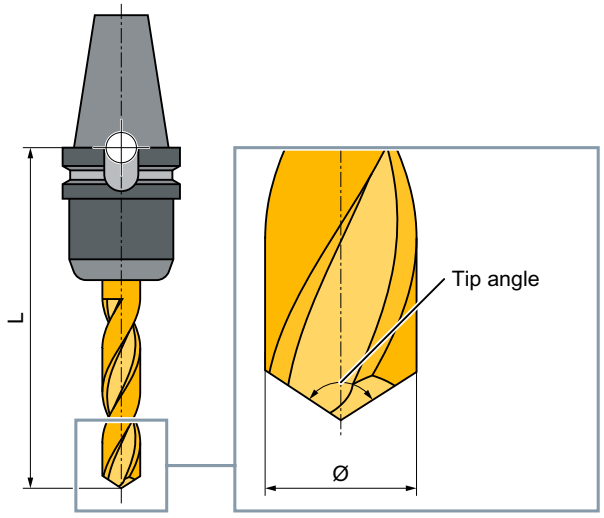


Figure 12-4 Drill (Type 200)

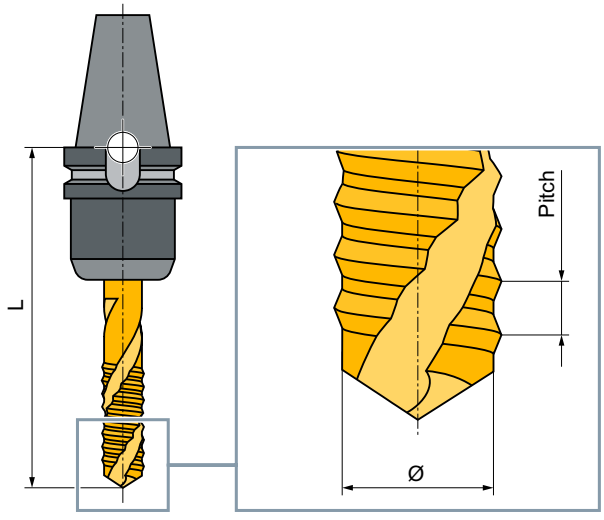


Figure 12-5 Tap (Type 240)

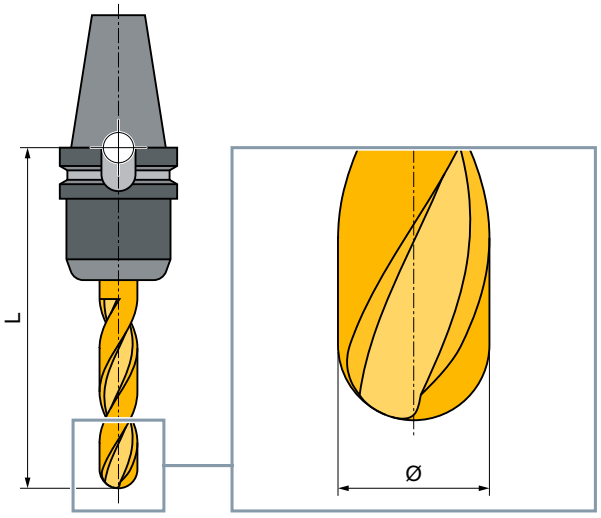


Figure 12-6 3D tool with an example of a cylindrical die-sinking cutter (Type 110)

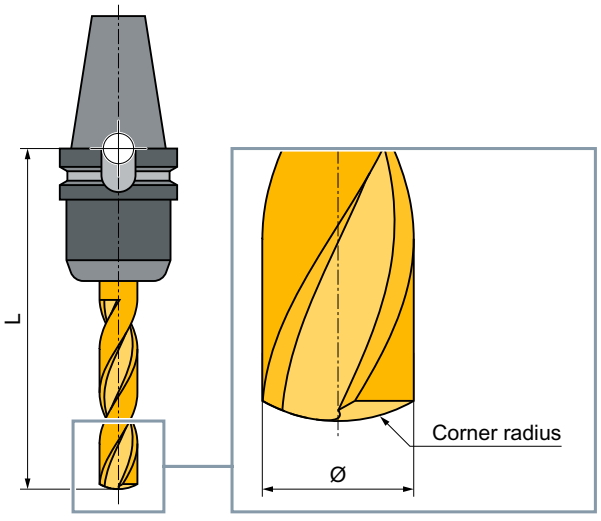


Figure 12-7 3D tool type with an example of a ballhead cutter (Type 111)

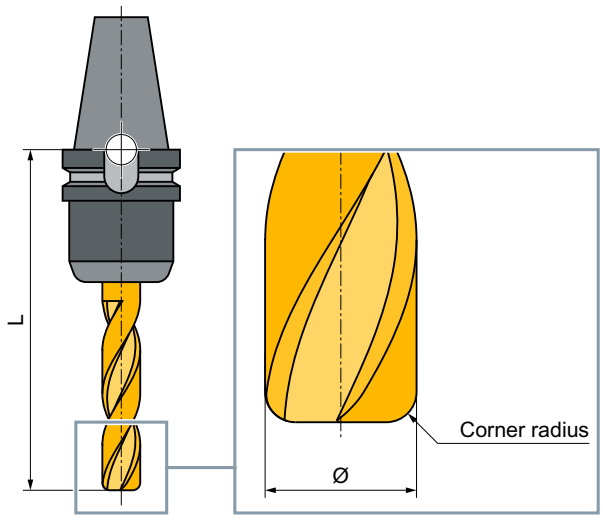


Figure 12-8 3D tool with an example of an end mill with corner rounding (Type 121)

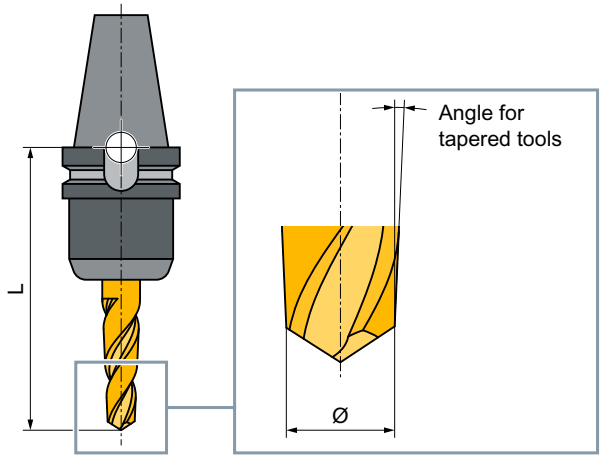


Figure 12-9 3D tool type with an example of a bevel cutter (Type 155)

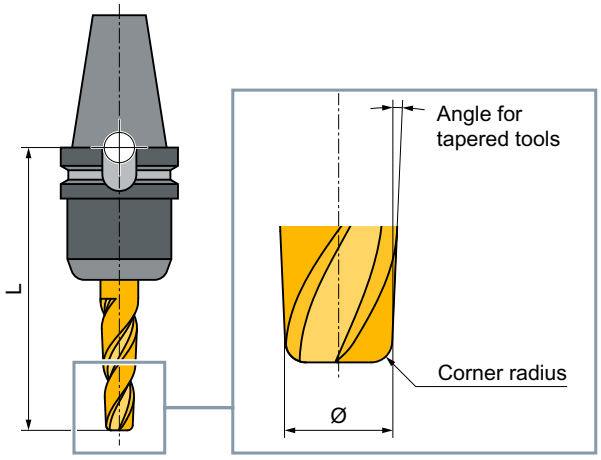


Figure 12-10 3D tool with an example of a bevel cutter with corner rounding (Type 156)

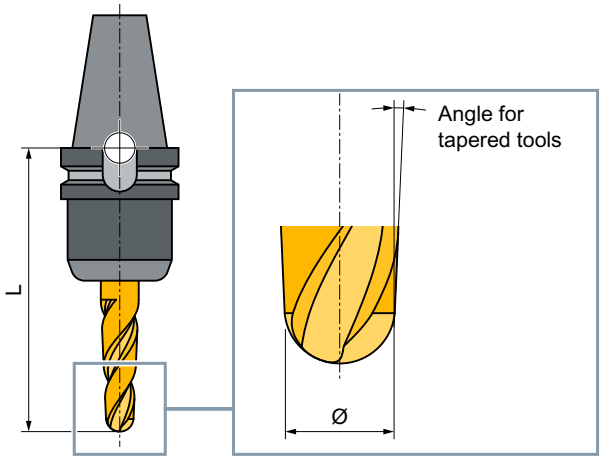
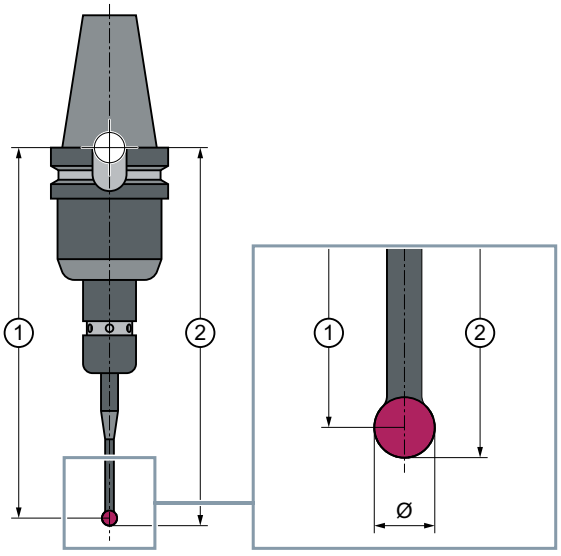


Figure 12-11 3D tool with an example of a tapered die-sinking cutter (Type 157)



① Length m
② Length u

Figure 12-12 3D probe

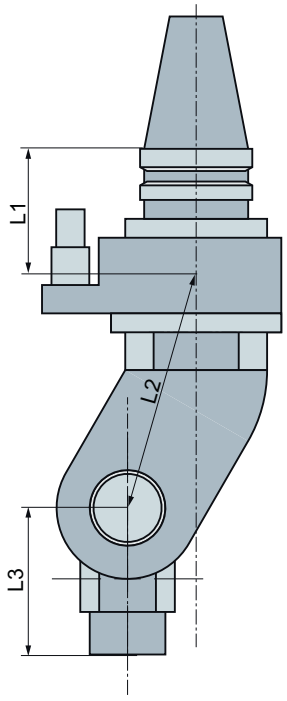


Figure 12-13 Crossover head adapter

L1, L2, L3 are offset geometry lengths.



Machine manufacturer

The tool length of the workpiece probe is measured to the center of the ball (length m) or to the ball circumference (length u).

Observe the information provided by the machine OEM.

Note

An electronic workpiece probe must be calibrated before use.

12.5 Tool list

All parameters and functions that are required to create and set up the tools are displayed in the tool list.

Each tool is uniquely identified by the tool identifier and the replacement tool number.

Tool parameters

Column heading	Meaning
Location BS   * If activated in magazine selection	Magazine/location number • Magazine location numbers The magazine number is specified first, followed by the location number in the magazine. If there is only one magazine, only the location number is displayed. • Load position in the load magazine The following icons can also be displayed for other magazine types (e.g. for a chain): • Spindle location as an icon • Locations for gripper 1 and gripper 2 (applies only when a spindle with dual gripper is used) as icons
Type	Tool type Specific tool offset data is displayed depending on the tool type (represented as an icon).
	You have the option of changing the tool type using the <SELECT> key.
Tool name	The tool is identified by the name and the replacement tool number. You may enter the names as text or numbers. **Note:** The maximum length of tool names is 31 ASCII characters. The number of characters is reduced for Asian characters or Unicode characters. The following special characters are not permitted: | # ".
ST	Replacement tool number (for replacement tool strategy)
D	Cutting edge number
Length	Tool length Geometry data, length
Radius	Tool radius
∅	Tool diameter
Tip angle or Pitch	Tip angle for Type 200 - twist drill and Type 220 - centering tool and Type 230 countersink Pitch for Type 240 - tap

Column heading	Meaning
N	Number of teeth for Type 100 - milling cutter, Type 110 - ballhead mill for cylindrical die-sinking cutter, Type 111 - ballhead mill for tapered die-sinking cutter, Type 120 - end mill, Type 121 - end mill with corner rounding, Type 130 - angle head cutter, Type 131 - angle head cutter with corner rounding, Type 140 - facing tool, Type 150 - side milling cutter, Type 155 - bevel cutter, Type 156 - bevel cutter with corner rounding, Type 157 - tapered die-sinking cutter and type 220 - centering tool.
	Direction of spindle rotation Spindle is not switched on CW spindle rotation CCW spindle rotation
	Coolant 1 and 2 (e.g. internal and external cooling) can be switched on and off. The coolant infeed at the machine does not necessarily have to be set up.
M1 - M4	Other tool-specific functions such as additional coolant infeed, monitoring functions for speed, tool breakage

Further parameters

If you have set up unique cutting edge numbers, these are displayed in the first column.

Column heading	Meaning
D no.	Unique cutting edge number
SN	Cutting edge number
EC	Setup offsets Display of the existing setup offsets

You use the configuration file to specify the selection of parameters in the list.

Machine manufacturer
Please observe the information provided by the machine manufacturer.

More information

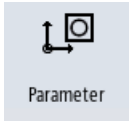
More information on configuring and setting up the tool list is provided in the Tool Management Function Manual.

Icons in the tool list

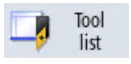
Icon/ Marking		Meaning
Tool type		
Red "X"		The tool is disabled.

Icon/ Marking		Meaning
Yellow triangle pointing downward		The prewarning limit has been reached.
Yellow triangle pointing upward		The tool is in a special state. Place the cursor on the marked tool. A tooltip provides a short description.
Green border		The tool is preselected.
Magazine/location number		
Green double arrow		The magazine location is positioned at the change position.
Gray double arrow (configurable)		The magazine location is positioned at the loading position.
Red "X"		The magazine location is disabled.

Procedure



1. Select the "Parameter" operating area.



2. Press the "Tool list" softkey.
   The "Tool List" window opens.

12.5.1 Creating a new tool

When creating a new tool, the "New tool - favorites" window offers you a number of selected tool types, known as "favorites".

If you do not find the desired tool type in the favorites list, then select the milling, drilling or special tool using the corresponding softkeys.

Procedure



1. The tool list is opened.

2. Place the cursor in the tool list at the position where the new tool should be stored.
   For this, you can select an empty magazine location or the NC tool memory outside of the magazine.
   You may also position the cursor on an existing tool in the area of the NC tool memory. Data from the displayed tool will not be overwritten.



3. Press the "New tool" softkey.



The "New tool - favorites" window opens.

- OR -



If you want to create a tool that is not in the "Favorites" list, press the softkey "Cutters 100-199", "Drill 200-299", or "Spec. tool 700-900".

...



The "New tool - milling cutter", "New tool - drill", or "New tool - special tools" window opens.

4. Select the tool by placing the cursor on the corresponding icon.



5. Press the "OK" softkey.

   The tool is added to the tool list with a predefined name. If the cursor is located on an empty magazine location in the tool list, then the tool is loaded to this magazine location.

The tool creation sequence can be defined differently.

Multiple load points

If you have configured several loading points for a magazine, then the "Select loading point" window appears when a tool is created directly in an empty magazine location or when the "Load" softkey is pressed.

Select the required load point and confirm with the "OK" softkey.

Additional data

If configured accordingly, the "New tool" window opens after the required tool has been selected and confirmed with "OK".

You can define the following data in this window:

- Names
- Tool location type
- Size of tool

Further information

Additional information on the configuring options is provided in the Tool Management Function Manual.

12.5.2 Additional data

The following tool types require geometry data that is not included in the tool list display.

Tools with additional geometry data

Tool type	Additional parameters
111 Tapered ballhead cutter	Head radius Corner radius Wear to the head radius Wear corner radius
114 Barrel milling cutter	Crown radius Reference length crown radius Corner radius Wear corner radius
115 Tapered milling cutter	Crown radius Corner radius Wear corner radius
116 Barrel milling cutter tapered	Crown radius Taper angle Corner radius Wear corner radius
121 End mill with corner rounding	Corner radius
130 Angle head cutter	Geometry length (length X, length Y, length Z) Wear length (Δ length X, Δ length Y, Δ length Z) Adapter length (length X, length Y, length Z) V (direction vector 1 - 6) Vector X, vector Y, vector Z
131 Angle head cutter with corner rounding	Geometry length (length X, length Y, length Z) Corner radius Wear length (Δ length X, Δ length Y, Δ length Z) Adapter length (length X, length Y, length Z) V (direction vector 1 - 6) Vector X, vector Y, vector Z
140 Facing tool	External radius Tool angle
155 Bevel cutter	Taper angle
156 Bevel cutter with corner rounding	Corner radius Taper angle
157 Tapered die-sinking cutter	Taper angle
585 Calibration tool	Geometry length (length X, length Y, length Z) Wear length (Δ length X, Δ length Y, Δ length Z)
700 Slotting saw	Geometry length (length X, length Y, length Z) Wear length (Δ length X, Δ length Y, Δ length Z) Adapter length (length X, length Y, length Z) Geometry (slot width, projection) Wear (slot width, projection)

Tool type	Additional parameters
710 3D-probe milling	Geometry length (length X, length Y, length Z) Wear length (Δ length X, Δ length Y, Δ length Z)
712 Mono-probe	Geometry length (length X, length Y, length Z) Wear length (Δ length X, Δ length Y, Δ length Z)
713 L-probe	Geometry length (length X, length Y, length Z) Wear length (Δ length X, Δ length Y, Δ length Z) Boom length (length)
714 Star-type probe	Geometry length (length X, length Y, length Z) Wear length (Δ length X, Δ length Y, Δ length Z) Outer diameter (⌀)
Crossover head	V1 - V3: Direction vectors L1 - L3: Offset geometry length L2A, L2B: Geometry lengths

You can use the configuration file to specify the data to be displayed for specific tool types in the "Additional Data" window.



Machine manufacturer

Please observe the machine manufacturer's specifications.

Procedure



1. The tool list is opened.
2. In the list, select an appropriate tool, e.g. an angle head cutter.



3. Press the "Additional data" softkey.

   The "Additional Data - ..." window opens.

   The "Additional data" softkey is only active if a tool for which the "Additional Data" window is configured is selected.

12.5.3 Managing several cutting edges

In the case of tools with more than one cutting edge, a separate set of offset data is assigned to each cutting edge. The number of possible cutting edges depends on the controller configuration.

Tool cutting edges that are not required can be deleted.

Procedure



1. The tool list is opened.
2. Position the cursor on the tool for which you would like to store more cutting edges.



3. Press the "Edges" softkey in the "Tool list".



4. Press the "New cutting edge" softkey.
   A new data set is stored in the list.
   The cutting edge number is incremented by one and the offset data is assigned the values of the cutting edge on which the cursor is positioned.
5. Enter the offset data for the 2nd cutting edge.
6. Repeat this process if you wish to create more tool edge offset data.



7. Position the cursor on the cutting edge that you want to delete and press the "Delete cutting edge" softkey.
   The data set is deleted from the list. The first tool cutting edge cannot be deleted.

12.5.4 Deleting a tool

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

Procedure



1. The tool list is opened.
2. In the tool list, place the cursor on the tool that you would like to delete.



3. Press the "Delete tool" softkey.
   A confirmation prompt is displayed.



4. Press the "OK" softkey if you really want to delete the selected tool.
   The tool is deleted.
   If the tool is in a magazine location, it is unloaded and then deleted.

Multiple loading points - tool in magazine location

If you configured several loading points for a magazine, then the "Loading Point Selection" window appears after pressing the "Delete tool" softkey.

Select the required load point and press the "OK" softkey to unload and delete the tool.

12.5.5 Loading and unloading tools

You can load and unload tools to and from a magazine via the tool list. When a tool is loaded, it is taken to a magazine location. When it is unloaded, it is removed from the magazine and stored in the NC memory.

When you are loading a tool, the application automatically suggests an empty location. You may also directly specify an empty magazine location.

You can unload tools from the magazine that you are not using at present. HMI then automatically saves the tool data in the NC memory.

Should you want to use the tool again later, simply load the tool with the tool data into the corresponding magazine location again. Then the same tool data does not have to be entered more than once.

Procedure



1. The tool list is opened.
2. Place the cursor on the tool that you want to load into the magazine (if the tools are sorted according to magazine location number you will find it at the end of the tool list).



3. Press the "Load" softkey.

   The "Load to... " window opens.

   The "... Location" field is defaulted with the number of the first empty magazine location.



4. Press the "OK" softkey to load the tool into the suggested location.

   - OR -



   Enter the location number you require and press the "OK" softkey.

   - OR -



   Press the "Spindle" softkey.

   The tool is loaded into the specified magazine location or spindle.

Loading empty magazine location directly with tool



1. Position the cursor at an empty magazine location where you want to load a tool and press the "Load" softkey.

   The "Load with ..." window opens.



   Select the desired tool in the " ... Tool" field and press the "OK" softkey.

Several magazines

If you have configured several magazines, the "Load to ..." window appears after pressing the "Load" softkey.

If you do not want to use the suggested empty location, then enter your desired magazine and magazine location. Confirm your selection with "OK".

Multiple load points

If you have configured several loading points for a magazine, then the "Loading Point Selection" window appears after pressing the "Load" softkey.

Select the required loading point and confirm with "OK".

Unloading tools



1. Place the cursor on the tool that you would like to unload from the magazine and press the "Unload" softkey.
2. Select the required load point in the "Loading Point Selection" window.



3. Confirm your selection with "OK".

- OR -



Undo your selection with "Cancel".

12.5.6 Selecting a magazine

You can directly select the buffer memory, the magazine, or the NC memory.

Procedure



1. The tool list is opened.



2. Press the "Magazine selection" softkey.

If there is only one magazine, you will move from one area to the next (i.e. from the buffer memory to the magazine, from the magazine to the NC memory, and from the NC memory back to the buffer memory) each time you press the softkey. The cursor is positioned at the beginning of the magazine each time.

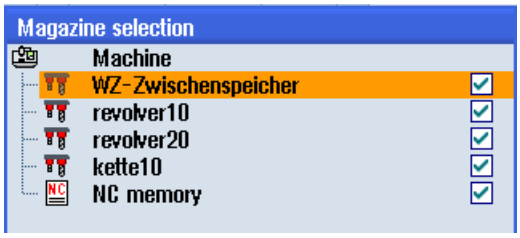
- OR -



If there is more than one magazine, the "Magazine Selection" window opens. Position the cursor on the desired magazine in this window and press the "Go to" softkey.

The cursor jumps directly to the beginning of the specified magazine.

Hiding magazines





Deactivate the checkbox next to the magazines that you do not want to appear in the magazine list.

The magazine selection behavior with multiple magazines can be configured in different ways.



Machine manufacturer

Please observe the information provided by the machine manufacturer.

Further information

Additional information on the configuring options is provided in the Tool Management Function Manual.

12.5.7 Code carrier connection

You have the option of configuring a code carrier connection.

This means that the following functions are available in SINUMERIK Operate:

- Creating a new tool from code carrier
- Unloading tools on code carrier



Software option

In order to use the functions, you require the option "Tool Ident Connection".

Further information

Additional information on tool management with code carrier and the configuration of the user interface in SINUMERIK Operate is provided in the Tool Management Function Manual.

With a code carrier connection, in the list of favorites, there is also a tool available.

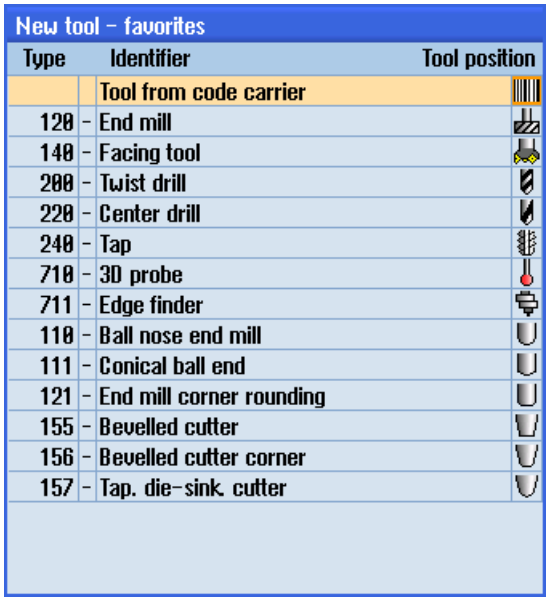


Figure 12-14 New tool from code carrier in the list of favorites

Creating a new tool from code carrier



1. The tool list is opened.

2. Place the cursor in the tool list at the position where the new tool should be created.

   To do this, you can select an empty magazine location or the NC tool memory outside of the magazine.

   You may also position the cursor on an existing tool in the area of the NC tool memory. Data from the displayed tool will not be overwritten.



3. Press the "New tool" softkey.



   The "New Tool - Favorites" window is opened.



4. Position the cursor on the entry "Tool from code carrier" and press the "OK" softkey.

   The tool data is read from the code carrier, and is displayed in the "New tool" window with the tool type, tool name and possibly with certain parameters.



5. Press the "OK" softkey.

   The tool is added to the tool list with the specified name. If the cursor is located on an empty magazine location in the tool list, then the tool is loaded to this magazine location.

The tool creation sequence can be defined differently.

Unloading tool on code carrier



1. The tool list is opened.
2. Place the cursor on the tool that you would like to unload from the magazine and press the "Unload" and "On code carrier" softkeys.
   The tool is unloaded and the data of the tool are written to the code carrier.

According to the corresponding setting, the unloaded tool on the code carrier is deleted from the NC memory after reading out the code carrier.

Deleting tool on code carrier



1. The tool list is opened.
2. Position the cursor on the tool on the code carrier that you want to delete.
3. Press the "Delete tool" and "On code carrier" softkeys.
   The tool is unloaded and the data of the tool are written to the code carrier. The tool is then deleted from the NC memory.

The deletion of the tool can be set differently, i.e. the "On code carrier" softkey is not available.

12.5.8 Managing a tool in a file

If the "Enable tool in/out file" option is activated in the settings for the tool list, an additional entry is available in the list of favorites.

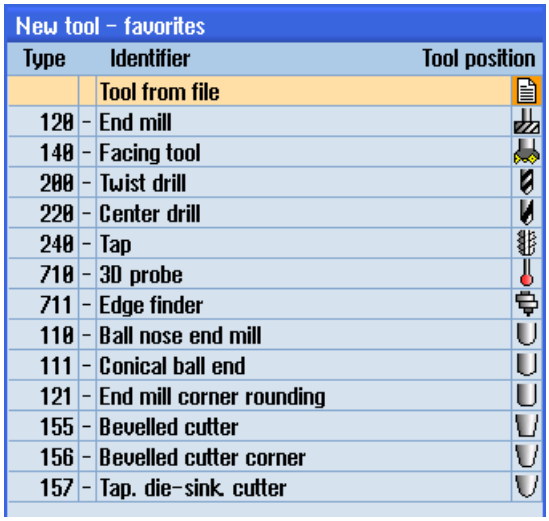


Figure 12-15 New tool from file in the list of favorites

Creating a new tool from a file



1. The tool list is open.

2. Place the cursor in the tool list at the position where the new tool should be created.
   To do this, you can select an empty magazine location or the NC tool memory outside of the magazine.
   You may also position the cursor on an existing tool in the area of the NC tool memory. Data from the displayed tool will not be overwritten.



3. Press the "New tool" softkey.



The "New Tool - Favorites" window is opened.



4. Position the cursor on the entry "Tool from file" and press the "OK" softkey.
   The "Load Tool Data" window opens.



5. Navigate to the required file and press the "OK" softkey.
   The tool data is read from the file, and is displayed in the "New Tool from File" window with the tool type, tool name and possibly with certain parameters.



6. Press the "OK" softkey.
   The tool is added to the tool list with the specified name. If the cursor is located on an empty magazine location in the tool list, then the tool is loaded to this magazine location.

The tool creation sequence can be defined differently.

Unloading a tool in a file



1. The tool list is open.

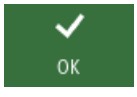




2. Place the cursor on the tool that you would like to unload from the magazine and press the "Unload" and "In file" softkeys.



3. Navigate to the required directory and press the "OK" softkey.



4. Enter the required file name in the "Name" field and press the "OK" softkey.
   The field is preassigned with tool names.
   The tool is unloaded and the tool data is written to the file.

According to the corresponding setting, the unloaded tool is deleted from the NC memory after it has been read out.

Deleting a tool in a file



1. The tool list is open.
2. Position the cursor on the tool that you wish to delete.



3. Press the "Delete tool" and "In file" softkeys.





3. Navigate to the required directory and press the "OK" softkey.



4. Enter the required file name in the "Name" field and press the "OK" softkey.
   The field is preassigned with tool names.
   The tool is unloaded and the tool data is written to the file. The tool is then deleted from the NC memory.

12.6 Tool wear

All parameters and functions that are required during operation are contained in the tool wear list.

Tools that are in use for long periods are subject to wear. You can measure this wear and enter it in the tool wear list. The controller then takes this information into account when calculating the tool length or radius compensation. This ensures a consistent level of accuracy during workpiece machining.

Monitoring types

You can automatically monitor the tools' working times via the workpiece count, tool life or wear.

Note

Combination of monitoring types

You have the option to activate the monitoring of a tool by type or any combination of monitoring types.

In addition, you can disable tools when you no longer wish to use them.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Tool parameters

Column heading	Meaning
Location BS   * If activated in magazine selection	Magazine/location number • The magazine location numbers The magazine number is specified first, followed by the location number in the magazine. If there is only one magazine, only the location number is displayed. • Load position in the load magazine The following icons can also be displayed for other magazine types (e.g. for a chain): • Spindle location as an icon • Locations for gripper 1 and gripper 2 (applies only when a spindle with dual gripper is used) as icons.
Type	Tool type Depending on the tool type (represented by an icon), certain tool offset data is enabled.

Column heading	Meaning
Tool name	The tool is identified by the name and the replacement tool number. You can enter the name as text or number. **Note**: The maximum length of tool names is 31 ASCII characters. The number of characters is reduced for Asian characters or Unicode characters. The following special characters are not permitted: | # ".
ST	Replacement tool number (for replacement tool strategy).
D	Cutting edge number
Δ Length	Length wear
Δ Radius	Radius wear
T C	Selection of tool monitoring - by tool life (T) - by count (C) - by wear (W) The wear monitoring is configured via a machine data item. Please refer to the machine manufacturer's instructions.
Tool life Workpiece count Wear * *Parameter depends on selection in TC	Tool life Number of workpieces Tool wear
Setpoint	Setpoint for tool life, workpiece count, or wear
Prewarning limit	Specification of the tool life, workpiece count or wear at which a warning is displayed.
G	The tool is disabled when the checkbox is selected.

Further parameters

If you have created unique cutting edge numbers, they will be displayed in the first column.

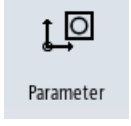
Column heading	Meaning
D no.	Unique cutting edge number
SN	Cutting edge number
SC 	Setting-up offsets Display of the available setup offsets

Icons in the wear list

Icon/ Marking		Meaning
Tool type		
Red "X"		The tool is disabled.

Icon/ Marking		Meaning
Yellow triangle pointing downward		The prewarning limit has been reached.
Yellow triangle pointing upward		The tool is in a special state. Place the cursor on the marked tool. A tooltip provides a short description.
Green border		The tool is preselected.
Magazine/location number		
Green double arrow		The magazine location is positioned at the change position.
Gray double arrow (configurable)		The magazine location is positioned at the loading position.
Red "X"		The magazine location is disabled.

Procedure



1. Select the "Parameter" operating area.



2. Press the "Tool wear" softkey.

See also

Changing a tool type (Page 304)

12.6.1 Reactivating a tool

You can replace disabled tools or make them ready for use again.

Requirements

In order to reactivate a tool, the monitoring function must be activated and a setpoint must be stored.

Procedure



1. The tool wear list is opened.
2. Position the cursor on the disabled tool which you would like to reuse.



3. Press the "Reactivate" softkey.

   The value entered as the setpoint is entered as the new tool life or workpiece count.

   The disabling of the tool is canceled.

Reactivating and positioning

When the "Reactivate with positioning" function is configured, the selected tool's magazine location will also be positioned at the loading point. You can exchange the tool.

Reactivation of all monitoring types

When the "Reactivation of all monitoring types" function is configured, all the monitoring types set in the NC for a tool are reset during reactivation.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Multiple load points

If you have configured several loading points for a magazine, then the "Loading Point Selection" window appears after pressing the "Load" softkey.

Select the required load point and confirm with the "OK" softkey.

Further information

Additional information on the configuring options is provided in the Tool Management Function Manual.

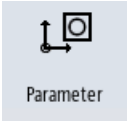
12.7 Tool data OEM

You have the option of configuring the list according to your requirements.

Further information

Additional information on configuring OEM tool data is provided in the Tool Management Function Manual.

Procedure



1. Select the "Parameter" operating area.



2. Press the "OEM tool" softkey.

12.8 Magazine

Tools are displayed with their magazine-related data in the magazine list. Here, you can take specific actions relating to the magazines and the magazine locations.

Individual magazine locations can be location-coded or disabled for existing tools.

Tool parameters

Column heading	Meaning
Location BS   * If activated in magazine selection	Magazine/location number • The magazine location numbers The magazine number is specified first, followed by the location number in the magazine. If there is only one magazine, only the location number is displayed. • Load position in the load magazine The following icons can also be displayed for other magazine types (e.g. for a chain): • Spindle location as an icon • Locations for gripper 1 and gripper 2 (applies only when a spindle with dual gripper is used) as icons
Type	Tool type Depending on the tool type (represented by an icon), certain tool offset data is enabled.
Tool name	The tool is identified by the name and the replacement tool number. You can enter the name as text or number. **Note**: The maximum length of tool names is 31 ASCII characters. The number of characters is reduced for Asian characters or Unicode characters. The following special characters are not permitted: | # ".
ST	Replacement tool number (for replacement tool strategy).
D	Cutting edge number
G	Disabling of the magazine location.
Mag.loc. type	Display of magazine location type.
Tool.loc. type	Display of tool location type.
Ü	Marking of a tool as oversized. The tool occupies two half locations left, two half locations right, one half location top and one half location bottom in a magazine.
P	Fixed location coding. The tool is permanently assigned to this magazine location.

Further parameters

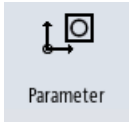
If you have created unique cutting edge numbers, they will be displayed in the first column.

Column heading	Meaning
D no.	Unique cutting edge number
SN	Cutting edge number

Magazine list icons

Icon/ Marking		Meaning
Tool type		
Red "X"		The tool is disabled.
Yellow triangle pointing downward		The prewarning limit has been reached.
Yellow triangle pointing upward		The tool is in a special state. Place the cursor on the marked tool. A tooltip provides a short description.
Green border		The tool is preselected.
Magazine/location number		
Green double arrow		The magazine location is positioned at the change position.
Gray double arrow (configurable)		The magazine location is positioned at the loading position.
Red "X"		The magazine location is disabled.

Procedure



1. Select the "Parameter" operating area.



2. Press the "Magazine" softkey.

See also

Changing a tool type (Page 304)

12.8.1 Position magazine

You can position magazine locations directly on the loading point.

Procedure



1. The magazine list is opened.
2. Place the cursor on the magazine location that you want to position onto the load point.



3. Press the "Position magazine" softkey.
   The magazine location is positioned on the loading point.

Multiple load points

If you have configured several loading points for a magazine, then the "Loading Point Selection" window appears after pressing the "Position magazine" softkey.

Select the desired loading point in this window and confirm your selection with "OK" to position the magazine location at the loading point.

12.8.2 Relocating a tool

Tools can be directly relocated within magazines to another magazine location, which means that you do not have to unload tools from the magazine in order to load them into a different location.

When you are relocating a tool, the application automatically suggests an empty location. You may also directly specify an empty magazine location.

Buffer

You have the option of relocating the tool to buffer locations.



Machine manufacturer
Please refer to the machine manufacturer's specifications.

Procedure



1. The magazine list is opened.
2. Position the cursor on the tool that you wish to relocate to a different magazine location.



3. Press the "Relocate" softkey.
   The "... relocate from location ... to location ..." window is displayed. The "Location" field is pre-assigned with the number of the first empty magazine location.



4. Press the "OK" softkey to relocate the tool to the recommended magazine location.

   - OR -



   Enter the required magazine, enter the location number and press the "OK" softkey.

   - OR -

   Enter the number "9998" or the number "9999" into the "... magazine" field in order to select the buffer as well as the required buffer location in the "Location" field.

   - OR -



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



   The tool is relocated to the specified magazine location, in the spindle or in the buffer.

Several magazines

If you have set up several magazines, then the "...relocate from magazine... location... to..." window appears after pressing the "Relocate" softkey.

Select the desired magazine and location, and confirm your selection with "OK" to load the tool.

12.8.3 Deleting / unloading / loading / relocating all tools

You have the option of deleting or unloading all tools in the magazine list, loading them into the magazine list or relocating them in the magazine list. With one task, the tools are deleted or unloaded from the list or loaded, relocated in the list.

Requirement

The following requirements must be satisfied so that the "Delete all", "Unload all", "Load all" or "Relocate all" softkey is displayed and available:

- Magazine management is set up
- There is no tool in the buffer / in the spindle



Machine manufacturer
Please refer to the machine manufacturer's specifications.

Procedure



1. The magazine list is open.



2. Press the "Delete all" softkey.

   - OR -



   Press the "Unload all" softkey.

   - OR -



   Press the "Load all" softkey.

   - OR -



   Press the "Relocate all" softkey.

   A prompt is displayed as to whether you really want to delete, unload, load or relocate all tools.



3. Press the "OK" softkey to continue with deleting, unloading, loading or relocation of the tools.
   The tools are deleted, unloaded, loaded or relocated in the magazine in ascending magazine location number order.
4. Press the "Cancel" softkey if you wish to cancel the unloading operation.

Multiple load points

For a magazine, if more than one loading point was set-up, using the "Select loading point" softkey, you have the option of opening a window in which you can assign a loading point to a magazine.

12.9 Tool details

12.9.1 Displaying tool details

The following parameters of the selected tool can be displayed using softkeys in the "Tool Details" window.

- Tool data
- Cutting edge data
- Monitoring data

Procedure



1. The tool list, the wear list, the OEM tool list or the magazine is opened.

...



2. Position the cursor to the desired tool.



3. If you are in the tool list or in the magazine, press the ">>" and "Details" softkey.



- OR -



If you are in the wear list or OEM tool list, press the "Details" softkey.



The "Tool Details" window is displayed.

All of the available tool data is displayed.



5. Press the softkey "Cutting edge data" if you wish to display the cutting edge data.



6. Press the softkey "Monitoring data" if you wish to display the monitoring data.

12.9.2 Cutting edge data

The "Tool Details" window provides the following data on the selected tool when the "Cutting edge data" softkey is active.

Parameter	Meaning
Magazine location	The magazine number is specified first, followed by the location number in the magazine. If there is only one magazine, only the location number is displayed.
Tool name	The tool is identified by the name and the sister tool number. You can enter the name as text or number.
ST	Sister tool number (for sister tool strategy)
D quantity	Number of created cutting edges
D	Cutting edge number
Tool type	Tool symbol with type number
	Length
Geometry	Tool length
Wear	Tool wear
	Ø (diameter)
Geometry	Tool diameter
Wear	Tool wear, diameter
Milling/turning machine For type 500 (rougher) and type 510 (finisher)	
 Cutting edge graphic	The cutting edge graphic shows the positioning defined by the holder angle, cut direction and cutting tip angle.
Reference direction 	The reference direction for the holder angle specifies the cut direction.
Holder angle	To determine the cutting edge positioning
Cutting tip angle	To determine the cutting edge positioning
Type 240 - tap	
Pitch	Height of the developed helix parallel to the screw axis
Type 200 - twist drill, type 220 - centering tool and Type 230 - countersink	
Tip angle	Angle is less than 180°
Type 520 -plunge cutter, type 530 - parting tool, type 540 - threading tool	
Cutting tip length	For displaying the tools during the simulation of the program execution.
Cutting tip width	Width of the plunge cutter
Type 110 - ball end mill for cylindrical die-sinking cutter, type 111 - ball end mill for tapered die-sinking cutter, type 120 - end mill, type 121 - end mill with corner rounding, type 130 - angle head cutter, type 140 - facing tool, type 150 - side mill, type 155 - bevel cutter, type 156 - bevel cutter with corner rounding and type 157 - tapered die-sinking cutter	
N	Number of teeth

Parameter		Meaning
For driven tools (drills and milling tools)		
Direction of spindle rotation		Spindle is not switched on
		CW spindle rotation
		CCW spindle rotation
		Coolant 1 and 2 (e.g. internal and external cooling) can be switched on and off. Please refer to the machine manufacturer's specifications
Cutting edge OEM parameters 1 - 2		

Software option

In order to be able to manage the parameter spindle direction of rotation, coolant and tool-specific functions (M1-M4), you require the "ShopMill/ShopTurn" option.

12.9.3 Tool data

The "Tool Details" window provides the following data on the selected tool when the "Tool data" softkey is active.

Parameter	Meaning	
Magazine location	The magazine number is specified first, followed by the location number in the magazine. If there is only one magazine, only the location number is displayed.	
Tool name	The tool is identified by the name and the replacement tool number. You can enter the name as text or number.	
ST	Replacement tool number (for replacement tool strategy)	
D quantity	Number of created cutting edges	
D	Cutting edge number	
Tool state	A	Activate tool
	F	Tool enabled
	G	Block tool
	M	Measure tool
	V	Reaching the prewarning limit
	W	Tool being changed
	P	Tool in fixed location The tool is permanently assigned to this magazine location

Parameter	Meaning	
	I	Tool has been in use: The identifier "Tool has been in use" is set under the following conditions: • The tool was in the spindle, • The status is set to "active", • A non-G0 block was selected/feedrate block was programmed (i.e. machining took place).
Tool size 	Normal	Tool does not require an additional location in a magazine.
	Oversize	The tool occupies two half locations left, two half locations right, one half location top and one half location bottom in a magazine.
	Special size	
	Left	Number of half locations to the left of the tool
	Right	Number of half locations to the right of the tool
Tool OEM parameters 1 - 6	Freely available parameters	

12.9.4 Monitoring data

The "Tool Details" window provides the following data on the selected tool when the "Monitoring data" softkey is active.

Parameter	Meaning
Magazine location	The magazine number is specified first, followed by the location number in the magazine. If there is only one magazine, only the location number is displayed.
Tool name	The tool is identified by the name and the sister tool number. You can enter the name as text or number.
ST	Sister tool number (for sister tool strategy)
D quantity	Number of created cutting edges
D	Cutting edge number
Monitoring type 	T - tool life C - count W - wear The wear monitoring is configured via machine data. Please note the specifications of the machine manufacturer.
	Actual value
Tool life, count and wear	Actual value for tool life, count or wear
	Setpoint
Tool life, count and wear	Setpoint for tool life, count or wear
	Prewarning limit

Parameter	Meaning
Tool life, count and wear	Specification of the tool life, the count or wear at which a warning is displayed.
Monitoring OEM parameters 1 - 8	

12.10 Changing a tool type

Procedure



1. The tool list, the wear list, the OEM tool list or the magazine is opened.

...



2. Position the cursor in the column "Type" of the tool that you wish to change.



3. Press the <SELECT> key.

   The "Tool types - Favorites" window opens.

4. Select the desired tool type in the list of favorites or select using the softkeys "Cutters 100-199", "Drill 200-299", or "Spec. tool 700-900".



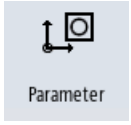
5. Press the "OK" softkey.

   The new tool type is accepted and the corresponding icon is displayed in the "Type" column.

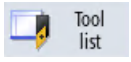
12.11 Sorting tool management lists

When you are working with many tools, with large magazines or several magazines, it is useful to display the tools sorted according to different criteria. Then you will be able to find a specific tool more easily in the lists.

Procedure



1. Select the "Parameter" operating area.



2. Press the "Tool list", "Tool wear" or "Magazine" softkey.

...





3. Press the ">>" and "Sort" softkeys.





The lists are displayed sorted numerically according to magazine location.

Tool types are used to sort tools with the same magazine location. Identical types (e.g. milling cutters), in turn, are sorted according to their radius value.



4. Press the "Acc. to type" softkey to display the tools arranged by tool type. Identical types (e.g. milling cutters) are sorted according to their radius value.

- OR -



Press the "Acc. to name" softkey to display the tool names in alphabetical order.

The replacement tool numbers are used to sort tools with the same names.

- OR -



Press the "Acc. to T number" softkey to display the tool names sorted numerically.

- OR -



Press the "Acc. to D number" softkey to display the tools sorted by D number.

The list is sorted according to the specified criteria.



Machine manufacturer
Please refer to the machine manufacturer's specifications.

12.12 Filtering the tool management lists

The filter function allows you to filter-out tools with specific properties in the tool management lists.

For instance, you have the option of displaying tools during machining that have already reached the prewarning limit in order to prepare the corresponding tools to be loaded.

Filter criteria

- Only display the first cutting edge
- Only tools that are ready to use
- Only tools with active code
- Only locked tools
- Only tools that have reached the prewarning limit
- Only tools with remaining quantity of ... to ...
- Only tools with residual tool life of ... to ...
- Only tools with unloading marking
- Only tools with loading marking



Machine manufacturer
Please observe the information provided by the machine manufacturer.

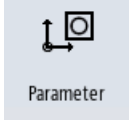
Note

Multiple selection

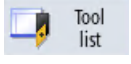
You have the option of selecting several criteria. You will receive an appropriate message if conflicting filter options are selected.

You can configure OR logic operations for the various filter criteria.

Procedure



1. Select the "Parameter" operating area.



2. Press the "Tool list", "Tool wear" or "Magazine" softkey.

...





3. Press the ">>" and "Filter" softkeys.
   The "Filter" window opens.





4. Activate the required filter criterion and press the "OK" softkey.
   The tools that correspond to the selection criteria are displayed in the list.
   The active filter is displayed in the window header.

12.13 Specific search in the tool management lists

There is a search function in all tool management lists, where you can search for the following objects:

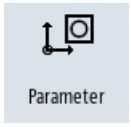
- **Tools**
  - You enter a tool name. You can narrow down your search by entering a replacement tool number.
    You have the option of only entering a part of the name as search term.
  - Enter the D number and, if necessary, activate the "Active D number" checkbox.
- **Magazine locations or magazines**
  If only one magazine is configured, then the search is only made for the magazine location.
  If several magazines are configured, then it is possible to search a specific magazine location in a specific magazine or just to search in a specific magazine.
- **Empty locations**
  If the lists with the location type are used, then the empty location search is made using the location type and location size.



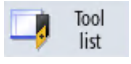
Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure



1. Select the "Parameter" operating area.



2. Press the "Tool list", "Tool wear" or "Magazine" softkey.

...





3. Press the ">>" and "Search" softkeys.

...





4. Press the "Tool" softkey if you wish to search for a specific tool.

   - OR -



   Press the "Magazine location" softkey if you wish to search for a specific magazine location or a specific magazine.

- OR -



Press the "Empty location" softkey if you wish to search for a specific empty location.

12.14 Multiple selection in the tool management lists

Multiple selection is available in the tool management lists.

You select several tools, e.g. to delete them from the list, load them, unload them, or move them to other magazine locations.

For multiple selection, you use either the "Select" softkey, the touch or mouse controls, or the cursor keys.

You can use multiple selection for the following tasks:

- Deleting a tool (Page 281)
- Loading and unloading a tool (Page 282)
- Relocating a tool (Page 296)
- Unloading or deleting a tool on the code carrier (Page 284)
- Equipping a multitool with tools (Page 315)
- Removing tools from a multitool (Page 316)
- Deleting a multitool (Page 317)
- Loading and unloading a multitool (Page 317)
- Relocating a multitool (Page 319)

12.15 Working with multitool

Multitool

Using a multitool you have the possibility of storing more than one tool at a magazine location.

The multitool itself has two or more locations to accept tools. The tools are directly mounted on the multitool. The multitool is located at a location in the magazine.

Geometrical arrangement of the tools on the multitool

The geometrical arrangement of the tools is defined by the clearance between the locations on the multitool.

The clearance between the locations can be defined as follows:

- Using the multitool location number or
- using the angle of the multitool location

If angle is selected here, then the value of the angle must be entered for each multitool location.

Regarding loading and unloading in a magazine, the multitool is treated as a single unit.

Further information

Additional information on the configuring options is provided in the Tool Management Function Manual.

12.15.1 Tool list for multitool

If you work with a multitool, the tool list is supplemented by the column for the multitool location number. As soon as the cursor is at a multitool in the tool list, certain column headings change.

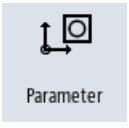
Column header	Meaning
Location	Magazine/location number
MT loc.	Multitool location number
TYPE	Symbol for multitool
Multitool name	Name of the multitool

TOA 1 Tool list — NC memory

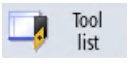
Loc.	MT LO.	Type	Tool name	ST	D	Length	⌀	Pitch			1	2
			Gewbo_M4-7G_SA	1	1	0.000	4.000	0.700			☑	☐
			Gewbo_M8-7G_SA	1	1	0.000	8.000	1.250			☑	☐
			Gewbo_M10-7G_SA	1	1	0.000	10.000	1.500			☑	☐
			GEWFORM_M12	1	1	0.000	12.000	1.250			☑	☐
		+	C-Schruppst_0.8	1	1	0.000	0.000	47.0	0		☑	☐

Figure 12-16 Tool list with multitool in the spindle

Procedure



1. Select the "Parameter" operating area.



2. Press the "Tool list" softkey.
   The "Tool list" window is opened.

12.15.2 Create multitool

The multitool can be selected in the list of favorites as well as in the list of special tool types.

New tool – favorites

Type		Identifier	Tool position
120	-	End mill	
140	-	Facing tool	
200	-	Twist drill	
220	-	Center drill	
240	-	Tap	
710	-	3D probe	
711	-	Edge finder	
110	-	Ball nose end mill	
111	-	Conical ball end	
121	-	End mill corner rounding	
155	-	Bevelled cutter	
156	-	Bevelled cutter corner	
157	-	Tap. die-sink. cutter	
		Multitool	

Figure 12-17 List of favorites with multitool

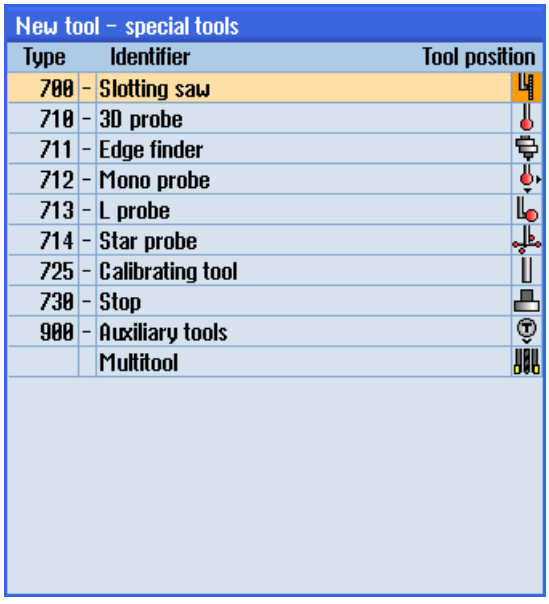


Figure 12-18 Selection list for special tools with multitool

Note

With respect to function "Turning on a milling machine", the multitool angle is not taken into account. For this function, the rotation must be implemented using the clamping angle.

Procedure



1. The tool list is opened.

2. Position the cursor at the position where the tool is to be created.
   For this, you can select an empty magazine location or the NC tool storage outside the magazine.
   You may also position the cursor on an existing tool in the NC tool storage area. Data from the displayed tool will not be overwritten.



3. Press the "New tool" softkey.

   The "New Tool - Favorites" window opens.

   - OR -



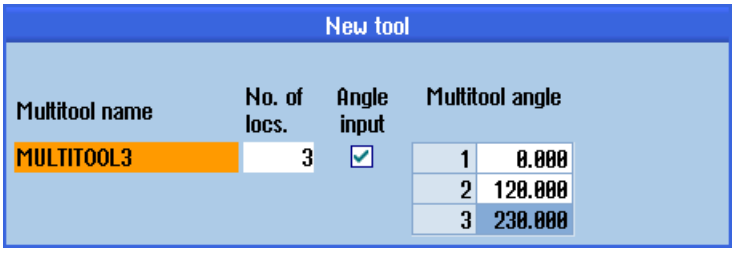
   Press the softkey "Special tool 700-900".



4. Select the multitool and press the "OK" softkey.
   The "New Tool" window appears.

5. Enter the multitool name and define the number of multitool locations.

   If you wish to define the clearance of the tools based on the angle, activate the "Angle input" checkbox, and for each multitool location, enter the clearance to the reference location as angular value.



   The multitool is created in the tool list.

Note

The tool creation sequence can be defined differently.

Machine manufacturer

Refer to the machine manufacturer's specifications.

12.15.3 Equipping multitool with tools

Precondition

A multitool has been created in the tool list.

Procedure

Tool list

1. The tool list is opened.

Equipping the multitool with a new tool

SELECT

2. Select the required multitool, position the cursor on an empty multitool location.

New tool

3. Press the "New tool" softkey.

4. Using the appropriate selection list, e.g. favorites, select the required tool.

Loading the multitool

SELECT

2. Select the required multitool, position the cursor on an empty multitool location.



3. Press the "Load" softkey.
   The "Load with..." window opens.

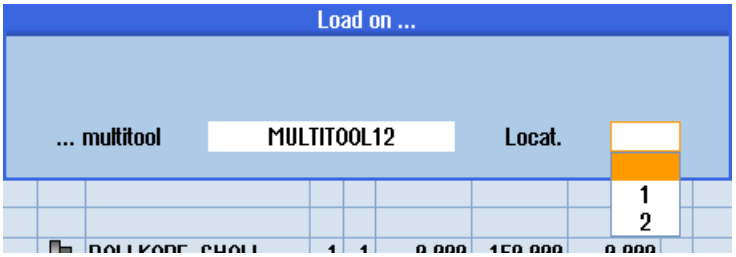
4. Select the required tool.

Loading the tool into the multitool

2. Position the cursor on the tool that you want to load into the multitool.



3. Press the "Load" and "Multitool" softkeys.
   The "Load on ..." window opens.



4. Select the required multitool and the multitool location to which you wish to load the tool.

12.15.4 Removing a tool from multitool

If the multitool was mechanically re-equipped, then the old tools in the tool list must be removed from the multitool.

To do this, the cursor is positioned at the line where the tool which is to be removed is located. When unloading, the tool is automatically saved in the tool list outside the magazine in the NC memory.

Procedure



1. The tool list is opened.



2. Position the cursor on the tool that you would like to unload from the multitool and press the "Unload" softkey.

   - OR -



   Position the cursor on the tool that you want to remove and delete from the multitool and press the "Delete tool" softkey.

12.15.5 Deleting multitool

Procedure



1. The tool list is opened.
2. Position the cursor on the multitool that you wish to delete.



3. Press the "Delete multitool" softkey.
   The multitool with all of the tools that are located in it is deleted.

12.15.6 Loading and unloading multitool

Procedure



1. The tool list is opened.

Loading a multitool into the magazine

2. Position the cursor at the multitool that you wish to load into the magazine.



3. Press the "Load" softkey.
   The "Load on ..." window opens.
   The field "... Locat." is initialized with the number of the first empty magazine location.



4. Press the "OK" softkey if you want to load the multitool to the recommended empty location.

   - OR -



   Enter the location number you require and press the "OK" softkey.

   The multitool together with the tools in it is loaded to the specified magazine location.

Loading a multitool into a magazine

2. Position the cursor on the required empty magazine location.



3. Press the "Load" softkey.
   The "Load with" window opens.



4. Select the required multitool.



5. Press the "OK" softkey.

Unloading a multitool

2. Position the cursor on the multitool that you wish to unload from the magazine.



3. Press the "Unload" softkey.

   The multitool is removed from the magazine and is saved in the NC memory at the end of the tool list.

12.15.7 Reactivating the multitool

Multitool and tools located on the multitool can be disabled independently of one another.

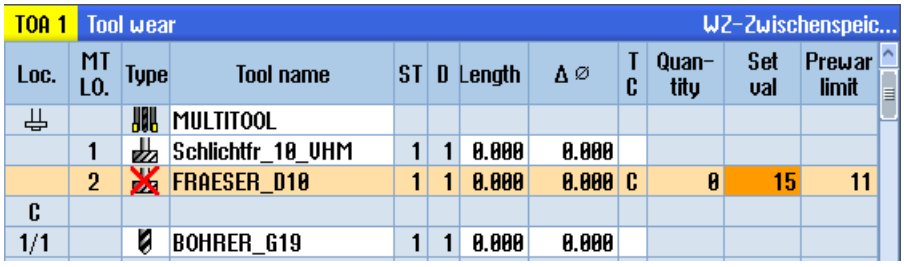
If a multitool is disabled, then the tools of the multitool can no longer be changed in using a tool change.

If only one tool on a multitool has a set monitoring function and the lifetime or the unit quantity has expired, then the tool and the multitool on which the tool is located are disabled. The other tools on the multitool are not disabled.

Machine manufacturer

Please observe the information provided by the machine manufacturer.

If several tools with monitoring are mounted on the multitool and the lifetime or unit quantity has expired for one tool, then only this tool is disabled.



Reactivating

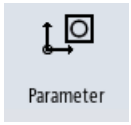
If a tool with expired lifetime or unit quantity that is mounted on a multitool is reactivated, then for this tool, the lifetime/unit quantity is set to the setpoint and the tool and the multitool are re-enabled (disable status is removed).

If a multitool is reactivated, on which tools with monitoring are mounted, then the lifetime/ unit quantity for all tools on the multitool are set to the setpoint no matter whether the tools are disabled or not.

Requirements

In order to reactivate a tool, the monitoring function must be activated and a setpoint must be stored.

Procedure



1. Select the "Parameter" operating area.



2. Press the "Tool wear" softkey.

3. Position the cursor at the multitool that is disabled and which you would like to reactivate.

   - OR -

   Position the cursor on the tool that you want to reactivate again.



4. Press the "Reactivate" softkey.

   The value entered as the setpoint is entered as the new tool life or workpiece count.

   The tool and the multitool are then no longer disabled (the disable is withdrawn).

Reactivating and positioning

When the "Reactivate with positioning" function is configured, then also the magazine location at which the selected multitool is located is positioned at the loading point. You can exchange the multitool.

Reactivation of all monitoring types

When the "Reactivation of all monitoring types" function is configured, all the monitoring types set in the NC for a tool are reset during reactivation.



Machine manufacturer

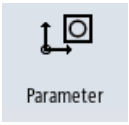
Please observe the information provided by the machine manufacturer.

12.15.8 Relocating a multitool

Multitools can be directly relocated within magazines to another magazine location, which means that you do not have to unload multitools with the associated tools from the magazine in order to relocate them to a different location.

When you are relocating a multitool, the system automatically recommends an empty location. You may also directly specify an empty magazine location.

Procedure



1. Select the "Parameter" operating area.



2. Press the "Magazine" softkey.

3. Position the cursor at the multitool that you wish to relocate to a different magazine location.



4. Press the "Relocate" softkey.

   The "... relocate from location ... to location ..." window is displayed. The "Location" field is pre-assigned with the number of the first empty magazine location.



5. Press the "OK" softkey to locate the multitool at the recommended magazine location.

   - OR -

   Enter the required magazine number in the "...magazine" field and the required magazine location number in "Location" field.

   Note:

   Please refer to the machine manufacturer's specifications.

   Press the "OK" softkey.



   The multitool with the tools is relocated to the specified magazine location.

12.15.9 Positioning a multitool

You can position a magazine. In this case, a magazine location is positioned to the loading point.

Multitools that are located in a spindle can also be positioned. The multitool is rotated and therefore the multitool location involved is brought into the machining position.

Procedure



1. The magazine list is opened.

   The multitool is in the spindle.

2. Position the cursor on the multitool location that you want to bring into the machining position.



3. Press the "Position multitool" softkey.

12.16 Settings for tool lists

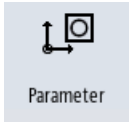
The "Settings" window provides the following options to set the view in the tool lists:

- Display only one magazine in the magazine sorting
  - You limit the display to one magazine. The magazine is displayed with the assigned buffer locations and the not-loaded tools.
  - You use a configuration to specify whether the "Magazine selection" softkey jumps to the next magazine or whether the "Magazine Selection" dialog for changing switches to any magazine.
- Display only spindles in the buffer
  In order to display during operation only the spindle location, the remaining locations of the buffer will be hidden.
- Enable tool in/out file
  - When creating a new tool, the tool data can be loaded from a file.
  - When deleting or unloading a tool, the tool data can be backed up in a file.
- Activating the adapter-transformed view
  - The geometry lengths and the operation offsets are displayed transformed in the tool list.
  - The wear lengths and the sum offsets are displayed transformed in the tool wear list.

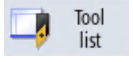


Machine manufacturer
Please observe the information provided by the machine manufacturer.

Procedure



1. Select the "Parameter" operating area.



2. Press the "Tool list", "Tool wear" or "Magazine" softkey.

...





3. Press the "Continue" and "Settings" softkeys.



4. Activate the checkbox for the desired setting.

Managing programs

13

13.1 Overview

You can access programs at any time via the Program Manager for execution, editing, copying, or renaming.

Programs that you no longer require can be deleted to release their storage space.

NOTICE
Execution from USB flash drive Direct execution or simulation from a USB flash drive is not recommended. There is no protection against contact problems, falling out, breakage through knocking or unintentional removal of the USB flash drive during operation. Disconnecting it during operation will result in the stopping of the machining and thus to the workpiece being damaged.

Storage for programs

Possible storage locations are:

- NC
- Local drive
- Network drives
- USB drives
- FTP drives
- V24



Software options

To display the "Local drive" softkey, you require option "Additional HMI user memory on NCU memory card".

Data exchange with other workstations

You have the following options for exchanging programs and data with other workstations:

- USB drives (e.g. USB flash drive)
- Network drives
- FTP drive

Choosing storage locations

In the horizontal softkey bar, you can select the storage location that contains the directories and programs that you want to display. In addition to the "NC" softkey, via which the file system data can be displayed, additional softkeys can be displayed.

The "USB" softkey can only be used when an external storage medium is connected (e.g. USB flash drive on the USB port of the operator panel).

Displaying documents

You can display documents on all drives of the Program Manager (e.g. in the local drive or USB) and via the data tree of the system data. Various data formats are supported:

- PDF
- HTML
  It is not possible to preview HTML documents.
- Various graphic formats (e.g. BMP or JPEG)
- DXF



Software options
You require the "DXF reader" option in order to display DXF files.

Note
FTP drive

It is not possible to preview documents on the FTP drive.

Structure of the directories

In the overview, the icons in the left-hand column have the following meaning:


Directory


Program

All directories have a plus sign when the program manager is called for the first time.

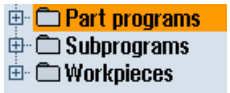


Figure 13-1 Program directory in the program manager

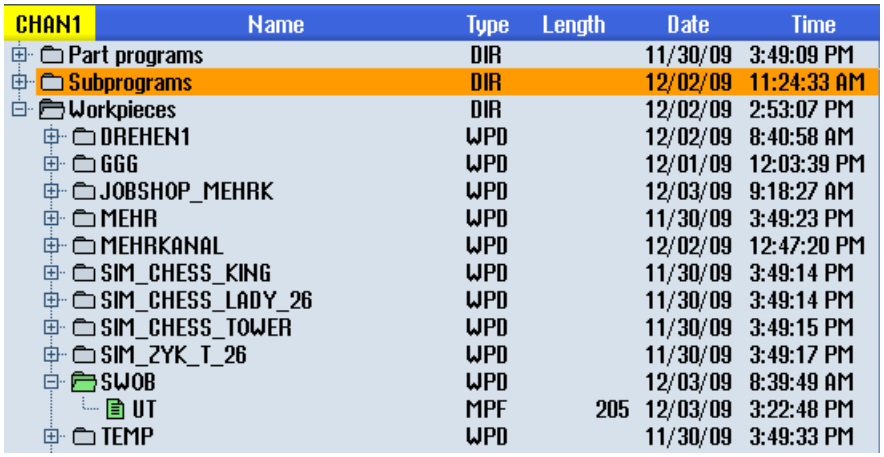
The plus sign in front of empty directories is removed after they have been read for the first time.

The directories and programs are always listed complete with the following information:

- Name
  The name can be a maximum of 24 characters long.
  Permissible characters include all upper-case letters (without accents), numbers and underscores.
- Type
  Directory: WPD
  Program: MPF
  Subprogram: SPF
  Initialization programs: INI
  Job lists: JOB
  Tool data: TOA
  Magazine assignment: TMA
  Zero points: UFR
  R parameters: RPA
  Global user data/definitions: GUD
  Setting data: SEA
  Protection zones: PRO
  Sag: CEC
- Size (in bytes)
- Date/time (of creation or last change)

Active programs

Selected, i.e. active programs are identified using a green icon.



13.1.1 NC memory

The complete NC working memory is displayed along with all tools and the main programs and subroutines.

You can create further subdirectories here.

Procedure



1. Select the "Program Manager" operating area.



2. Press the "NC" softkey.

13.1.2 Local drive

Workpieces, main and subprograms that are saved in the user memory of the memory card or on the local hard disk are displayed.

For archiving, you have the option of mapping the structure of the NC memory system or to create a separate archiving system.

You can create any number of subdirectories here, in which you can store any files (e.g. text files with notes).



Software options
To display the "Local Drive." you require option "Additional HMI user memory on NCU memory card".

Procedure



1. Select the "Program Manager" operating area.



2. Press the "Local drive" softkey.

On the local drive, you have the option of mapping the directory structure of the NC memory. This also simplifies the search sequence.

Creating Directories



1. The local drive is selected.



2. Position the cursor on the main directory.



3. Press the "New" and "Directory" softkeys.
   The "New Directory" window opens.





4. In the "Name" entry field, enter "mpf.dir", "spf.dir" and "wks.dir" and press the "OK" softkey.
   The directories "Part programs", "Subprograms" and "Workpieces" are created below the main directory.

13.1.3 USB drives

USB drives enable you to exchange data. For example, you can copy to the NC and execute programs that were created externally.

NOTICE
Interruption of operation
Direct execution and simulation from the USB flash drive is not recommended, because machining can be undesirably interrupted, therefore resulting in workpiece damage.

Partitioned USB flash drive (TCU)

If the USB flash drive has several partitions, these are displayed in a tree structure as a subtree (01,02,...).

For EXTCALL calls, enter the partition (e.g. USB:/02/... or //ACTTCU/FRONT/02/... or //ACTTCU/FRONT,2/... or //TCU/TCU1/FRONT/02/...)

You can also configure any partition (e.g. //ACTTCU/FRONT,3).

Procedure



1. Select the "Program Manager" operating area.



2. Press the "USB" softkey.

Note

The "USB" softkey can only be operated when a USB flash drive is inserted in the front interface of the operator panel.

13.1.4 FTP drive

The FTP drive offers you the following options - to transfer data, e.g. part programs, between your control system and an external FTP server.

You have the option of archiving any files in the FTP server by creating new directories and subdirectories.

Note

Selecting a program / execution

It is not possible to select a program directly on the FTP drive, and change to execution in the "Machine" operating area.

Precondition

User name and password have been set up in the FTP server.

Procedure



1. Select the "Program manager" operating area.



2. Press the "FTP" softkey.

   When selecting the FTP drive for the first time, a login window is displayed.



3. Enter the user name and password and press the "OK" softkey to log into the FTP server.

   The content of the FTP server with its folders is displayed.



4. Press the "Log out" softkey after the required data processing has been completed.

   The connection to the FTP server is disconnected. In order to reselect the FTP drive, you must log on again.

13.2 Opening and closing a program

To view a program in more detail or modify it, open the program in the editor.

With programs that are in the NCK memory, navigation is already possible when opening. The program blocks can only be edited when the program has been opened completely. You can follow the opening of the program in the dialog line.

With programs that are opened via local network, USB FlashDrive or network connections, navigation is only possible when the program has been opened completely. A progress message box is displayed when opening the program.

Note

Channel changeover in the editor

When opening the program, the editor is opened for the currently selected channel. This channel is used to simulate the program.

If you change over a channel in the editor, this does not influence the editor. Only when closing the editor do you change into the other channel.

Procedure



1. Select the "Program manager" operating area.
2. Select the desired storage location and position the cursor on the program that you would like to edit.
3. Press the "Open" softkey.



- OR -



Press the <INPUT> key.

- OR -



Press the <Cursor right> key.

- OR -

Double-click the program.

The selected program is opened in the "Editor" operating area.

4. Now make the necessary program changes.
5. Press the "NC Select" softkey to switch to the "Machine" operating area and begin execution.



When the program is running, the softkey is deactivated.



Closing the program



Press the ">>" and "Exit" softkeys to close the program and editor again.



- OR -



If you are at the start of the first line of the program, press the <Cursor left> key to close the program and the editor.



To reopen a program you have exited with "Close", press the "Program" key.

Note

A program does not have to be closed in order for it to be executed.

13.3 Executing a program

When you select a program for execution, the control switches automatically to the "Machine" operating area.

Program selection

Select the workpieces (WPD), main programs (MPF) or subprograms (SPF) by placing the cursor on the desired program or workpiece.

For workpieces, the workpiece directory must contain a program with the same name. This program is automatically selected for execution (e.g. when you select the workpiece SHAFT.WPD, the main program SHAFT.MPF is automatically selected).

If an INI file of the same name exists (e.g. SHAFT.INI), it will be executed once at the first part program start after selection of the part program. Any additional INI files are executed in accordance with machine data MD11280 $MN_WPD_INI_MODE.

MD11280 $MN_WPD_INI_MODE=0:

The INI file with the same name as the selected workpiece is executed. For example, when you select SHAFT1.MPF, the SHAFT1.INI file is executed upon <CYCLE START>.

MD11280 $MN_WPD_INI_MODE=1:

All files of type SEA, GUD, RPA, UFR, PRO, TOA, TMA and CEC which have the same name as the selected main program are executed in the specified sequence. The main programs stored in a workpiece directory can be selected and processed by several channels.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure



1. Select the "Program manager" operating area.

2. Select the desired storage location, and position the cursor on the workpiece/program that you would like to execute.



3. Press the "Select" softkey.

   The control switches automatically into the "Machine" operating area.

   - OR -



   If the selected program is already opened in the "Program" operating area, press the "Execute NC" softkey.



   Press the <CYCLE START> key.

   Machining of the workpiece is started.

Note

Program selection from external media

If you execute programs from an external drive (e.g. network drive), you require the "Execution from external storage (EES)" software option.

13.4 Creating a directory/program/job list/program list

13.4.1 File and directory names

The following rules are to be observed when assigning names to files and directories:

- All letters are permissible (with the exception of umlauts, special characters, language-specific special characters, Asian or Cyrillic characters)
- All digits
- Underscores (_).
- The name can be a maximum of 24 characters long

Note

To avoid problems with Windows applications, do **not** use the following terms as program names or directory titles:

- CON, PRN, AUX, NUL
- COM1, COM2, COM3, COM4, COM5, COM6, COM7, COM8, COM9
- LPT1, LPT2, LPT3, LPT4, LPT5, LPT6, LPT7, LPT8, LPT9

Please note that these terms, including those with extensions (e.g. LPT1.MPF, CON.INI) can lead to problems if they are transferred to a Windows environment by copying, archiving or uploading, for example.

13.4.2 Creating a new directory

Directory structures help you to manage your program and data transparently. At all storage locations, you can create subdirectories for this purpose in a directory.

In a subdirectory, in turn, you can create programs and then create program blocks for them.

Note

Restrictions

- Directory names must end in .DIR or .WPD.
- The maximum name length is 28 characters including the extension.
- The maximum path length for nested workpieces, including all supplementary characters, is 100 characters.
- These names are automatically converted to upper-case letters.
  This limitation does not apply for work on USB/network drives.

Procedure



1. Select the "Program manager" operating area.





2. Select the desired storage medium, i.e. a local or USB drive.





3. If you want to create a new directory in the local network, place the cursor on the topmost folder and press the "New" and "Directory" softkeys.
   The "New Directory" window opens.



4. Enter the desired directory name and press the "OK" softkey.

13.4.3 Creating a new workpiece

You can set up various types of files such as main programs, initialization files, tool offsets, etc. in a workpiece.

Note

Workpiece directories

You have the option of nesting tool directories. You must note that the length of the call line is restricted. You will be informed if the maximum number of characters is reached when entering the workpiece name.

Procedure



1. Select the "Program manager" operating area.





2. Select the desired storage location and position the cursor on the folder, in which you would like to create a workpiece.



3. Press the "New" softkey.
   The "New Workpiece" window appears.
4. If necessary, select a template if any are available.



5. Enter the desired workpiece name and press the "OK" softkey.

   The directory type (WPD) is set by default.
   A new folder with the workpiece name will be created.
   The "New G Code Program" window will open.



6. Press the "OK" softkey again if you want to create the program.

   The program will open in the editor.

See also

Storing any new file (Page 336)

13.4.4 Creating a new G code program

You can create G code programs and then render G code blocks for them in a directory/ workpiece.

Procedure



1. Select the "Program manager" operating area.
2. Select the desired storage location and position the cursor on the folder, in which you would like to store the program.



3. Press the "New" softkey.



   The "New G Code Program" window appears.

4. If necessary, select a template if any are available.
5. Select the file type (MPF or SPF).

   If you are in the NC memory and have selected the file "Subprograms" or "Part programs", you can create only one subprogram (SPF) or main program (MPF) respectively.



6. Enter the desired program name and press the "OK" softkey.

   The program type is appropriately specified.

13.4.5 Storing any new file

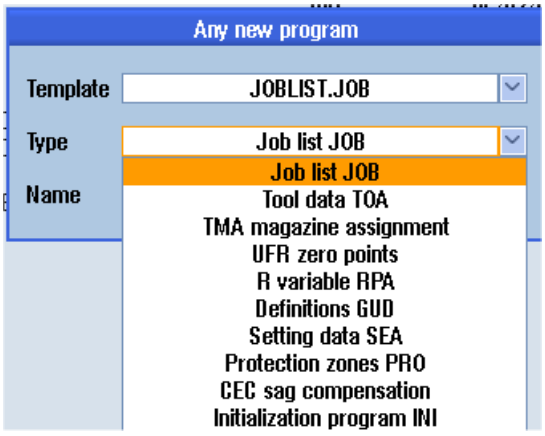
In each directory or subdirectory you can create a file in any format that you specify.

Note

File extensions

In the NC memory, the extension must have 3 characters, and DIR or WPD are not permitted.

In the NC memory, you can create the following file types under a workpiece using the "Any" softkey.



Procedure



1. Select the "Program manager" operating area.

2. Select the desired storage location and position the cursor on the folder in which you would like to create the file.



3. Press the "New" and "Any" softkeys.
   The "Any New Program" window opens.



4. Select a file type from the "Type" selection field (for example, "Definitions GUD") and enter the name of the file to be created when you have selected a workpiece directory in the NC memory.
   The file automatically has the selected file format.
   - OR -

Enter a name and file format for the file to be created (e.g. My_Text.txt).



5. Press the "OK" softkey.

13.4.6 Creating a job list

For every workpiece, you can create a job list for extended workpiece selection.

In the job list, you specify instructions for program selection in different channels.

Syntax

The job list contains the SELECT instructions.

SELECT <program> CH=<channel number> [DISK]

The SELECT instruction selects a program for execution in a specific NC channel. The selected program must be loaded into the working memory of the NC. The DISK parameter enables the selection of external execution (memory card, USB data carrier, network drive).

- <Program>
  Absolute or relative path specification of the program to be selected.
  Examples:
  - //NC/WCS.DIR/SHAFT.WPD/SHAFT1.MPF
  - SHAFT2.MPF
- <Channel number>
  Number of the NC channel in which the program is to be selected.
  Example:
  CH=2
- [DISK]
  Optional parameter for programs that are not in the NC memory and are to be executed "externally".
  Example:
  SELECT //remote/myshare/shaft3.mpf CH=1 DISK

Comment

Comments are identified in the job list by ";" at the start of the line or by round brackets.

Template

You can select a template from Siemens or the machine manufacturer when creating a new job list.

Executing a workpiece

If the "Select" softkey is selected for a workpiece, the syntax of the associated job list is checked and then executed. The cursor can also be placed on the job list for selection.

Procedure



1. Select the "Program Manager" operating area.



2. Press the "NC" softkey, and in directory "Workpieces" place the cursor on the program for which you wish to create a job list.



3. Press the "New" and "Any" softkeys.
   The "Any New Program" window opens.



4. Select entry "Job list JOB" from the "Type" selection field and enter a name and press the "OK" softkey.

13.4.7 Creating a program list

You can also enter programs in a program list that are then selected and executed from the PLC.

The program list may contain up to 100 entries.



Machine manufacturer
Please refer to the machine manufacturer's specifications.

Procedure



1. Select the "Program manager" operating area.





2. Press the menu forward key and the "Program list" softkey.
   The "Prog.-list" window opens.

3. Place the cursor in the desired line (program number).



4. Press the "Select program" softkey.
   The "Programs" window opens. The data tree of the NC memory with workpiece, part program and subprogram directory is displayed.



5. Place the cursor on the desired program and press the "OK" softkey.
   The selected program is inserted in the first line of the list together with its path.
   - OR -

Enter the program name directly in the list.

If you are making entries manually, check that the path is correct (e.g. //NC/WKS.DIR/MEINPROGRAMM.WPD/MEINPROGRAMM.MPF).

//NC and the extension (.MPF) may be added automatically.

With multi-channel machines, you can specify in which channel the program is to be selected.

Delete

6. To remove a program from the list, place the cursor on the appropriate line and press the "Delete" softkey.

- OR -

Delete all

To delete all programs from the program list, press the "Delete all" softkey.

13.5 Creating templates

You can store your own templates to be used for creating part programs and workpieces. These templates provide the basic framework for further editing.

You can use them for any part programs or workpieces you have created.

Storage location for templates

The templates used to create part programs or workpieces are stored in the following directories:

HMI Data/Templates/Manufacturer/Part programs or Workpieces

HMI Data/Templates/User/Part programs or Workpieces

Procedure



1. Select the "Start-up" operating area.



2. Press the "System data" softkey.



3. Position the cursor on the file that you wish to store as a template and press the "Copy" softkey.



4. Select the directory in which you want to store the data - "Part programs" or "Workpieces" - and press the "Paste" softkey.

   Stored templates can be selected when a part program or a workpiece is being created.

13.6 Searching directories and files

You have the possibility of searching in the Program Manager for certain directories and files.

Note

Search with place holders

The following place holders simplify the search:

- "*": Replaces any character string
- "?": Replaces any character

If you use place holders, only directories and files are found that correspond exactly to the search pattern.

Without place holders, directories and files are found that contain the search pattern at an arbitrary position.

Search strategy

The search is made in all of the selected directories and their subdirectories.

If the cursor is positioned on a file, then a search is made from the higher-level directory.

Note

Searching in opened directories

Open the closed directories for a successful search.

Procedure



1. Select the "Program Manager" operating area.





2. Select the storage location in which you wish to perform the search and then press the ">>" and "Search" softkeys.
   The "Find File" window opens.

3. Enter the desired search term in the "Text" field.
   Note: When searching for a file with place holders, enter the complete name with extension (e.g. DRILLING.MPF).
4. When required, activate the "Observe upper and lower case" checkbox.



5. Press the "OK" softkey to start the search.
6. If a corresponding directory or file is found, then it is marked.





7. Press the "Continue search" and "OK" softkeys if the directory or the file does not correspond to the required result.

- OR -



Press the "Cancel" softkey when you want to cancel the search.

13.7 Displaying the program in the Preview.

You can show the content on a program in a preview before you start editing.

Procedure



1. Select the "Program manager" operating area.
2. Select a storage location and place the cursor on the relevant program.
3. Press the ">>" and "Preview window" softkeys.
   The "Preview: ..." window opens.







4. Press the "Preview window" softkey again to close the window.

13.8 Selecting several directories/programs

You can select several files and directories for further processing. When you select a directory, all directories and files located beneath it are also selected.

Note

Selected files

If you have selected individual files in a directory, then this selection is canceled when the directory is closed.

If the complete directory with all of the files included in it are selected, then this selection is kept when closing the directory.

Procedure



1. Select the "Program manager" operating area.
2. Choose the desired storage location and position the cursor on the file or directory from which you would like your selection to start.



3. Press the "Select" softkey.



The softkey is active.

4. Select the required directories/programs with the cursor keys or mouse.



5. Press the "Select" softkey again to deactivate the cursor keys.

Canceling a selection

By reselecting an element, the existing selection is canceled.

Selecting via keys

Key combination	Meaning
	Renders or expands a selection. You can only select individual elements.
	Renders a consecutive selection.
	A previously existing selection is canceled.

Selecting with the mouse

Key combination	Meaning
Left mouse	Click on element: The element is selected. A previously existing selection is canceled.
Left mouse +  Pressed	Expand selection consecutively up to the next click.
Left mouse +  Pressed	Expand selection to individual elements by clicking. An existing selection will expand to include the element you clicked.

13.9 Copying and pasting a directory/program

To create a new directory or program that is similar to an existing program, you can save time by copying the old directory or program and only changing selected programs or program blocks.

The capability of copying and pasting directories and programs can also be used to exchange data with other systems via USB/network drives (e.g. USB FlashDrive).

Copied files or directories can be pasted at a different location.

Note

You can only paste directories on local drives and on USB or network drives.

Note

Write rights

If the current directory is write-protected for the user, then the function is not listed.

Note

When you copy directories, any missing endings are added automatically.

All letters (except accented characters), numbers, and underscores are permitted when assigning names. The names are automatically converted to upper-case letters, and extra dots are converted to underscores.

Example

If the name is not changed when copying, a copy is created automatically:

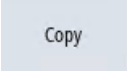
MYPROGRAM.MPF is copied to MYPROGRAM__1.MPF. The next time it is copied, it is changed to MYPROGRAM__2.MPF, etc.

If the files MYPROGRAM.MPF, MYPROGRAM__1.MPF, and MYPROGRAM__3.MPF already exist in a directory, MYPROGRAM__2.MPF is created as the next copy of MYPROGRAM.MPF.

Procedure



1. Select the "Program Manager" operating area.
2. Choose the desired storage location and position the cursor on the file or directory which you would like to copy.



3. Press the "Copy" softkey.
4. Select the directory in which you want to paste your copied directory/ program.



5. Press the "Paste" softkey.

   An appropriate note is displayed if a directory/program with the same name exists in this directory. You are requested to assign a new name, otherwise the directory/program is assigned a name by the system.

   If the name contains illegal characters or is too long, a prompt will appear for you to enter a permissible name.



6. Press the "OK" or "Overwrite all" softkey if you want to overwrite existing directories/programs.



   - OR -



   Press the "No overwriting" softkey if you do not want to overwrite already existing directories/programs.

   - OR -



   Press the "Skip" softkey if the copy operation is to be continued with the next file.

   - OR -



   Enter another name if you want to paste the directory/program under another name and press the "OK" softkey.

Note

Copying files in the same directory

You cannot copy files to the same directory. You must copy the file under a new name.

13.10 Deleting a directory/program

Delete programs or directories from time to time that you are no longer using to maintain a clearer overview of your data management. Back up the data beforehand, if necessary, on an external data medium (e.g. USB FlashDrive) or on a network drive.

Please note that when you delete a directory, all programs, tool data and zero point data and subdirectories that this directory contains are deleted.

Procedure



1. Select the "Program manager" operating area.

2. Choose the desired storage location and position the cursor on the file or directory that you would like to delete.



3. Press the ">>" and "Delete" softkeys.

   A prompt appears as to whether you really want to delete the file or directory.





4. Press the "OK" softkey to delete the program/directory.

   - OR -.

   Press the "Cancel" softkey to cancel the process.



13.11 Changing file and directory properties

Information on directories and files can be displayed in the "Properties for ..." window.

Information on the creation date is displayed near the file's path and name.

You can change names.

Changing access rights

Access rights for execution, writing, listing and reading are displayed in the "Properties" window.

- Execute: Is used for the selection for execution
- Write: Controls the changing and deletion of a file or a directory

For NC files, you have the option to set the access rights from keyswitch 0 to the current access level, to be set separately for each file.

If an access level is higher than the current access level, it cannot be changed.

For external files (e.g. on a local drive), the access rights are displayed to you only if settings have been made for these files by the machine manufacturer. They cannot be changed via the "Properties" window.

Settings for the access rights to directories and files

Via a configuration file and MD 51050, access rights of the directories and file types of the NC and user memory (local drive) can be changed and pre-assigned.

Further information: SINUMERIK Operate Commissioning Manual

Procedure



1. Select the program manager.





2. Choose the desired storage location and position the cursor on the file or directory whose properties you want to display or change.



3. Press the ">>" and "Properties" softkeys.
   The "Properties from ..." window appears.



4. Enter any necessary changes.
   Note: You can save changes via the user interface in the NC memory.
5. 

   Press the "OK" softkey to save the changes.

13.12 Set up drives

13.12.1 Overview

Up to 21 connections to so-called logical drives (data storage mediums) can be configured. These drives can be accessed in the "Program manager" and "Commissioning" operating areas.

The following logical drives can be set up:

- USB interface
- Network drives
- SD card
- SD card of the NCU, only for SINUMERIK Operate in the NCU
- Local hard disk of the PCU, only for SINUMERIK Operate on PCU



Software option

In order to use the SD card as a data storage medium, you require the option "Additional HMI user memory on NCU memory card" (not for SINUMERIK Operate on PCU/PC).

Note

The USB interfaces of the NCU are not available for SINUMERIK Operate and can therefore not be configured.

Note

USB flash drives with multiple partitions are supported with the restriction that only individual partitions can be linked with the softkeys, but USB flash drives as a whole cannot be managed. It is therefore **not** recommended to configure the entire USB flash drive as a softkey since the file system cannot manage the individual files on the USB flash drive as a whole, but only individually on the partitions.

13.12.2 Setting up drives

The "Set Up Drives" window is available in the "Setup" operating area for configuring the softkeys in the Program Manager.

Note

Reserved softkeys

Softkeys 4, 7 and 16 are not available to be freely configured.



Machine manufacturer
Refer to the machine manufacturer's specifications.

File

The created configuration data is stored in the "logdrive.ini" file. This file is located in the /user/sinumerik/hmi/cfg directory.

General information

Entry		Meaning
Drives 1 - 24		
Type	No drive	No drive defined
	NC program memory	Access to the NC memory
	USB local	Access to the USB interface of the active operator unit
	USB global	All of the TCUs in the plant network can access the USB memory medium.
	NW Windows	Network drive in Windows systems.
	NW Linux	Network drive in Linux systems.
	Local drive	Local drive. Hard disk or user memory on the memory card.
	FTP	Access to an external FTP server. The drive cannot be used as global part program memory.
	User cycles	Access to the user cycle directory of the memory card
	Manufacturer cycles	Access to the manufacturer cycle directory of the memory card
	Drive Windows	Access to a local PCU/PC directory.

Specifications for USB

Entry		Meaning
Device		Names of the TCU to which the USB storage medium is connected, e.g. tcu1. The NCU must already know the TCU name.
Connection	Front	USB interface that is located at the front of the operator panel.
	X203/X204	USB interface X203/X204 that is located at the rear of the operator panel.
	X61/X62	For SIMATIC Thin Client, the USB interfaces are X61 and X62.
	X212/X213	TCU20.2/20.3
	X20	OP 08T
	X60.P1/P2/P3/P4	PCU
Symbolic		Symbolic drive name.
Additional parameters under Details		
Partition		Partition number on the USB storage medium, e.g. 1 or all. If a USB hub is being used, then specify the USB port of the hub.
USB path		Path to the USB hub. **Note:** This value is not currently evaluated.

Specifications for local drives

Entry		Meaning
Symbolic		Symbolic drive name. Assignment of the names under Details
Additional parameters under Details		
Use drive as:	LOCAL_DRIVE	The activation of the checkbox assigns the symbolic name to the drive. If an assignment exists already for the drive, no change can be made. All checkboxes are active as preassignment.
	SD_CARD	
	SYS_DRIVE	

Specifications for network drives

Entry		Meaning
Computer name		Logical name of the server or the IP address.
Release name	Only for network drives in Windows systems.	Name, under which the network drive was released
Path		Start directory. The path is specified relative to the released directory.

Entry		Meaning
User name Password		Enter the user name and the corresponding password for which the directory is enabled on the server. The password is displayed in encoded form as string of "*" characters and is stored in the "logdrive.ini" file.
Symbolic		Symbolic drive name. Maximum 12 characters can be entered (letters, digits, underscore). The names NC, GDIR and FTP are reserved. They are also used to label the softkey if a softkey text is not specified.

Specifications for FTP

Entry		Meaning
Computer name		Logical name of the FTP server or the IP address.
Path		Start directory on the FTP server. The path is specified relative to the home directory.
User name Password		User names and the associated password for login to the FTP server. The password is displayed in encoded form as string of "*" characters and is stored in the "logdrive.ini" file.
Additional parameters under Details		
Port		Interface for the FTP connection. The default port is 21.
Disconnect		After a disconnect timeout, the FTP connection is disconnected. The timeout can be between 1 and 150 s. 10 s is the default setting.

Additional specifications when using the "Execution from external storage (EES)" function

Machine manufacturer
Refer to the machine manufacturer's specifications.

Entry		Meaning
Enable drive	Only for "Drive Windows (PCU)" type	The drive is enabled in the network. A user name is required. The checkbox must be activated if the local drive serves as global part program memory.
Global part program memory	Only for local drives, network drives and global USB drives	The checkbox indicates that all system nodes have access to the configured logical drive. The nodes can directly execute part programs from the drive. The setting can only be changed under Details.
Use this drive for EES program execution	Only for USB drives	Allows a local USB storage medium to be used to execute programs using EES.
Additional parameters under Details		
Windows user name Windows password	Only for USB drives, local drives and local directories	User name and the associated password for release of the configured drive. The specifications from the "Global Settings" window are used as default setting.
Global part program memory	Only for local drives, network drives and global USB drives	The checkbox defines whether all system nodes have access to the configured logical drive. Only one drive can be selected as global part program memory (GDIR). If another drive has already been defined as GDIR and the checkbox is activated, the original setting is deleted.

Specifications for the configured softkey

Entry		Meaning
Access level		Assign access rights to the connections: From access level 7 (keyswitch position 0) to access level 1 (manufacturer). The particular assigned access level applies to all operating areas.
Softkey text		Two lines are available as labeling text for the softkey. %n is accepted as a line separator. If the first line is too long, then a line break is automatically inserted. If a space is present, it is taken as a line separator. For language-dependent softkey texts, the text ID is entered, which is used to search in the text file. If nothing is specified in the entry field, then the symbolic drive name is used as softkey text.

Entry		Meaning
Softkey icon	No icon	No icon is displayed on the softkey.
	sk_usb_front.png 	File names of the icon displayed on the softkey.
	sk_local_drive.png 	
	sk_net-work_drive_ftp.png 	
Text file	slpmdialog	File for language-dependent softkey text. If nothing is specified in the input fields, the text appears on the softkey as was specified in the "Softkey text" input field.
Text context	SlPmDialog	

Procedure

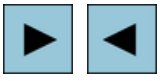


1. Select the "Setup" operating area.





2. Press the "HMI" and "Log. drive" softkeys.
   The "Set Up Drives" window opens.



3. Select the softkey that you want to configure.



4. To configure softkeys 9 to 16 or softkeys 17 to 24, click the ">> level" softkey.



5. To allow entry fields to be edited, press the "Change" softkey.

6. Select the data for the corresponding drive or enter the necessary data.



7. Press the "Details" softkey if you want to enter additional parameters.
   Press the "Details" softkey to return to the "Set Up Drives" window.



8. Press softkey "OK".
   The entries are checked.



A window with the appropriate message opens if the data is incomplete or incorrect. Acknowledge the message with "OK" softkey.



If you press the "Cancel" softkey, then all of the data that has not been activated is rejected.

9. Restart the control in order to activate the configuration and to obtain the softkeys in the "Program Manager" operating area.

Note

If a drive is presently being used, e.g. a program is being executed or a file is being copied, do not create any of the drives as new, independent of which drive is used.

Before setting up the drives, wait until the operation has been completed. Otherwise, a drive can become unusable, which can only be resolved by restarting the control system.

Entering the default settings for drive release

Note

This function is available only on Windows systems when the "Execution from external storage (EES)" software option is activated.



1. Select the "Setup" operating area.





2. Press the "HMI" and "Log. drive" softkeys.
   The "Set Up Drives" window opens.



3. Press the "Glob. settings" softkey.

4. Enter the user name and the associated password for the configured drives to be released.



5. Press softkey "OK".
   The specifications are transferred as default setting for the Windows release.



If you press the "Cancel" softkey, then all of the data that has not been activated is rejected.

13.13 Viewing PDF documents

You have the option of displaying HTML documents, as well as PDFs, on all drives of the program manager via the data tree of the system data.

Note

A preview of the documents is only possible for PDFs.

Procedure





1. In the "Program manager" operating area, select the desired storage medium.

   - OR -

   Select the desired storage location in the "Commissioning" operating area in the data tree of the "System data".





2. Position the cursor on the PDF or the HTML file that you want to display, and press the "Open" softkey.

   The selected file is displayed on the screen.

   The storage path of the document is displayed in the status bar. The current page as well as the total number of pages of the document are displayed.





3. Press the "Zoom +" or "Zoom -" softkey to enlarge of reduce the size of the display.

Navigation and search for specific texts



1. Press the "Search" softkey.

   A new vertical softkey bar is displayed.



2. Press the "Go to start" softkey to navigate to the first page of the document.



3. Press the "Go to end" softkey to navigate to the last page of the document.



4. Press the "Go to page" softkey to navigate to a specific page of the document.



5. Press the "Search" softkey if you want to search for specific texts in the PDF.



6. Enter the required term in the search form and confirm with "OK".
   The cursor is positioned on the first entry that corresponds to the search term.



7. Press the "Continue search" softkey if the text that is found is not the specific text that you are looking for.



8. Press the "Back" softkey to return to the higher-level softkey bar.

Note

If you change the language when viewing a PDF document, the PDF document is reloaded in the respective language. If no PDF document is available for the language set, the English PDF document is displayed.

The position in the PDF document is retained beyond language switchover across sessions if the PDF document contains bookmarks.

Changing the display



1. Press the "View" softkey to change the representation of the PDF.
   A new vertical softkey bar is displayed.



2. Press the "Zoom page width" softkey to display the document in full width on the screen.

   - OR -



   Press the "Zoom page length" softkey to display the document with full height on the screen.

   - OR -



   Press the "Rotate left" softkey to rotate the document through 90 degrees to the left.

   - OR -



   Press the "Rotate right" softkey to rotate the document through 90 degrees to the right.

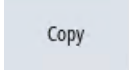


3. Press the "Back" softkey to return to the higher-level softkey bar.

Copying text



1. Press the "Mark" softkey.
   The "Copy" softkey can be used.
2. Select the required text either by touching or with the mouse.



3. Press the "Copy" softkey.
   "The marked text is available for insertion in the editor"



4. Press the "Mark" softkey again to shift the displayed section by touching.
   The "Copy" softkey cannot be used.

Closing the PDF



1. Press the "Close" softkey to close the PDF.

13.14 EXTCALL

The EXTCALL command can be used to access files on a local drive, USB data storage medium or network drives from a part program.

The programmer can set the source directory with the setting data SD $SC42700 EXT_PROG_PATH and then specify the file name of the subprogram to be loaded with the EXTCALL command.

Supplementary conditions

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

- You can only call files with the MPF or SPF extension via EXTCALL from a network drive.
- The files and paths must comply with the NCK naming conventions (max. 25 characters for the name, 3 characters for the identifier).
- A program is found on a network drive with the EXTCALL command if
  - with SD $SC42700 EXT_PROG_PATH the search path refers to the network drive or a directory contained on the network drive. The program must be stored directly on that level, no subdirectories are searched.
  - without SD $SC42700 the correct location of the program is specified in the EXTCALL call itself by means of a fully qualified path that can also point to a subdirectory of the network drive.
- For programs that were generated on external storage media (Windows system) observe the upper- and lower-case syntax.

Note

Maximum path length for EXTCALL

The path length must not exceed 112 characters. The path comprises the contents of the setting data (SD $SC42700) and the path data for EXTCALL call from the part program.

Note

Path data

Use the IP address of your computer in the path data for the EXTCALL command. The computer name or DNS cannot be used for this purpose.

Examples of EXTCALL calls

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

- Call of USB drive on TCU (USB storage device on interface X203), if SD42700 is empty: e.g. EXTCALL "//TCU/TCU1 /X203 ,1/TEST.SPF"
  - OR -
  Call of USB drive on TCU (USB storage device on interface X203), if SD42700 "//TCU/TCU1 /X203 ,1" contains: EXTCALL "TEST.SPF"
- Call of the USB front connection (USB flash drive), if SD $SC 42700 is empty: e.g. EXTCALL "//ACTTCU/FRONT,1/TEST.SPF"
  - OR -
  Call of USB front connection (USB flash drive), if SD42700 "//ACTTCU/FRONT,1" contains: EXTCALL "TEST.SPF"
- Call of network drive, if SD42700 is empty: e.g. EXTCALL "//IP address/shared drive/TEST.SPF"
  - OR -
  Network drive is called if SD $SC42700 "//IP address/shared drive" contains: EXTCALL "TEST.SPF"
- Use of the HMI user memory (local drive):
  - On the local drive, you have created the directories part programs (mpf.dir), subprograms (spf.dir) and workpieces (wks.dir) with the respective workpiece directories (.wpd): SD42700 is empty: EXTCALL "TEST.SPF"
    The same search sequence is used on the memory card as in the NCK part program memory.
  - On the local drive, you have created your own directory (e.g. my.dir):
    Specification of the complete path: e.g. EXTCALL "/user/sinumerik/data/prog/my.dir/TEST.SPF"
    A search is performed for the specified file.

Note

Abbreviations for local drive, memory card and USB front connection

As abbreviation for the local drive, the memory card and the USB front connection, you can use LOCAL_DRIVE:, SD_CARD: and USB: (e.g. EXTCALL "LOCAL_DRIVE:/spf.dir/TEST.SPF").

Alternatively, you can also use the abbreviations SD_Card and LOCAL_DRIVE.



Software options

To display the "Local drive" softkey, you require option "Additional HMI user memory on NCU memory card".

NOTICE
Possible interruption when executing from USB flash drive Direct execution from a USB flash drive is not recommended. There is no protection against contact problems, falling out, breakage through knocking or unintentional removal of the USB flash drive during operation. Disconnecting it during operation will result in immediate stopping of the machining and, thus, to the workpiece being damaged.



Machine manufacturer

Processing EXTCALL calls can be enabled and disabled.

Observe the information provided by the machine OEM.

13.15 Execution from external memory (EES)

The "Execution from external storage" function allows you to directly execute any size of part program from an appropriately configured drive. The behavior is the same as that for execution from the NC part program memory without the restrictions that apply to "EXTCALL".



Software option

You require the "CNC user memory extended" software option in order to use this function in the user memory (100 MB) of the memory card.



Software option

In order to use this function without restrictions, e.g. for a network drive or a USB drive, you require the "Execution from external storage (EES)" software option.

Note

Not possible to teach-in program

When an EES program has been selected, it is not possible to teach-in programs.



Machine manufacturer

Please observe the information provided by the machine manufacturer.

You have the option of processing the G code programs saved on the configured external drives as usual in the editor.

When executing the G code programs, you obtain a current block display, as usual. You can edit the programs directly in the Reset state.

In addition to the current block display, you can also show a basic block display. You can make corrections with the "Program correction" function, as usual.

13.16 Backing up data

13.16.1 Generating an archive in the Program Manager

You have the option of archiving individual files from the NC memory and the local drive.

Archive formats

You can save your archive in the binary format.

Save target

The archive folder of the system data in the "Startup" operating area as well as USB and network drives are available as save target.

Procedure



1. Select the "Program Manager" operating area.



2. Select the storage location for the file/files to be archived.

3. In the directories, select the required file from which you want to create an archive.

   - OR -

   If you want to back up several files or directories, press the "Select" softkey.

   Make the selection using the cursor keys or mouse.





4. Press the ">>" and "Archive" softkeys.





5. Press the "Generate archive" softkey.

   The "Generate Archive: Select archiving" window opens.



6. Position the cursor to the required storage location, press the "Search" softkey, enter the required search term in the search dialog and press the "OK" softkey if you wish to search for a specific directory or subdirectory.

   Note: The place holders "*" (for any character string) and "?" (for any character) make it easier for you to perform a search.



   - OR -



Select the required storage location, press the "New directory" softkey, enter the required name in the "New directory" window and press the "OK" softkey to create a directory.



7. Press "OK".
   The "Generate Archive: Name" window opens.



9. Select the format, enter the required name and press the "OK" softkey.
   A message informs you if archiving was successful.



13.16.2 Generating an archive via the system data

If you only want to backup specific data, then you can select the desired files directly from the data tree and generate an archive.

Archive formats

You can save your archive in the binary format.

You can display the content of the selected files (XML, ini, hsp, syf files, programs) using a preview.

You can display information about the file, such as path, name, date of creation and change, in a Properties window.

Requirement

The access rights depend on the relevant areas and range from protection level 7 (keyswitch position 0) to protection level 2 (password: Service).

Storage locations

- SD Card under
  /**user**/sinumerik/data/archive or
  /**oem**/sinumerik/data/archive
- All configured logical drives (USB, network drives)



Software option
In order to save archives on the SD Card in the user area, you require the "Additional HMI user memory on NCU memory card" option.

NOTICE
Possible data loss when using USB flash drives
USB flash drives are not suitable as persistent memory media.

Procedure



1. Select the "Setup" operating area.



2. Press the "System data" softkey.
   The data tree opens.
3. In the data tree, select the required files from which you want to generate an archive.
   - OR -
   If you want to back up several files or directories, press the "Select" softkey.
   Make the selection using the cursor keys or mouse.





4. If you press the ">>" softkey, further softkeys are displayed on the vertical bar.



5. Press the "Preview window" softkey.
   The contents of the selected file are displayed in a small window.
   Press the "Preview window" softkey again to close the window.



6. Press the "Properties" softkey.
   Information about the selected file is displayed in a small window.
   Press the "OK" softkey to close the window.





7. Press the "Search" softkey.
   Enter the required search term in the search dialog and press the "OK" softkey if you wish to search for a specific directory or subdirectory.
   Note: The place holders "*" (for any character string) and "?" (for any character) make it easier for you to perform a search.







8. Press the "Archive" and "Generate archive" softkeys.
   The "Generate Archive: Select Storage Location" window opens.
   The "Archive" folder with the subfolders "User" and "Manufacturer" as well as the storage media (e.g. USB) are displayed.



9. Select the required location for archiving and press the "New directory" softkey to create a suitable subdirectory.
   The "New Directory" window opens.



10. Enter the required name and press the "OK" softkey.
    The directory is created below the selected folder.



11. Press the "OK" softkey.
    The "Generate Archive: Name" window opens.



12. Select the format, enter the required name and press the "OK" softkey to archive the file/files.
    A message informs you if archiving was successful.



13. Press the "OK" softkey to confirm the message and end the archiving operation.
    An archive file is created in the selected directory.

13.16.3 Reading in an archive in the Program Manager

In the "Program Manager" operating area, you have the option of reading in archives from the archive folder of the system data as well as from configured USB and network drives.



Software option
You require the option "Additional HMI user memory on NCU memory card" in order to read in user archives in the "Program Manager" operating area.

Procedure



1. Select the "Program Manager" operating area.





2. Press the "Archive" and "Read in archive" softkeys.
    The "Read in archive: Select archive" window opens.

3. Select the archive storage location and position the cursor on the required archive.
    Note: When the option is not set, the folder for user archives is only displayed if the folder contains at least one archive.
    - OR -



    Press the "Search" softkey and in the search dialog, enter the name of the archive file with file extension if you wish to search for a specific archive and press the "OK" softkey.





4. Press the "OK" or "Overwrite all" softkey to overwrite existing files.

...



- OR -



Press the "Do not overwrite" softkey if you do not want to overwrite already existing files.

- OR -



Press the "Skip" softkey if the read-in operation is to be continued with the next file.

The "Read In Archive" window opens and a progress indicator appears for the read-in process.

You will then obtain a "Read error log for archive" in which the skipped or overwritten files are listed.



5. Press the "Cancel" softkey to cancel the read-in process.

13.16.4 Read in archive from system data

If you want to read in a specific archive, you can select this directly from the data tree.

Procedure



1. Select the "Startup" operating area.



2. Press the "System data" softkey.

3. In the data tree below the "Archive" directory, in the "User" folder, select the file that you wish to read in.



4. Press the "Read in" softkey.



5. Press the "OK" or "Overwrite all" softkey to overwrite existing files.

...



- OR -



Press the "Do not overwrite" softkey if you do not want to overwrite already existing files.

- OR -



Press the "Skip" softkey if the read-in operation is to be continued with the next file.

The "Read In Archive" window opens and a progress message box appears for the read-in process.

You will then obtain a "Read error log for archive" in which the skipped or overwritten files are listed.

6. 

Press the "Cancel" softkey to cancel the read-in process.

13.17 Setup data

Apart from programs, you can also save tool data and zero point settings.

You can use this option, for example, to backup the required tools and zero point data for a specific G code program. If you want to execute this program at a later point in time, you will then have quick access to the relevant settings.

Even tool data that you have measured on an external tool setting station can be copied easily into the tool management system using this option.

Note

Backing up setup data from part programs

Setup data from part programs can only be backed up if they have been saved in the "Workpieces" directory.

For part programs, which are located in the "Part programs" directory, "Save setup data" is not listed.

Backing up data

Data	
Tool data	• No • Complete tool list
Magazine assignment	• Yes • No
Zero points	• No The selection box "Basis zero point" is hidden • All
Basic zero points	• No • Yes
Directory	The directory is displayed, in which the selected program is located.
File name	Here you have the option of changing the suggested file names.

Note

Magazine assignment

You can only read out the magazine assignments if your system provides support for loading and unloading tool data to and from the magazine.

Procedure



1. Select the "Program Manager" operating area.



2. Position the cursor on the program whose tool and zero point data you wish to back up.

...





3. Press the ">>" and "Archive" softkeys.





4. Press the "Setup data" softkey.
   The "Backup setup data" window opens.
5. Select the data you want to back up.
6. When required, change the specified name of the originally selected program here in the "File name" field.



7. Press the "OK" softkey.
   The setup data will be set up in the same directory in which the selected program is stored.
   The file is automatically saved as INI file.

Note

Program selection

If a main program as well as an INI file with the same name are in a directory, when selecting the main program, initially, the INI file is automatically started. In this way, unwanted tool data can be changed.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

13.17.1 Reading-in set-up data

When reading-in, select which of the backed-up data you require:

- Tool data
- Magazine assignment
- Zero points
- Basic zero point

Tool data

Depending on which data you have selected, the system behaves as follows:

- Complete tool list
  First, all tool management data are deleted and then the saved data are imported.
- All tool data used in the program
  If at least one of the tools to be read in already exists in the tool management system, you can choose between the following options.



Select the "Replace all" softkey to import all tool data. Any existing tools will now be overwritten without a warning prompt.

- OR -



Press the "Do not overwrite" softkey if existing tools must not be overwritten.

Already existing tools are skipped, without you receiving any queries.

- OR -



Press the "Skip" softkey if already existing tools are not to be overwritten.

For an already existing tool, you receive a query.

Selecting loading point

For a magazine, if more than one loading point was set-up, using the "Select loading point" softkey, you have the option of opening a window in which you can assign a loading point to a magazine.

Procedure



1. Select the "Program Manager" operating area.



2. Position the cursor on the file with the backed-up tool and zero point data (*.INI) that you wish to re-import.





3. Press the <Cursor right> key

- OR -
Double-click the file.
The "Read-in setup data" window opens.



4. Select the data (e.g. magazine assignment) that you wish to read-in.



5. Press the "OK" softkey.

13.18 Recording tools and determining the demand

13.18.1 Overview

The "Record tools and determine tool demand" function allows you to record all the tools you need when executing and simulating part programs. In this way you can determine the tool demand.



Software options

In order to use the "Record tools and determine demand" function, you need the "Determine tool demand" option.



Machine manufacturer

Please observe the information provided by the machine manufacturer.

The "Record tools and determine demand" function can be helpful in the following work steps on single-channel and multi-channel machines:

- Preparation of a new workpiece while the old workpiece is still being machined:
  - All new tools must be loaded additionally
- Change of the workpiece at the machine:
  - All old tools can be unloaded
  - All new tools must be loaded

You activate the recording of tools in the settings for automatic operation. Recording takes place during processing.

You can also activate recording for the simulation.

The recorded tool data are stored in a TTD file (Tool Time Data). The TTD file is always located with the associated part program.

Note

The recorded tool data of a tool path is only valid if the program has been run through completely. Otherwise the data will not be saved and the previous TTD file will be retained.

13.18.2 Opening tool data

Introduction

The recorded tool data are stored in a TTD file (Tool Time Data). The TTD file is always located with the associated part program and contains the following information:

- Tool data
- Magazine assignment
- Operating time, idle time, tool number, D number, ID per block

You can determine the tool requirement for one or several TTD files.

Note

You can only open TTD files that contain the tool for the TOA area of the active channel.

Opening a TTD file



1. Select the "Program Manager" operating area.





2. Under part programs, open the corresponding file with the extension *. TTD.



3. Acknowledge with "OK".

The "Tools for program" window opens.
The file contents are displayed in the form of a tool list.

Note

If the TTD file is older than the associated part program, a message informs you of this.

Note

To open several TTD files at once, press softkey "Add TTD file", select the required file in window "Load TTD file" and acknowledge with "OK".
Alternatively, select several TTD files in the directory and then click on "Open".

All tools from the selected TTD files are displayed in a tool list.

13.18.3 Checking the loading

When you click the "Check loading" softkey, the "Check loading" window opens.

You can select the following options:

- Only check loading
- Check loading and tool life

Option "Only check loading"

If you have selected option "Only check loading", then the view in window "Check loading" is subdivided into the following sections:

- Missing tools: tool not available
- Tools still to be loaded: tool available but not loaded
- Loaded tools: tools available and loaded
- Tools not required: tools available and loaded, but not required

Note

The check is performed for all opened TTD files.

You can adjust all tool data displayed in the "Check loading" window as required, load and unload tools individually or through multiple selection, enter the tool lengths for newly measured tools directly.

The list is updated after each action.

Option "Check loading and tool life"

If you have selected option "Check loading and tool life", then you can determine how many tools are required in order to machine the required number of workpieces.

If you have selected option "Check loading and tool life", then you can determine how many tools are required in order to machine the required number of workpieces. For each TTD file,

you can enter the number of parts to be machined in field "Number of parts to be machined". The tool requirement is determined and displayed corresponding to the number of parts to be machined.

Note

If you enter an excessively high number of parts to be machined, an error message indicates that too many tools were determined, and the number of parts to be machined must be reduced.

13.19 Backing up parameters

In addition to the programs, you can also save R-parameters and global user variables.

You can use this option, for example, to back up the required arithmetic parameters and user variables for a specific program. If you want to execute this program at a later point in time, you will then have quick access to the relevant data.

Note

Backing up parameters from part programs

Parameters from part programs can only be backed up if they have been saved in the "Workpieces" directory.

For part programs that are located in the "Part programs" or "Subprograms" directory, "Save parameters" is not listed.

Backing up data

Which data is offered for backup depends on the machine configuration:

Data	
R-parameters	• No • Yes - all channel-specific arithmetic parameters
Global R-parameters	• No • Yes - all global arithmetic parameters
UGUD parameters	• No • Yes - all channel-specific variables of the user
Global UGUD parameters	• No • Yes - all global variables of the user
MGUD parameters	• No • Yes - all channel-specific variables of the machine manufacturer
Global MGUD parameters	• No • Yes - all global variables of the machine manufacturer
Directory	The directory is displayed, in which the selected program is located.
File name	Here you have the option of changing the suggested file names.

For multi-channel machines, the parameters of the active channel are always backed up.

Job lists

If you select Back up parameters for a job list, the parameters of all programs that it contains are backed up.

The name of the job list does not agree with the names of the programs it contains. To nevertheless permit the unique assignment of the parameter files, they are always assigned the same name as the associated program. You cannot change these file names.

Procedure



1. Select the "Program Manager" operating area.



2. Select the drive on which the program is saved.

...



3. Position the cursor on the program whose parameters you want to back up.



4. Press the ">>" and "Archive" softkeys.





5. Press the "Save parameters" softkey.
   The "Save parameters" window appears.
6. Select the data you want to back up.



7. Press the <CHANNEL> key or click on the channel display if you want to change the active channel.

- OR -



8. Change the specified name of the originally selected program in the "File name" field as required.



9. Press the "OK" softkey.
   The parameters are saved in the same directory in which the selected program is stored.
   The R-parameters (*.RPA) and the user variables (*.GUD) are saved in separate files.

Note

Program selection

If a directory contains a main program as well as an RPA file or a GUD file with the same name, initially these files are automatically started when the main program is selected. Tool data or parameters may accidentally be changed as a result.

Note

The "Save parameters" softkey is only available if the machine data MD11280 $MN_WPD_INI_MODE is set to 1.



Machine manufacturer

Please observe the information provided by the machine manufacturer.

13.20 RS-232-C

13.20.1 Reading-in and reading-out archives via a serial interface

You have the option of reading out and reading in archives in the "Program manager" operating area as well as in the "Startup" operating area via the V24 serial interface.

Availability of the V24 serial interface

If you want to change the availability of the V24 interface, you can adjust the following parameters in file "slpmconfig.ini":

Parameters	Description	
`[V24]`	Describes the section in which the relevant setting parameters are located.	
`useV24`	Setting for the availability of the V24 serial interface	
	`= true`	Interface and softkeys are available (default)
	`= false`	Interface and softkeys are not available

Storage of file "slpmconfig.ini"

The template of the file "slpmconfig.ini" for SINUMERIK Operate is stored in the following directory:

<Installation path>/siemens/sinumerik/hmi/template/cfg

Copy the file to one of the following directories:

<Installation path>/**user**/sinumerik/hmi/cfg

<Installation path>/**oem**/sinumerik/hmi/cfg

Note

If you want to achieve a better overview of the changes you have made yourself, simply delete the unchanged parameters from the file copy "slpmconfig.ini".

Reading-out archives

The files to be sent (directories, individual files) are zipped in an archive (*.arc). If you send an archive (*.arc), this is sent directly without being additionally zipped. If you have selected an archive (*.arc) together with an additional file (e.g. directory), then these are zipped into a new archive and are then sent.

Reading-in archives

Use interface V24 if you want to read in archives. They are transferred and then subsequently unzipped.

Note

Reading in commissioning archives

When you read in a commissioning archive via the V24 interface, then this is immediately activated.

13.20.2 Setting V24 in the program manager

V24 setting	Meaning
Protocol	The following protocols are supported for transfer via the V24 interface: • RTS/CTS (default setting) • Xon/Xoff
Transfer	Data transfer using a secured protocol (ZMODEM protocol): • Normal (default setting) • secure For the selected interface, secure data transfer is set in conjunction with handshake RTS/CTS.
Baud rate	Transfer rate: up to 115 kbaud data transfer rate. The baud rate that can be used depends on the connected device, the cable length and the general electrical conditions. • 110 • .... • 19200 (default) • ... • 115200
Archive format	• Binary format (PC format)
V24 settings (details)	
Interface	• COM1
Parity	Parity bits are used for error detection: The parity bits are added to the coded characters to make the number of positions set to "1" an uneven number (uneven parity) or to an even number (even parity). • None (default setting) • Odd • Even
Stop bits	Number of stop bits for asynchronous data transfer. • 1 (default) • 2

V24 setting	Meaning
Data bits	Number of data bits for asynchronous data transfer. • 5 bits • ... • 8 bits (default setting)
XON (hex)	Only with protocol: Xon/Xoff
XOFF (hex)	Only with protocol: Xon/Xoff
Wait for XON for start receive V24	Only with protocol: Xon/Xoff
Time monitoring (sec)	Time monitoring For data transfer problems or at the end of data transfer (without end of data transfer character) data transfer is interrupted after the specified number of seconds. The time monitoring is controlled by a time generator (clock) that is started with the first character and is reset with each transferred character. The time monitoring can be set (seconds).

Procedure



1. Select the "Program Manager" operating area.





2. Press softkey "NC" or "Local Drive".





3. Press the ">>" and "Archive" softkeys.



4. Select the "V24 settings" softkey.
   Window "Interface: V24" is opened.
5. The interface settings are displayed.



6. Press the "Details" softkey if you wish to view and process additional settings for the interface.

Teaching in a program

14

14.1 Overview

The "Teach in" function can be used to edit programs in the "AUTO" and "MDA" modes. You can create and modify simple traversing blocks.

You traverse the axes manually to specific positions in order to implement simple machining sequences and make them reproducible. The positions you approach are applied.

In the "AUTO" teach-in mode, the selected program is "taught".

In the "MDA" teach-in mode, you teach to the MDA buffer.

External programs, which may have been generated offline, can therefore be adapted and modified according to your specific requirements.

Note

Not possible to teach-in program

When an EES program has been selected, it is not possible to teach-in programs.

General sequence

1. Activate teach-in mode.
2. Insert a block.
   To do this, position the cursor at the desired point in the program and insert an empty line. Press the relevant softkey "Teach position", "Rapid traverse G01", "Straight line G1", or "Circle interpolation position CIP" and "Circle end position CIP".
   - OR -
3. Change an existing block.
   To do this, mark the desired program block, and press the corresponding softkey "Teach in position ", "Rapid traverse G01", "Straight line G1", or "Circle interpolation point CIP" and "Circle end point CIP".
   You can only overwrite a block with a block of the same type.
4. Traverse the axes.
5. Press the "Accept" softkey to teach-in the modified or newly created program block.

Note

Teach in multiple blocks

All defined axes are "taught in" in the first teach-in block. In all additional teach-in blocks, only axes modified by axis traversing or manual input are "taught in".

If you exit teach-in mode, this sequence begins again.

Note

Selection of axes and parameters for teach-in

You can select the axes to be included in the teach-in block in the "Settings" window.

You also specify here whether motion and transition parameters are offered for teach-in.

Operating mode or operating area switchover

If you switch to another operating mode or operating area while in teach-in mode, the position changes will be canceled and teach-in mode will be cleared.

14.2 Select teach in mode

Change to Teach in mode to adapt the current program.

Requirement

"AUTO" mode: The program to be edited is selected.

"MDI" mode The program to be edited is loaded into the MDI buffer.

Procedure



1. Select the "Machine" operating area.



2. Press the <AUTO> or <MDA> key.





3. Press the <TEACH IN> key.



4. Press the "Teach prog." softkey.

14.3 Processing a program

14.3.1 Inserting a block

The cursor must be positioned on an empty line.

The windows for pasting program blocks contain input and output fields for the actual values in the WCS. Depending on the default setting, selection fields with parameters for motion behavior and motion transition are available.

When first selected, the input fields are empty, unless axes had already been traversed before the window was selected.

All data from the input/output fields are transferred to the program with the "Accept" softkey.

Procedure

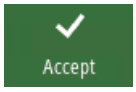


1. Teach-in mode is active.
2. Position the cursor at the desired point in the program.
   If an empty row is not available, insert one.
3. Press the softkeys "Rap. tra. G0", "Straight line G1", or "Circ. interm. pos. CIP" and "Circ. end pos. CIP".





   The relevant windows with the input fields are displayed.
4. Traverse the axes to the relevant position.
5. Press the "Accept" softkey.
   A new program block will be inserted at the cursor position.



   - OR -
   Press the "Cancel" softkey to cancel your input.



14.3.2 Editing a block

You can only overwrite a program block with a teach-in block of the same type.

The axis values displayed in the relevant window are actual values, not the values to be overwritten in the block.

Note

If you wish to change any variable in a block in the program block window other than the position and its parameters, then we recommend alphanumerical input.

Procedure



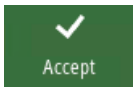
1. Teach-in mode is active.

2. Select the program block to be edited.





3. Press the relevant softkey "Teach position, "Rap. tra. G0", "Straight line G1", or "Circ. interm. pos." CIP", and "Circ. end pos. CIP".
   The relevant windows with the input fields are displayed.



4. Traverse the axes to the desired position and press the "Accept" softkey.
   The program block is taught with the modified values.

   - OR -



   Press the "Cancel" softkey to cancel the changes.

14.3.3 Selecting a block

You have the option of setting the interrupt pointer to the current cursor position. The next time the program is started, processing will resume from this point.

With teach-in, you can also change program areas that have already been executed. This automatically disables program processing.

You must press reset or select a block to resume the program.

Procedure



1. Teach-in mode is active.

2. Place the cursor on the desired program block.



3. Press the "Block selection" softkey.

14.3.4 Deleting a block

In teach-in mode, you can delete both a teach-in block and a program block entirely.

Procedure



1. Teach-in mode is active.

2. Select the program block to be deleted.



3. Press the ">>" and "Delete block" softkeys.
   The program block on which the cursor is positioned is deleted.



14.4 Teach sets

Teach in position

You traverse the axes and write the current actual values directly into a positioning block.

Teach-in rapid traverse G0

You traverse the axes and teach-in a rapid traverse block with the approached positions.

Teach in straight G1

You traverse the axes and teach-in a machining block (G1) with the approached positions.

Teach in circular interpolation CIP

Enter the intermediate and end positions for the circle interpolation CIP. You teach-in each of these separately in a separate block. The order in which you program these two points is not specified.

Note

Make sure that the cursor position does not change during teach-in of the two positions.

You teach-in the intermediate position in the "Circle intermediate position CIP" window.

You teach-in the end position in the "Circle end position CIP" window.

The intermediate or interpolation point is only taught-in with geometry axes. For this reason, at least 2 geometry axes must be set up for the transfer.

Teach-in A-spline

For Akima-spline interpolation, you enter interpolation points that are connected by a smooth curve.

Enter a starting point and specify a transition at the beginning and end.

You teach-in each interpolation point via "Teach in of position".



Software option

You require the "Spline-Interpolation" option for A Spline interpolation.



Machine manufacturer

Please observe the information provided by the machine manufacturer.

Procedure



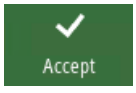
1. Teach-in mode is active.



2. Press the ">>" and "ASPLINE" softkeys.
   The "Akima-spline" window opens with the input fields.



3. Traverse the axes to the required position and, if necessary, set the transition type for the starting point and end point.



4. Press the "Accept" softkey.
   The new program block is inserted at the cursor-position.

   - OR -



   Press the "Cancel" softkey to cancel your input.

14.4.1 Input parameters for teach-in blocks

Parameters of axes for teach in

Parameter	Description
X	Axis position in the X direction
Y	Axis position in the Y direction
Z	Axis position in the Z direction
I	Coordinate of the circle intermediate points in the X direction
J	Coordinate of the circle intermediate points in the Y direction
K	Coordinate of the circle intermediate points in the Z direction

Feedrate (only for G1 and circle end position CIP)

Parameter	Description	Unit
F	Feedrate	mm/rev mm/min

Transition modes

Parameter	Description
G60	Exact stop
G64	Smoothing

Parameter	Description
G641	Programmable corner rounding
G642	Axis-specific corner rounding
G643	Block-internal corner rounding
G644	Axis dynamics corner rounding
G645	Smoothing

Motion types

Parameter	Description
CP	Path-synchronous
PTP	Point-to-point
PTPG0	Only G0 point-to-point

Transitional behavior of the spline curve

Parameter	Description
Start	Transitional behavior at the beginning • BAUTO - automatic calculation • BNAT - curvature is zero or natural • BTAN - tangential
End	Transitional behavior at the beginning • EAUTO - automatic calculation • ENAT - curvature is zero or natural • ETAN - tangential

14.5 Settings for teach-in

In the "Settings" window, you define which axes are to be included in the teach-in block and whether motion-type and continuous-path mode parameters are to be provided.

Procedure



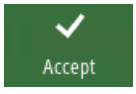
1. Teach-in mode is active.



2. Press the ">>" and "Settings" softkeys.
   The "Settings" window opens.



3. Under "Axes to be taught" and "Parameters to be taught", select the check boxes for the desired settings.



4. Press the "Accept" softkey to confirm the settings.

Handheld Terminal HT 10 for multi-touch operation 15

15.1 HT 10: Overview

The mobile handheld terminal HT 10 combines the functions of an operator panel and a machine control panel. You are thus provided the possibility of monitoring, operating, teaching and programming in immediate proximity to the machine.

The HT 10 can be operated with user interface "SINUMERIK Operate Generation 2".



① EMERGENCY STOP button
② Display / touch screen
③ Handwheel (optional)
④ Rot. over. switch
⑤ LED function keys

Operation

The fully graphic touch sensitive 10.1" TFT color display of the HT 10 allows touch operation. The 1280 x 800 pixel resolution is suitable for using the "Display Manager".

You can either use the handwheel or the mechanical traversing keys ("+"/"-") to traverse the axes.

The HT 10 is equipped with an EMERGENCY STOP button and a two-channel three-stage acknowledgment button.

You can also connect an external keyboard.

You can configure user-specific keys.

User-specific keys

The user-specific keys can be freely assigned. You can label the keys with your own texts in the national language.



Machine manufacturer
Please observe the information provided by the machine manufacturer.

Integrated machine control panel

The HT 10 has an integrated MCP. It comprises mechanical keys (e.g. start and stop) as well as keys emulated as softkeys.

A description of individual keys can be obtained in the chapter "Controls on the machine control panel".

Note

PLC interface signals that are triggered via the softkeys of the machine control panel menus are edge triggered.

Acknowledgment button

The HT 10 has an acknowledgment button. You can then activate the acknowledgment function for operator actions that require approval (e.g. displaying traversing keys).

Acknowledgment buttons are available for the following button positions:

- Released (no activation)
- Acknowledgment (center position) - acknowledgment for channels 1 and 2 is on the same switch
- Panic (completely pushed through)

Traversing keys

To traverse the axes of your machine using the mechanical traversing keys, "JOG" or "MDI" mode and the "REF. POINT" or "TEACH IN" functions must have been selected. You have to press the acknowledgment button depending on the setting.



Machine manufacturer
Please observe the information provided by the machine manufacturer.

Note

You can also traverse your machine axes using touch operation.

Virtual keyboard

A virtual keyboard is available for the easy entry of values.

Changing the channel

- You are able to switch the channel by touch operation in the status display:
  - In the Machine operating area (large status display), by touch operation of the channel display in the status display.
  - In the other operating areas (no status display), by touch operation of the channel display in the screen headers (yellow field).
- The "1... n CHANNEL" softkey is available in the machine control panel menu that can be reached via the operator menu key "U".

Operating area switchover

You display the operating area menu by touching the display symbol for the active operating area in the status display.

Handwheel

The HT 10 is equipped with a handwheel.

If you are using an HT 10, then you can traverse your machine axes via the handwheel.



Machine manufacturer

Please observe the information provided by the machine manufacturer.

More information

More information about connecting, commissioning and configuring user-specific keys is provided in the Handheld Terminal HT 10 Equipment Manual.

15.2 Machine control panel menu

You select keys from the machine control panel which are reproduced by the software by touch operation of the relevant softkeys.

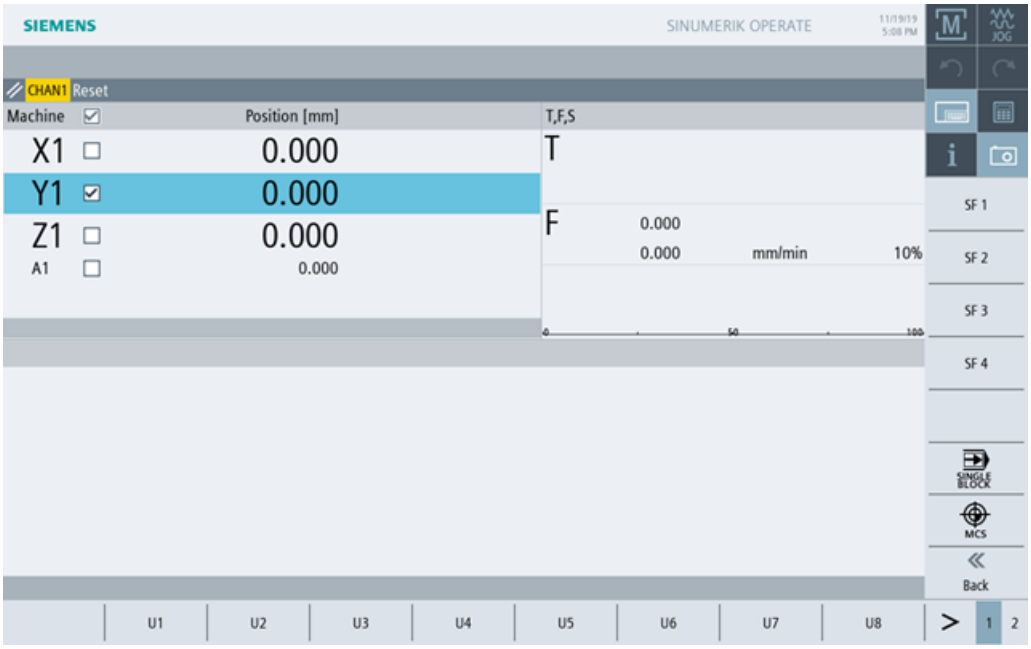
See Section "Controls on the machine control panel" for a description of the individual keys.

Note

PLC interface signals that are triggered via the softkeys of the machine control panel menus are edge triggered.

Showing and hiding

The user menu key "U" displays the CPF softkey bar (vertical softkey bar) and the user softkey bar (horizontal softkey bar).



Press the menu forward key to extend the horizontal user softkey bar. This means that additional softkeys are available.

Softkeys on the machine control panel menu

Available softkeys:

SF1- SF4, U1- 8	Customer keys, can be labeled in the local language
"WCS MCS" softkey	Switching between WCS and MCS
"Single Block" softkey	Switch single block execution on/off

Note

The window will automatically disappear when changing regions areas with the <MENU SELECT> key.

Axis selection

You select an axis in the actual value window by activating the checkbox in the header line of the actual value window.

By tapping the check box, a check box is displayed next to the axis name of the axis that has been enabled for axis selection.



Machine manufacturer

Please observe the information provided by the machine manufacturer.

You select an axis by activating the appropriate checkbox.

Note

To assign an axis to the handwheel, activate the handwheel via the "Handwheel" operator control element on the touch screen, and then select the axis using the checkbox.

Note

Orientation axes cannot be assigned to the handwheel.

Then traverse the axes at fixed increments using the operator control elements.

15.3 Virtual keyboard

The virtual keyboard is used as the input device for touch operator panels.

Open the virtual keyboard by double-clicking on an input-enabled operator control (program editor, editing fields). It is possible to position the virtual keyboard anywhere within the user interface.

You can choose between a full keyboard and a downsized keyboard featuring the numeric keypad only. With the full keyboard, it is possible to switch between English keyboard layout and the keyboard layout which corresponds with the actual language set for the respective country.



① Shift key for uppercase and lowercase letters
② Shift key for letters and special characters
③ Shift key for country-specific keyboard assignment
④ Shift key for full keyboard and numerical key block

Accepting the entered values

The entered values are accepted using the <INPUT> key.

Positioning of the virtual keyboard

Press and hold the open area to the left of the icon for "Close window" with the stylus or a finger. Move the keyboard to the desired position.

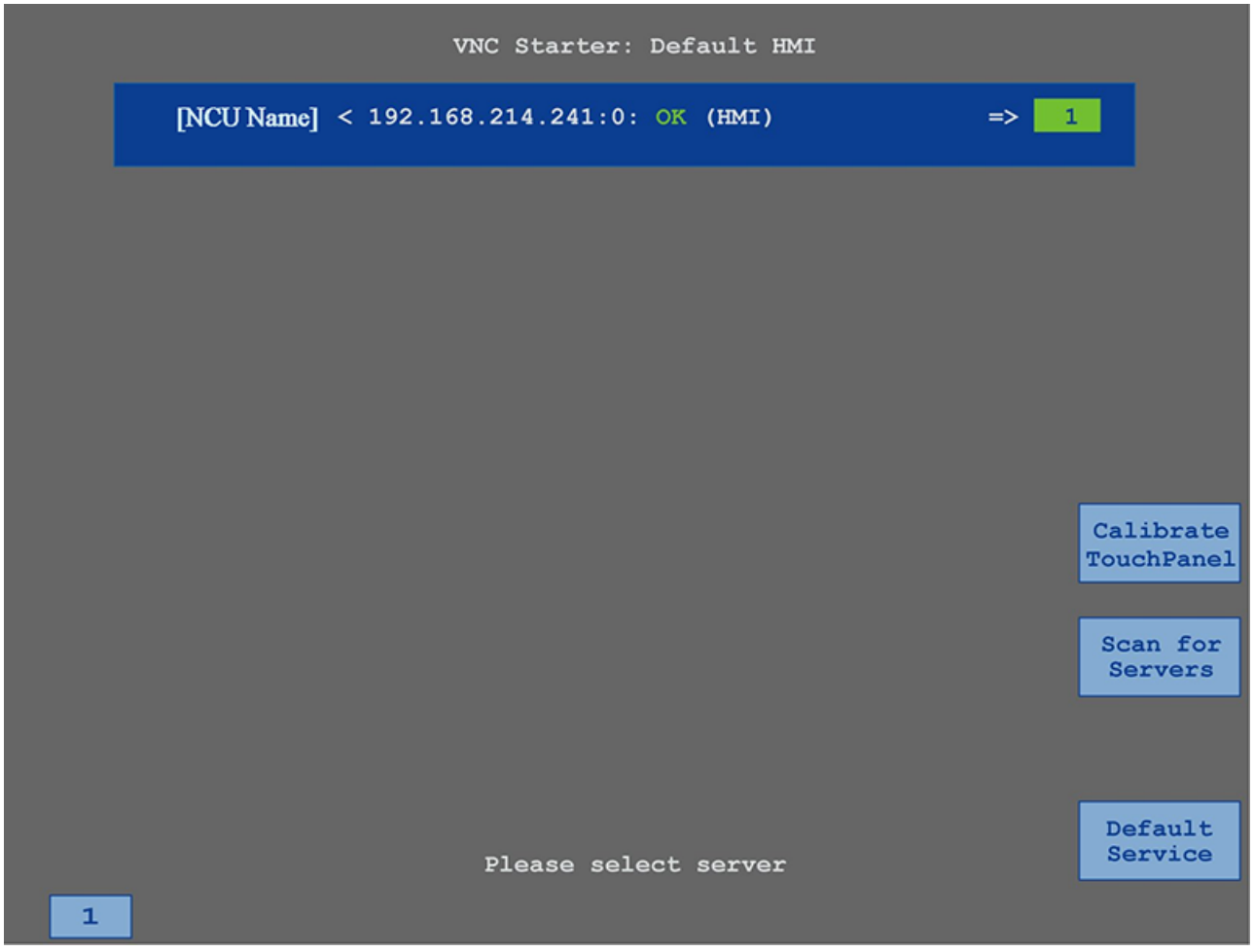
15.4 Calibrating the touch panel

It is necessary to calibrate the touch panel upon first connection to the controller.

Note

Recalibration

If the operation is not exact, then redo the calibration.



Procedure





1. Press the menu back key and the <MENU SELECT> key at the same time to start the TCU service screen.

2. Touch the "Calibrate TouchPanel" button.
   The calibration process will be started.

3. Follow the instructions on the screen and touch the three calibration points one after the other.

   The calibration process has terminated.
4. Touch the horizontal softkey "1" or the key with the number "1" to close the TCU service screen.

Ctrl-Energy

16

16.1 Functions

The "Ctrl-Energy" function provides you with the following options to improve the energy utilization of your machine.

Ctrl-E Analysis: Measuring and visualizing energy consumption

Acquiring the actual energy consumption is the first step to achieving better energy efficiency. The energy consumption is measured and displayed at the control using the SENTRON PAC energy meter.

Depending on the configuration and connection of the SENTRON PAC, you have the possibility of either measuring the power of the whole machine (including all auxiliaries) or only a specific load.

Independent of this, the power is determined directly from the drives and displayed.

You can determine both mechanical as well as electrical energy.



Software option

You require option "Ctrl-E analysis Advanced" for function "Ctrl-E analysis - Acquiring electrical power/energy".

Ctrl-E Profiles: Control of energy saving states of the machine

To optimize energy consumption, you have the option of defining up to 8 energy saving profiles and saving them.

Of these, 3 profiles are already predefined and can be used on your machine:

- Basic (machine standby) energy-saving mode
- Full (NC standby) energy-saving mode
- Maximum (Auto-Shut-Off) energy-saving mode

These predefined profiles are linked to specific conditions. As soon as these conditions are satisfied, you can use the signal provided at the PLC interface for your PLC application.

Via the user interface, you have the possibility of individually activating or deactivating these energy-saving profiles (e.g. to create a "coffee break key" on your machine control panel).

Note

Ctrl-E Deactivating profiles

Disable Ctrl-E profiles before a series commissioning in order to prevent the NCU unintentionally shutting down.



Machine manufacturer
Refer to the information provided by the machine manufacturer.

Note
Calling of the function via shortcut key

Press keys <CTRL> + <E> to call the "Ctrl Energy" function.

More information

Additional information on configuring is provided in the Ctrl-Energy System Manual.

16.2 Ctrl-E analysis

16.2.1 Displaying energy consumption

The SINUMERIK Ctrl-Energy entry screen provides an easy-to-interpret overview of the energy consumption of the machine. To display the values and the graphical representation, a Sentron PAC must be connected and a long-term measurement configured.

This shows a consumption display with the following bar chart:

- Current power display
- Energy consumption today
- Energy consumption previous day

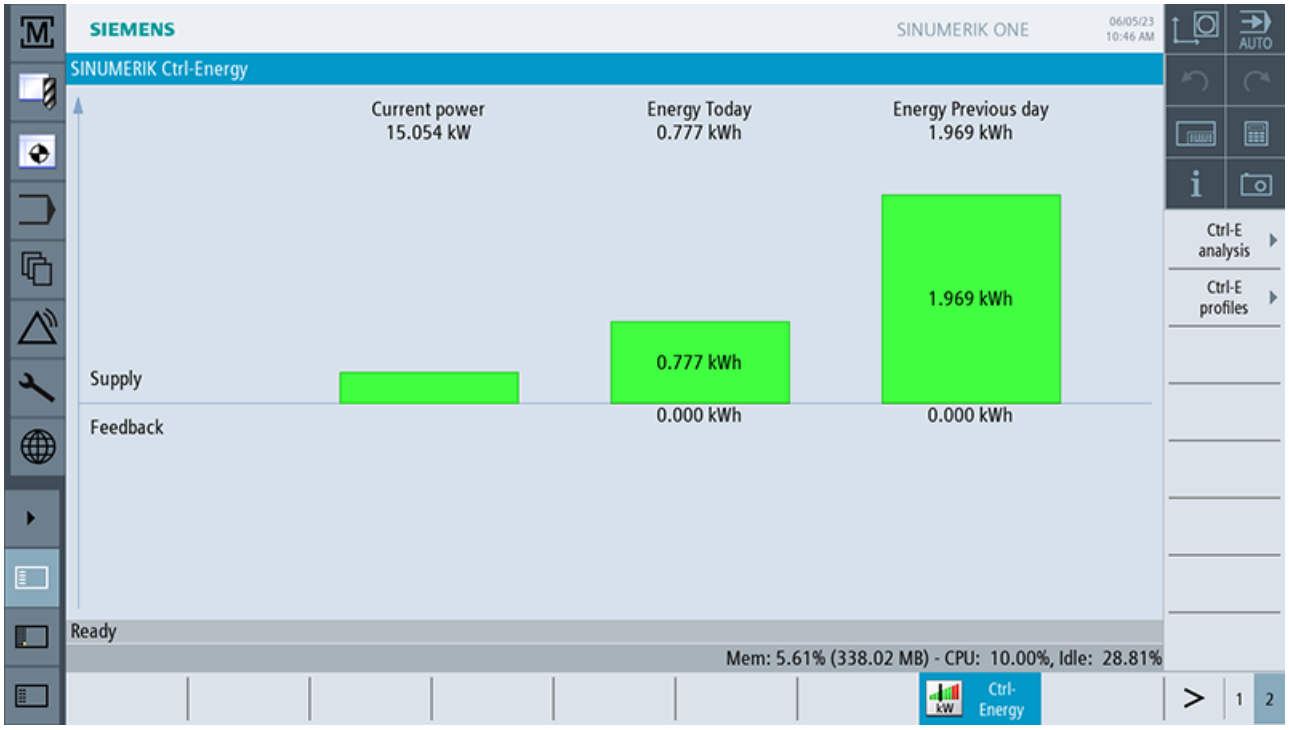


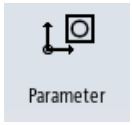
Figure 16-1 Ctrl-Energy entry screen with display of the current energy consumption

Display in the "Machine" operating area

The first row of the status display shows the current power status of the machine.

Display	Meaning
	A red bar indicates that the machine is not producing, but is still consuming energy (e.g. standby).
	A dark-green bar in the positive direction indicates that the machine is producing and is drawing energy from the line supply.
	A light-green bar in the negative direction indicates that the machine is producing and is feeding energy back into the line supply.

Procedure



1. Select the "Parameter" operating area.



2. Press the menu forward key and then the "Ctrl-Energy" softkey.



- OR -



+

Press the <Ctrl> + <E> keys.



The "SINUMERIK Ctrl-Energy" window opens.

16.2.2 Displaying the energy analyses

You can obtain a detailed overview of the energy usage in the "Ctrl-E analysis" window.



Software option

You require option "Ctrl-E analysis Advanced" for function "Ctrl-E analysis - Acquiring electrical power/energy".

You obtain the usage display for the following components:

- Sum of the SINAMICS drives
- Sum of the auxiliary units, if they are configured in the PLC
- Sentron PAC
- Sum of the machine

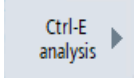
Detailed display of the energy usage

Further, you also have the option of listing the usage values for all drives and where relevant, all auxiliary units.

Procedure



1. You are in the "SINUMERIK Ctrl-Energy" entry window.



2. Press the "Ctrl-E analysis" softkey.

   The "Ctrl-E analysis" window opens. The summed consumption values of the components are displayed.



3. To display the energy consumption of individual drives and auxiliary units, select the particular component with the folder symbol in front of it and click on the
<Cursor right> key or symbol "Arrow to the right".
The selection of the subelements contained in it is opened, for example axes or auxiliary units.



4. To reclose the selection, select any subelement and click on <Cursor left> key or symbol "Arrow to the left".

16.2.3 Assigning language-dependent names for auxiliary units

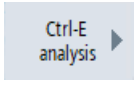
Introduction

You have the option of assigning language-dependent names to auxiliary units.

Procedure



1. You are in the "SINUMERIK Ctrl-Energy" entry window.



2. Press the "Ctrl-E analysis" softkey.
The "Ctrl-E analysis" window opens. The summed consumption values of the components are displayed.
You can find the list of auxiliary units under "Sum totals".



3. Select directory "Sum totals" and click on the
<Cursor right> key or symbol "Arrow to the right".
The selection of the auxiliary units contained under "Sum totals" opens.



4. Press softkey "Change texts".
You can now edit the names of the auxiliary units.



5. Press softkey "OK".
The new names for the auxiliary units are adopted.

Note

To change the names of the auxiliary units into other available languages, select the required language and then repeat steps 3 to 5.

If no language-dependent names for auxiliary units were saved in another language, then standard texts are displayed in the respective language. However, if names for auxiliary units were saved in English, then these are displayed in this case.

16.2.4 Making settings for measurements

Set the following in dialog "Settings":

- Energy measuring methods for drives
- Maximum memory for all measurements
- Maximum power consumption of the machine
- Sampling time and maximum possible measurement duration
- Time specification for ending the measurement

You define the measuring methods for drives for future measurements.

The maximum memory space for all measurements is the memory space that all measurements are allowed to occupy on the local data storage medium. Defining the maximum memory space prevents the local memory from being overfilled.
If you wish to save a measurement, and the defined memory space for has been fully used, then you must delete old measurements under "Manage measurements".

Setting "Maximum power consumption of the machine" serves as reference to display the power in the CTRL-E entry screen.

Using setting "Sampling interval", you can specify the duration of the energy measurements and therefore completely measure part programs with longer runtimes.



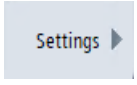
Machine manufacturer
Observe the information provided by the machine OEM.

Requirement

- You are in window "Ctrl-E analysis".

Procedure

Proceed as follows to make settings for measurements:



1. Press softkey "Settings".
   The "Settings" dialog opens.
2. Under "Energy measurement method for drives", select the required measuring method.
   You can select between electrical and mechanical measuring methods.
3. Under "Maximum memory for all measurements", enter the required value in megabytes.
4. Under "Maximum power consumption of the machine", enter the required value in KW, e.g. "100".

5. Set the sampling interval.

   By preselecting a suitable sampling interval, you can adapt the "Maximum possible measurement duration" to your specific application.

   Select the sampling interval so that the "Maximum possible measurement duration" only lies slightly above the actual measurement duration of a typical application.

   Note

   The higher the sampling rate, the more inaccurate the measurement.
6. Under "End measurement after", enter the time after which the measurement should be automatically ended.

16.2.5 Measuring and saving the energy consumption

For the currently selected axes, auxiliary units, SentronPAC or the complete machine you have the option of measuring and recording the energy consumption.

Measuring the energy consumption from part programs and starting and stopping part program sections

You can start and stop an energy measurement from a part program to acquire the energy consumption for specific machining steps.

Individual drives are taken into account for the measurement. You specify in which channel the start and stop of the part program should be initiated and how many repetitions you want to measure.

You also have the option of carrying out the energy measurement for part program sections if you activate check box "Measure by section".

In this case, measured values are only recorded if a section is active. The areas before and after are shown as 0 line in the graphic view From the part program, the energy measurement is started and stopped using softkeys "Start measurement" and "Stop measurement".

For the case that softkey "Stop measurement" is not actuated, the measurement ends at the latest after the specified number of repeats for the particular measurement have been performed or the measurement is terminated by actuating softkey "Cancel measurement". The measurement series of the individual repeats are displayed next to one another in the graphic view of the measurement results.

When externally evaluating the measurement results, the individually saved measurements that have been repeated can be used to generate an average value, for example.

Save measurements

Save the measurement history of the measured device, which was recorded while measuring a part program, for subsequent comparison.

Note

Using option "Ctrl-E analysis Advanced", you can save any number of measurements and assign your own file names to the measurements.

Measurement duration

The measurement duration is limited. The measurement is terminated if the maximum measurement time has been reached. The corresponding message is output in the dialog line.



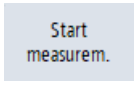
Machine manufacturer

Refer to the information provided by the machine manufacturer.

Requirement

- You are in window "Ctrl-E analysis".

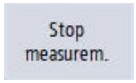
Procedure



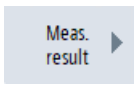
1. Press the "Start measurement" softkey.
   The "Setting Measurement: Select Device" window opens.
2. Select the desired device in the list, possibly activate the "Measure part program" check box, enter the number of repetitions and select the required channel.
   Activate check box "Measure by section" if you wish to record measured values only if a section is active.



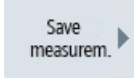
3. Press softkey "OK".
   The trace is started.



4. Press the "Stop measurement" softkey.
   The measurement is terminated.



5. Press the "Meas. result" softkey.



6. Press the "Save measurement" softkey to save the consumption values of the actual measurement.
   Dialog "Save measurement" is displayed.
7. In dialog "Save measurement", select a suggested name from the selection list or assign your own name for the measurement.
   The suggested names have a complete time stamp: Year, month, day, hours, minutes, seconds.

16.2.6 Evaluating measurement results

16.2.6.1 Overview

The subsequent evaluation options for your measurements are available under "Ctrl-E analysis > Measurement result":

- Graphic representation of the actual and saved measurements (Page 411)
- Tabular representation of measurement data and consumption values (Page 412)
- Comparative representation of the consumption values that are drawn in and fed back (Page 413)
- Display of the long time measurement of the energy consumption (Page 413)

If you press softkey "Meas. result", then in addition to the actual measurement, measurements that were last configured via "Select measurements" are displayed.

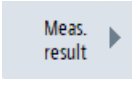
16.2.6.2 Displaying the measurement history

You have the option of displaying the measurement history as measurement curve.

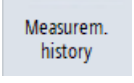
Requirement

- You are in window "Ctrl-E analysis".

Displaying measurement history as measurement curve



1. Press the "Meas. result" softkey.

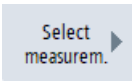


2. Press softkey "Measurement history".

   The current measurement (if available) as well as a maximum of the three last saved measurements (depending on the setting in "Select measurements") are displayed as measurement curve.

   You can zoom in or zoom out of the curve that is displayed using the zoom function.

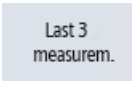
   You can also shift the curve that is displayed.



3. Press softkey "Select measurements" to graphically display additional measurements.

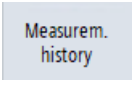
   Select those measurements that you would like to display in addition to the actual measurement.

   - OR -



   Press softkey "Last 3 measurements" to graphically display the 3 measurements that were last saved.

   In this case, 3 measurement curves with different colors are shown together with the measurement time.



4. Press softkey "Measurement history" again.

   All selected measurements are shown as measurement curve.

16.2.6.3 Displaying consumption values

You have the option of displaying the actual and saved usage values in a detailed table.

Display	Meaning
Name	Name of the measurement
Start of the measurement	Shows the time at which the measurement was started by pressing the "Start measurement" softkey.
Duration of the measurement [s]	Shows the measurement duration in seconds until the "Stop measurement" softkey is pressed.
Device	Displays the selected measured component. • Sentron PAC • Sum totals (if defined in the PLC) • Sum of the axes • Total, machine
Supplied energy [kWh]	Displays the energy drawn by the selected measured component in kilowatts per hour.
Regenerated energy [kWh]	Shows the regenerated energy of the selected measured component in kilowatts per hour.
Energy totals [kWh]	Shows the total of all measured drive values or the total of all axes as well as fixed value and Sentron PAC.

Display in the "Ctrl-E analysis" window: Table"

Requirement

- You are in window "Ctrl-E analysis".
- You have already saved measurements.

Procedure



1. Select one or several components and press the "Meas. result" softkey.



2. Press softkey "Measured values".
   In the window "Ctrl-E analysis: Measured values", measurement data and consumption values of up to three measurements last saved – and possibly an actual measurement – are displayed in a table.

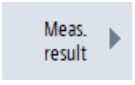
16.2.6.4 Comparing usage values

You have the option of comparing usage values (power drawn and power fed back) of actual and saved measurements.

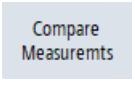
Requirement

- You are in window "Ctrl-E analysis".
- You have already saved measurements.

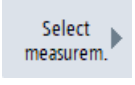
Procedure



1. Press the "Meas. result" softkey.

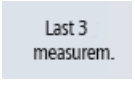


2. Press the "Compare measurements" softkey.
   Window "Ctrl-E analysis: Compare" opens.
   The power drawn and the recovered power of the actual measurement are displayed in a bar chart.



3. Press softkey "Select measurements" and select those measurements that you wish to compare with one another.

   - OR -



   Press softkey "Last 3 measurements" to display the comparison of the 3 measurements that were last saved.

4. If you only wish to see the actual measurement, then close window "Select measurements" with "OK".

16.2.6.5 Displaying the long-term measurement of the energy consumption

The long-term measurement of energy consumption is performed in the PLC and saved. The values from times in which the HMI is not active are also recorded.

Measured values

The infeed and regenerative power values as well as the sum of the power are displayed for the following periods:

- Current and previous day
- Current and previous month
- Current and previous year

Requirement

- You are in window "Ctrl-E analysis".
- The SENTRON PAC energy meter is connected and configured so that the energy of the complete machine can be measured.

Procedure



1. Press the "Long time measurement" softkey.
   The "SINUMERIK Ctrl-Energy Analysis Long-term Measurement" window opens.
   The results of the long-term measurement are displayed.



2. Press softkey "Back" to exit the long-term measurement.

16.2.7 Manage measurements

You can export, delete or rename saved measurements in area "Manage measurements".

Deleting and exporting are protected using protection levels.

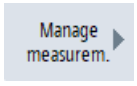
Machine manufacturer
Refer to the information provided by the machine manufacturer.

Requirement

- You are in window "Ctrl-E analysis > Measurement result".
- You have already saved measurements.

Exporting measurements

Proceed as follows to export measurements for further processing or analysis:



1. Press softkey "Manage measurements".
   Window "Ctrl-E analysis: Manage measurements" opens.



2. Select the measurements that you wish to export or press softkey "Select all" to select all measurements.



3. Click on softkey "Export".

4. Select the target directory.



5. Confirm with "OK".
   The data are saved in the target directory.

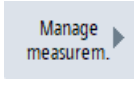
Note

You can rename measurements using softkey "Properties".

Deleting measurements

You can delete measurements to release memory space for new measurements or to prevent that existing measurements are automatically overwritten by new measurements.

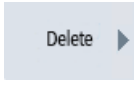
Proceed as follows to delete measurements:



1. Press softkey "Manage measurements".
   Window "Ctrl-E analysis: Manage measurements" opens.



2. Select the measurements that you wish to delete or press softkey "Select all" to select all measurements.



3. Click on softkey "Delete".



4. Confirm the prompt with "OK".
   The measurements are deleted.

16.3 Ctrl-E profiles

16.3.1 Creating and editing energy-saving profiles

You can make the following changes in the list of energy-saving profiles in the commissioning mode:

- Creating new energy-saving profiles with prewarning time and activation time period.
- Changing state conditions, prewarning time and activation time period
- Deleting energy-saving profiles
- Importing externally generated energy-saving profiles into the system
- Backing up energy-saving profiles on an external medium

Note

Maximum number of energy-saving profiles

A maximum of 8 energy-saving profiles can be created.

If the maximum number of profiles already exists, then the "New" softkey is no longer available. To create a new energy-saving profile, remove a profile using the "Delete" softkey or change an existing profile using the "Change" softkey.

Note

Access level of the energy-saving profiles for users

You specify the access level for the "Energy-saving profiles" softkey in the "Parameter" operating area using the machine data 51071 $MNS_ ACTIVATE_CTRL_E (protection level 1 as standard; this must be explicitly enabled by the manufacturer).

Note

Designations of energy-saving profiles that have been created

The names of the new energy-saving profiles that have been created are only visible at the control where you generated them.

Energy conditions

The following criteria are available for configuring an energy-saving profile:

State conditions	Meaning
No key on the operator panel pressed	To activate the energy-saving profile, it is not permissible that an operator panel key is pressed.
No key on the machine control panel pressed	To activate the energy-saving profile, it is not permissible that a machine control panel key is pressed.
No screen change realized	To activate the energy-saving profile it is not permissible that a screen is being changed.

State conditions	Meaning
No communication with an external device active (USB, Ethernet, V24)	To activate the energy-saving profile, it is not permissible that data is being exchanged with an external device, for example.
NC channel/channels in Reset	To activate the energy-saving profile, NC channel/channels must be in the Reset state.
Master computer signal prohibits	To activate the energy-saving profile, it is not permissible that a signal is received from the master computer.
PLC user signal prohibits	To activate the energy-saving profile, it is not permissible that a signal is received from the PLC user program. The OEM can logically combine other state conditions with this signal and influence the profiles.
Activation of the energy-saving profile after	Here, you enter the time period after which the energy-saving profile is activated.
Prewarning after	Here, you enter when the system starts to display the prewarning. **Note:** If you enter the same value into the "Activation of the energy-saving profile after" and "Prewarning after" fields, then no message is displayed specifying the remaining time up until the energy-saving profile is activated.

Note

Protection levels for editing energy-saving profiles

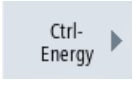
You define the protection level that is required in order to create or edit an energy-saving profile in machine data 51072 $MNS_ACCESS_EDIT CTRL_E (default, protection level 2 (service)).

Procedure



1. Select the "Startup" operating area.





2. Press the "HMI" and "Ctrl-Energy" softkeys.
   The window "SINUMERIK Ctrl-Energy: Definition of the energy-saving profile" is opened.

Newly creating/changing energy profiles



3. Position the cursor on the line where you wish to create a new energy-saving profile and press the "New" softkey.

   The window "SINUMERIK Ctrl-Energy: Definition of the energy-saving profile" is opened.

   In the "Designation" field, enter the desired name for the new energy-saving profile. Activate the required state conditions. Specify the times to display the prewarning limit and to activate the energy-saving profile.

   - OR -



   Position the cursor on the energy-saving profile that you want to edit and press the "Change" softkey.

   The window "SINUMERIK Ctrl-Energy: Definition of the energy-saving profile" is opened.

   In the "Designation" entry field, enter the modified name of the energy-saving profile. Select or deselect the state conditions that should be effective for the modified energy-saving profile - and when required, change the time data.



4. Press the "OK" softkey.

   The new or modified energy-saving profiles are activated.

   The modified texts (def_conditions<long>.ts) are saved in the \oem\sinumerik\hmi\lng directory.

Importing energy-saving profiles



5. Press the "Import profile" softkey.

   The "Import energy-saving profiles" window is displayed.

   Select the required XML file and press the "OK" softkey. After a prompt, the energy-saving profiles are downloaded into the PLC.



   The associated text files (def_conditions<long>.ts) are saved in the \oem\sinumerik\hmi\lng directory.

   Note:

   After importing the energy-saving profiles, a restart is required in order to make the profiles effective.

Exporting energy-saving profiles



6. Press the "Export profiles" softkey.

   The window "Export energy-saving profiles: Select storage location" opens.



7. Select the required storage location and press the "OK" softkey.

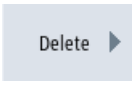
   The language-neutral state data are stored in the XML format, the associated texts in a .ts file. You have the option of selecting a name for the XML file (default: def_conditions.xml).

   The text files are saved under a fixed name. All of the defined energy-saving profiles are always exported.

   Note:

   Please note that the energy-saving profiles supplied from Siemens are also deleted. It is recommended that the energy-saving profiles are always exported and backed up.

Deleting energy-saving profiles



8. Select the energy-saving profile that you want to delete from the list and press the "Delete" softkey.



9. Confirm the prompt with "OK" to remove the profile.

16.3.2 Creating and editing energy-saving profiles

You can make the following changes in the list of energy-saving profiles in the commissioning mode:

- Creating new energy-saving profiles with prewarning time and activation time period.
- Changing state conditions, prewarning time and activation time period
- Deleting energy-saving profiles
- Importing externally generated energy-saving profiles into the system
- Backing up energy-saving profiles on an external medium

Note

Maximum number of energy-saving profiles

A maximum of 8 energy-saving profiles can be created.

If the maximum number of profiles already exists, then the "New" softkey is no longer available. To create a new energy-saving profile, remove a profile using the "Delete" softkey or change an existing profile using the "Change" softkey.

Note

Access level of the energy-saving profiles for users

You specify the access level for the "Energy-saving profiles" softkey in the "Parameter" operating area using the machine data 51071 $MNS_ ACTIVATE_CTRL_E (protection level 1 as standard; this must be explicitly enabled by the manufacturer).

Note

Designations of energy-saving profiles that have been created

The names of the new energy-saving profiles that have been created are only visible at the control where you generated them.

Energy conditions

The following criteria are available for configuring an energy-saving profile:

State conditions	Meaning
No key on the operator panel pressed	To activate the energy-saving profile, it is not permissible that an operator panel key is pressed.
No key on the machine control panel pressed	To activate the energy-saving profile, it is not permissible that a machine control panel key is pressed.
No screen change realized	To activate the energy-saving profile it is not permissible that a screen is being changed.
No communication with an external device active (USB, Ethernet)	To activate the energy-saving profile, it is not permissible that data is being exchanged with an external device, for example.
NC channel/channels in Reset	To activate the energy-saving profile, NC channel/channels must be in the Reset state.
Master computer signal prohibits	To activate the energy-saving profile, it is not permissible that a signal is received from the master computer.
PLC user signal prohibits	To activate the energy-saving profile, it is not permissible that a signal is received from the PLC user program. The OEM can logically combine other state conditions with this signal and influence the profiles.
Activation of the energy-saving profile after	Here, you enter the time period after which the energy-saving profile is activated.
Prewarning after	Here, you enter when the system starts to display the prewarning. **Note:** If you enter the same value into the "Activation of the energy-saving profile after" and "Prewarning after" fields, then no message is displayed specifying the remaining time up until the energy-saving profile is activated.

Note

Protection levels for editing energy-saving profiles

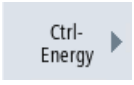
You define the protection level that is required in order to create or edit an energy-saving profile in machine data 51072 $MNS_ACCESS_EDIT CTRL_E (default, protection level 2 (service)).

Procedure



1. Select the "Setup" operating area.





2. Press the "HMI" and "Ctrl-Energy" softkeys.
   The window "SINUMERIK Ctrl-Energy: Definition of the energy-saving profile" is opened.

Newly creating/changing energy profiles



3. Position the cursor on the line where you wish to create a new energy-saving profile and press the "New" softkey.
   The window "SINUMERIK Ctrl-Energy: Definition of the energy-saving profile" is opened.
   In the "Designation" field, enter the desired name for the new energy-saving profile. Activate the required state conditions. Specify the times to display the prewarning limit and to activate the energy-saving profile.
   - OR -



   Position the cursor on the energy-saving profile that you want to edit and press the "Change" softkey.
   The window "SINUMERIK Ctrl-Energy: Definition of the energy-saving profile" is opened.
   In the "Designation" entry field, enter the modified name of the energy-saving profile. Select or deselect the state conditions that should be effective for the modified energy-saving profile - and when required, change the time data.



4. Press the "OK" softkey.
   The new or modified energy-saving profiles are activated.
   The modified texts (def_conditions<long>.ts) are saved in the \oem\sinumerik\hmi\lng directory.

Importing energy-saving profiles



5. Press the "Import profile" softkey.
   The "Import energy-saving profiles" window is displayed.
   Select the required XML file and press the "OK" softkey. After a prompt, the energy-saving profiles are downloaded into the PLC.



   The associated text files (def_conditions<long>.ts) are saved in the \oem\sinumerik\hmi\lng directory.
   Note:
   After importing the energy-saving profiles, a restart is required in order to make the profiles effective.

Exporting energy-saving profiles



6. Press the "Export profiles" softkey.
   The window "Export energy-saving profiles: Select storage location" opens.



7. Select the required storage location and press the "OK" softkey.

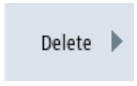
   The language-neutral state data are stored in the XML format, the associated texts in a .ts file. You have the option of selecting a name for the XML file (default: def_conditions.xml).

   The text files are saved under a fixed name. All of the defined energy-saving profiles are always exported.

   Note:

   Please note that the energy-saving profiles supplied from Siemens are also deleted. It is recommended that the energy-saving profiles are always exported and backed up.

Deleting energy-saving profiles



8. Select the energy-saving profile that you want to delete from the list and press the "Delete" softkey.



9. Confirm the prompt with "OK" to remove the profile.

16.3.3 Using the energy-saving profile

In the "Ctrl-E profiles" window, you can display all of the defined energy-saving profiles. Here, you have the option of directly activating or inhibiting a required energy-saving profile, or re-enabling profiles.

SINUMERIK Ctrl-Energy energy-saving profiles

Display	Meaning
Energy-saving profile	All energy-saving profiles are listed.
active in [min]	The remaining time until the defined profile is reached is displayed.

Note

Disable all energy-saving profiles

For example, in order not to disturb the machine while measurements are being made, select "Disable all".

Once the pre-warning time of a profile has been reached, an alarm window that shows the remaining time is displayed. Once the energy-saving mode has been reached, then an appropriate message is displayed in the alarm line.

Predefined energy-saving profiles

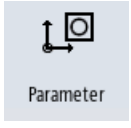
Energy-saving profile	Meaning
Simple energy-saving mode (machine standby)	Machine units that are not required are either throttled or shut down. When required, the machine is immediately ready to operate again.
Full energy-saving mode (NC standby)	Machine units that are not required are either throttled or shut down. Wait times are incurred at the transition into the ready to operate state.
Maximum energy-saving mode (auto shut-off)	The machine is completely switched-off. Longer wait times are incurred at the transition into the ready to operate state.

Machine manufacturer

The selection and function of the displayed energy-saving profiles can differ.

Please observe the information provided by the machine manufacturer.

Procedure



1. Select the "Parameter" operating area.

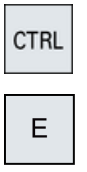


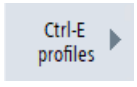


2. Press the menu forward key and then the "Ctrl-Energy" softkey.

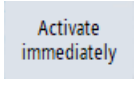
- OR -

Press the <CTRL> + <E> keys.





3. Press the "Ctrl-E profile" softkey.
   The "Ctrl-E Profile" window opens.

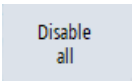


4. Position the cursor on the required energy-saving profile and press the "Activate immediately" softkey if you wish to directly activate this state.



5. Position the cursor on the required energy-saving profile and press the "Disable profile" softkey if you wish to disable this state.
   The profile is inhibited and does not become active. The energy-saving profile is grayed-out and displayed without any time information.
   The labeling of the "Disable profile" softkey changes to "Enable profile".
   Press the "Enable profile" softkey in order to withdraw the energy-saving profile disable.





5. Press the "Disable all" softkey in order to disable all states.
   All the profiles are disabled and cannot become active.
   The labeling of the "Disable all" softkey changes to "Enable all".



6. Press the "Enable all" softkey to withdraw the disable for all profiles.

Editing the PLC user program

17

17.1 Introduction

A PLC user program consists to a large degree of logical operations to implement safety functions and to support process sequences. These logical operations include the linking of various contacts and relays. These logic operations are displayed in a ladder diagram.

Editing ladder diagrams

You can edit the ladder diagrams in the Ladder editor.

This allows you to use all the operations supported by the particular PLC type.

Some of the functions differ in scope from that described in the manual.



Machine manufacturer

Please observe the information provided by the machine manufacturer.

Further information

You can find further information in the PLC Function Manual.

17.2 Displaying and editing PLC properties

17.2.1 Displaying PLC properties

The following PLC properties can be displayed in the "SIMATIC LAD" window:

- Operating state
- Name of the PLC project
- PLC system version
- Cycle time
- Processing time
  The processing time can be reset.

You can also update the project data or load a new PLC user program.

Procedure



1. Select the "Start-up" operating area.





2. Press the "PLC" softkey.
   The ladder diagram representation opens and displays the PLC information.

17.2.2 Resetting the processing time

You can reset the processing time of the PLC user program.

Procedure



1. The Ladder editor is open.



2. Press the "Reset proc. time" softkey.
   The data of the processing time is reset.

17.2.3 Loading modified PLC user program

Load the project data to the PLC if some changes have been made to the project data and a new PLC user program is available.

When the project data is loaded, the data classes are saved and loaded to the PLC.

Requirement

Check whether the PLC is in the Stop state.

Note

PLC in the RUN state

If the PLC is in the RUN state, a corresponding message is displayed and the "Load in Stop" and "Load in Run" softkeys appear.

With "Load in Stop", the PLC is set to the Stop state and the project is stored and loaded into the CPU.

With "Load in Run", the loading operation is continued and the PLC project loaded to the PLC. Only those data classes that have actually been changed are loaded, i.e. generally INDIVIDUAL data classes.

Procedure



1. The Ladder editor is open.
   You have changed project data.



2. Press the "PLC Stop" softkey if the PLC is in the Run state.



3. Press the "Load to CPU" softkey to start the loading operation.
   All data classes are loaded.



4. When the PLC project has been loaded, press the "PLC Start" softkey to switch the PLC to the Run state.

17.3 Displaying and editing PLC and NC variables

Changes can only be made to the NC/PLC variables with the appropriate password.

⚠ WARNING

Incorrect parameterization

Changes in the states of NC/PLC variables have a considerable influence on the machine. Incorrect configuration of the parameters can endanger life and cause damage to the machine.

In the "NC/PLC Variables" window, enter the NC system variables and PLC variables that you want to monitor or change in the list:

- Variable
  Address for NC/PLC variable.
  Incorrect variables have a red background and are displayed with a # character in the value column.
- Comment
  Any comment on the variables.
  The columns can be displayed and hidden.
- Format
  Specify the format in which the variable is to be displayed.
  The format can be specified (e.g. floating point).
- Value
  Displays the actual value of the NC/PLC variables

Note

Syntax for symbolic addressing

In addition to the icon, the area (IArea, QArea, MArea) must always be specified. The area for data blocks is the block name.

The subsequent table explains the differences between absolute and symbolic addressing.

PLC variables	Absolute addressing	Symbolic addressing
Inputs	• Input bit (Ix), input byte (IBx), input word (IWx), input double word (IDx) Example: I2.0, IB2, IW2, ID2 • Input bit (Ex), input byte (EBx), input word (EWx), input double word (EDx) Example: E2.0, EB2, EW2, ED2	IArea.<variable name> Example: IArea.VarInput2, IArea.McpIn.KeypadAxis.feedrateStart
Outputs	• Output bit (Qx), output byte (QBx), output word (QWx), output double word (QDx) Example: Q2.0, QB2, QW2, QD2 • Output bit (Ex), output byte (EBx), output word (EWx), output double word (EDx) Example: A2.0, AB2, AW2, AD2	QArea.<variable name> Example: QArea.VarOutput2, QArea.McpOut.KeypadAxis.feedrateStart
Bit memory	• Memory bit (Mx), memory byte (MBx), memory word (MWx), memory double word (MDx) Example: M2.0, MB2, MW2, MD2	MArea.<variable name> Example: MArea.VarNote2
Data	• Data block (DBx): Data bit (DBXx), data byte (DBBx), data word (DBWx), data double word (DBDx) Example: DB2.DBX2.0, DB2.DBB2, DB2.DBW2, DB2.DBD2	<DBname>.<variable name> Example: LBP_NC.E_NCKready, LBP_Axis1.A_DriveEnable, ModMcp.fromMcp.KeyPadModeGroup.reset
Times	-	Times can only be accessed with symbolic address
Counters	-	Counters can only be accessed with symbolic address

Formats	
B	Binary
H	Hexadecimal
D	Decimal without sign
+/-D	Decimal with sign
F	Floating point (for double words)
A	ASCII character

Notation examples

Permissible notation for variables:

- PLC variables:
  - Addresses: EB2, A1.2, DB2.DBW2
  - Names: LBP_NC.E_NCKready, LBP_HMI.A_MCPChan, IArea.VarInput1, QArea.VarOutput1

  Note

  With symbolic addressing, the operand must be set in quotation marks (" ") depending on the characters and digits it contains.

  More information can be found in the FAQ entry "When must identifiers or operands be place in "quotation marks" in STEP 7 (TIA Portal)? (https://support.industry.siemens.com/cs/document/109477857)"

- NC variables:
  - NC system variables: Notation $AA_IM[1]
  - User variables / GUD: Notation GUD/MyVariable[1,3]
  - OPI notation: /CHANNEL/PARAMETER/R[u1,2]

Note

If the PLC user program writes a string in an NC/PLC variable, the string will only be displayed correctly if the variable is parameterized as a field variable of the type "A" (ASCII) on the NC side.

Example of a field variable

Variable	Format
DBx.DBBy[<number>]	A
LBP_ModeGroup.MG[2].A_MGReset	B

Inserting variables

The start value for "Filter/Search" of variables differs. For example, to insert the variable $R[0], enter the following start value:

- The start value is 0 if you filter according to "System variables".
- The start value is 1 if you filter according to "All (no filter)". In this case, all signals are displayed and shown in the OPI notation.

The GUD from the machine data is only displayed in the Search window for the variable selection when the associated definition file has been activated. Otherwise, the sought variables must be entered manually, e.g. GUD/SYG_RM[1]

The following machine data is representative for all variable types (INT, BOOL, AXIS, CHAR, STRING): MD18660 $MN_MM_NUM_SYNACT_GUD_REAL[1].

Note

Display of NC/PLC variables

- System variables can be dependent on the channel. When the channel is switched over, the values from the selected channel are displayed.
  You have the option of having the variable displayed for a specific channel, e.g. $R1:CHAN1 and $R1:CHAN2. The values of channel 1 and channel 2 are displayed regardless of the channel you are in.
- For user variables (GUD), it is not necessary to make a specification according to global or channel-specific GUD. The first element of a GUD array starts with index 0 as for NC variables.
- Using the tooltip, you can display the OPI notation for NC variables (except for GUD).

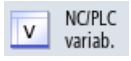
Servo variables

Servo variables can only be selected and displayed at "Diagnostics" → "Trace".

Changing and deleting values



1. Select the "Diagnostics" operating area.



2. Press the "NC/PLC variables" softkey.

   The "NC/PLC Variables" window opens.

3. Position the cursor in the "Variable" column and enter the required variable.



4. Press the <INPUT> key.

   The operand is displayed with the value.



5. Press the "Details" softkey.

   The "NC/PLC Variables: Details" window opens. The information for "Variable", "Comment" and "Value" is displayed in full length.



6. Position the cursor in the "Format" field and select the required format with <SELECT>.



7. Press the "Display comments" softkey.

   The "Comments" column is displayed. You have the option of creating comments or editing existing comments.

   Press the "Display comments" softkey once again to hide the column.





8. Press the "Change" softkey if you would like to edit the value.

   The "Value" column can be edited.



9. Press the "Insert variable" softkey if you wish to select a variable from a list of all existing variables and insert this.

   The "Select variable" window opens.



10. Press the "Filter/search" softkey to restrict the display of variables (e.g. to mode group variables) using the "Filter" selection box and/or select the desired variable using the "Search" input box.



11. Press the "Delete all" softkey if you would like to delete all the entries for the operands.



12. Press the "OK" softkey to confirm the changes or the deletion.

    - OR -



    Press the "Cancel" softkey to cancel the changes.

Editing a variable list

You can edit the variable list using the "Insert line" and "Delete line" softkeys.



If you press the softkey, a new line inserted before the line marked by the cursor.

You can only use the "Insert line" softkey if there is at least one empty line at the end of the variable list.

The softkey is deactivated if there is no empty line.



If you press the "Delete line" softkey, the line marked by the cursor is deleted.

An empty line will be added at the bottom of the variable list.

Changing operands

Depending on the type of operand, you can increment or decrement the address by 1 place at a time using the "Operand +" and "Operand -" softkeys.

Note

Axis names as index

For axis names, the "Operand +" and "Operand -" softkeys do not act as index, e.g. for $AA_IM[X1].





Examples

DB97.DBX2.5
Result: DB97.DBX2.6
$AA_IM[1]
Result: $AA_IM[2]
MB201
Result: MB200
/Channel/Parameter/R[u1,3]
Result: /Channel/Parameter/R[u1,2]

17.4 Displaying and editing PLC signals in the status list

PLC signals are displayed and can be changed in the "PLC Status List" window.

The following lists are shown

Inputs (IB)

Bit memories (MB)

Outputs (QB)

Variables (VB)

Data (DB)

Setting the address

You can go directly to the desired PLC address to monitor the signals.

Editing

You can edit the data.

Procedure



1. Select the "Commissioning" operating area.



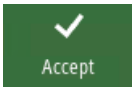
2. The Ladder editor is open.



3. Press the "Status list" softkey.
   The "Status List" window appears.



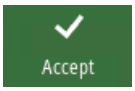
4. Press the "Set address" softkey.
   The "Set Address" window appears.



5. Activate the desired address type (e.g. DB), enter the value and press the "Accept" softkey.
   The cursor jumps to the specified address.



6. Press the "Change" softkey.
   The "RW" input field can be edited.



7. Enter the desired value and press the "Accept" softkey.

17.5 View of the program blocks

17.5.1 Displaying information on the program blocks

You can display all the logic and graphic information of a program block.

- Logic information
  The following information is displayed in a ladder diagram (LAD):
  - Networks with program parts and current paths
  - Electrical current flow through a number of logical operations
- Select program block
  Select the program block that you want to display.
- Program status
  You call information about the program status.
- Symbolic address
  You select between specification in absolute or symbolic address.
- Zoom
  You can enlarge or reduce the ladder diagram.
- Search
  You can use the "Search" function to go directly to a position in the PLC user program that you wish to edit, for example.
- Edit
  You can insert, edit or delete networks.
- Symbol information
  You can display all symbolic identifiers used in the selected network.

Procedure



1. Select the "Startup" operating area.



2. Press the "PLC" softkey.





3. Press the "Window 1" or "Window 2" softkey.

17.5.2 Structure of the user interface

The following figure shows the user interface.

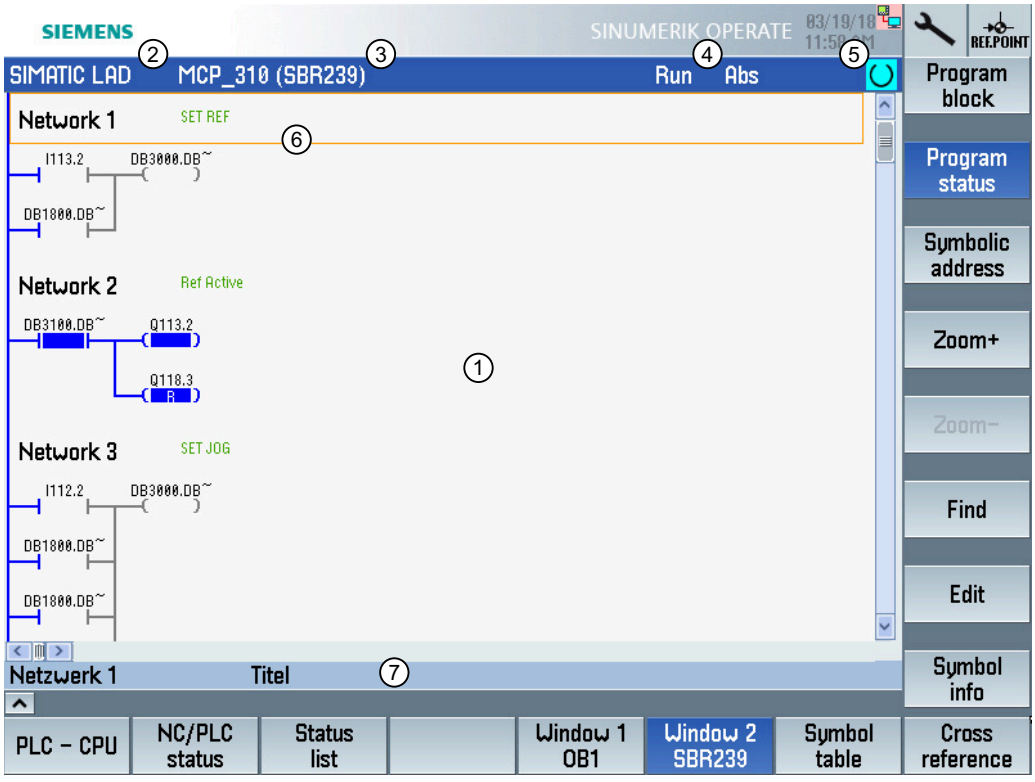


Figure 17-1 Screen layout

Table 17-1 Key to screen layout

Screen element	Display	Meaning
1	Application area	
2	Supported PLC program language "LAD"	
	LAD*	Program change exists
3	Name of the active program block Representation: Symbolic name (absolute name)	
4	Program status	
	Run	Program is running
	Stop	Program is stopped
	Status of the application area	
	Sym	Symbolic representation
	Abs	Absolute representation
5	Display of the active keys (<INPUT>, <SELECT>)	

Screen element	Display	Meaning
6	Focus Performs the tasks of the cursor	
7	Information line Displays information, e.g. for searching	

17.5.3 Control options

In addition to the softkeys and the navigation keys, there are further shortcuts in this area.

Shortcuts

The cursor keys move the focus over the PLC user program. When the window borders are reached, scrolling is performed automatically.

Shortcuts	Action
NEXT WINDOW CTRL + ◀	To the first column of the row
END CTRL + ▶	To the last column of the row
PAGE UP	Up a screen
PAGE DOWN	Down a screen
◀ ▶ ▲ ▼	One field to the left, right, up or down
CTRL + NEXT WINDOW - OR - CTRL + ▲	To the first field of the first network

Shortcuts		Action
CTRL	END	To the last field of the last network
- OR -		
CTRL	▼	
CTRL	PAGE UP	Open the next program block in the same window
CTRL	PAGE DOWN	Open the previous program block in the same window
SELECT		The function of the Select key depends on the position of the input focus. • Table row: Displays the complete text row • Network title: Displays the network comment • Command: Displays all operands
INPUT		If the input focus is positioned on a command, all operands including the comments are displayed.

17.5.4 Displaying the program status

You can display the program status.

The following Information is displayed:

- Program status: "Run" or "Stop"
- Status of the application area: "Sym" or "Abs"

Displaying the program status

If your PLC has the "Progress status" function, the status values as well as the information flow are displayed at the time of execution of the operations. The status of the local data memory and the accumulators is also displayed.

The "Progress status" display is also controlled using the "Program stat." softkey

Colors for displaying the program status

In the progress status, different colors are used to display information.

Display	Color
Signal flow of the busbar when the status is active	Blue
Signal flow in the networks	Blue
All operations that are active and that are executed without error (corresponds to the signal flow)	Blue
Status of the Boolean operations (corresponds to the signal flow)	Blue
Timers and counters active	Green

Error during execution	Red
No signal flow	Gray
No network executed	Gray
STOP operating state	Gray

Procedure

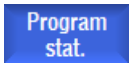




1. The program block view is open.



2. Press the "Program stat." softkey to display the program status display in the status display.



3. Press the "Program stat." softkey again to hide the program status display in the status display again.

17.5.5 Changing the address display

You can choose between specification in absolute or symbolic address.

Elements for which there are no symbolic identifiers are automatically displayed with absolute identifiers.

Procedure





1. The program block view is open.



2. Press the "Symbol. address" softkey.
   The list of operands is displayed sorted according to symbolic address.



3. To return to the display showing the absolute addresses press the "Symbol. address" softkey again.

17.5.6 Enlarging/reducing the ladder diagram

You can enlarge or reduce the representation of the ladder diagram.

Procedure



1. The program block view is open.





2. Press the "Zoom +" softkey to enlarge the section of the ladder diagram.
   After enlarging, the "Zoom -" softkey is available.



3. Press the "Zoom -" softkey to reduce the section of the ladder diagram again.

17.5.7 Program block

17.5.7.1 Displaying and editing the program block

You can create and edit program blocks and display further information as follows:

- Local variables
  You can display local variables of a block.
- Create new program block
  You can create a new program block.
- Open program block
  You can display all the logic and graphical information of the program block and you can edit the block.
- Properties
  You can display the properties of a block and edit them, when required.
- Protection
  You can protect the block with a password. Then, the block cannot be opened without entering the password.



Machine manufacturer
Please observe the information provided by the machine manufacturer.

Procedure



1. Select the "Startup" operating area.



2. Press the "PLC" softkey.





3. Press the softkey:
   - "Window 1 OB1"
     - OR -
   - "Window 2 SBRO"



4. Press the "Program block" softkey.

17.5.7.2 Displaying local variable table

You have the option of displaying the local variable table of a block.

The following information is listed in the tables.

Name	Freely assigned
Variable type	Selection: • IN • IN_OUT • OUT • TEMP
Data type	Selection: • BOOL • BYTE • WORD • INT • DWORD • DINT • REAL
Comment	Freely assigned

Procedure

1. The "Program Block" window is open.



3. Press the "Local variables" softkey.
   The "Local Variables" window appears and lists the created variables.

17.5.7.3 Creating a program block

Overview

Using the Ladder editor, you can perform a PLC diagnosis to find fault causes or program errors.

If an input or output fails, a disturbance in the plant could result, making modifications to the PLC user program necessary. For this, you can create additional program blocks.

Creating a program block

If program blocks are missing, you can add them via the vertical softkey bar. You can also delete blocks via the vertical softkey bar. You can also modify the networks of interrupt routines and subprograms on the control and save and load your changes.

Marshalling data

You can "rewire" inputs (via INT_100) or outputs (via INT_101) for service purposes.

Note

Saving the PLC project when changing the operating area

If you have created blocks or inserted, removed, or edited networks in a block, you must save the project before you change from the PLC area to another operating area. Transfer the project to the PLC using the "Load to CPU" softkey. If this is not done, all of the changes will be lost and must be re-entered.

Please note the corresponding program information.



Machine manufacturer

Please observe the information provided by the machine manufacturer.

Creating a new block

You create new program blocks in the Ladder editor.

Name	SBR, INT_100, INT_101, INT_0 The number from the selection field "Number of subprogram" is taken for the name of the INT block.
Author	Maximum 48 characters are permitted.
Number of subprogram	You must select a free subprogram number between 0 and 255. For INT100, INT101, and INT0, the field is filled automatically and cannot be edited.
Data class	Individual / Manufacturer This data class is automatically filled by the system and cannot be edited.
Comment	Maximum 100 lines and 4096 characters are permitted.

Note

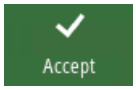
Access protection

You have the option of protecting blocks that have been newly created against being accessed.

Procedure



1. The "Program Block" window is open.
2. Press the "New" softkey.
   The "Properties" window opens.
3. Select a block and enter the name of the author, the number of the sub-program and, if relevant, a comment.
   Note:
   To insert a line break, use the key combination <Alt> + <INPUT>.
   When you have completed your entries, press the "Accept" softkey.



17.5.7.4 Opening a program block in the window

You can display all the logic and graphical information of a program block.

Procedure



1. The relevant block is selected and the "Program Block" window is open.
2. Select the desired block and press "Open".

   The block is displayed in the currently active window 1 or window 2.

17.5.7.5 Displaying/canceling the access protection

You can password protect your program organizational units (POUs) in the PLC 828 programming tool. This prevents other users from accessing this part of the program. This means that it is invisible to other users and is encrypted when it is downloaded.

A lock symbol is used to show a password-protected POU in the block overview and in the ladder diagram.

Procedure



1. The relevant block is selected and the "Program Block" window is open.
2. Press the "Protection" softkey.
   The "Protection" window opens.

Removing protection

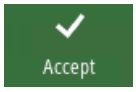
3. Enter the password.
   - "This program block remains protected" is activated:
     You have the option of editing or deleting the block. Protection is reactivated when you load the PLC user program to the PLC.
   - "This program block remains protected" is deactivated:
     Block protection is permanently withdrawn. After being loaded to the PLC, the PLC user program is not protected.

Setting protection

4. Enter the required password in the first line "Please enter password" and re-enter the password in the second line.
5. Activate the "Protect all program blocks using this password" checkbox, if you wish to protect all of the user program blocks.

   Note:

   Program blocks that are already password-protected are not affected.



6. Press the "Accept" softkey.

17.5.7.6 Editing block properties subsequently

You can edit the title, author and comments of a block.

Note:

To insert a line break in comment, use the key combination <ALT> + <INPUT>.

Note

Block names, subprogram names and data class assignments cannot be edited.

Procedure

1. The relevant block is selected and the "Program Block" window is open.



2. Press the "Properties" softkey.
   The "Properties" window appears.

17.5.8 Editing a program block

17.5.8.1 Editing the PLC user program

You can change and extend PLC user programs.

All the operations supported by the PLC type are available for the editing. Subprograms and interrupt programs can be added and deleted.

Note

Saving changes

If you make changes in the program, you must save the project before you change from the PLC area to another operating area. You can transfer the project to the PLC using the "Load to CPU" softkey. If this is not done, all of the changes will be lost and must be re-entered.

Please note the corresponding program information.

Editing functions

- Edit block
  - Create connecting lines, contacts, coils, and boxes
  - Change operands
  - Delete operations
- Network
  - Create
    Create and then edit a new network
  - Edit
    Subsequently edit networks
  - Delete
    Delete networks

Further information

You can find further information in the PLC Function Manual.

17.5.8.2 Editing a program block

You can edit program blocks.

Requirement

Before you can edit program blocks, the program status must have stopped.

If the status is active, you will receive a message that the program status must have stopped.

1. To stop the program status automatically, press the "OK" softkey.
   - OR -
   To stop the program status, press the "Program stat." softkey again.

Procedure

1. The ladder logic display (LAD) is open.



2. Press the "Program block" softkey and select the block that you want to edit.



3. Press the "Open" softkey.
   The program block is opened in the appropriate window.



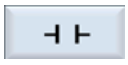
4. Press the "Change" softkey to open the editing mode.
   If the program status display is active, a message is displayed that you can confirm with "OK".



5. If you want to insert connecting lines, position the cursor at the desired position and press the appropriate softkey, e.g. "-->".



   - OR -



   Press the "Contacts" softkey and select the desired operation in the list that opens.

   - OR -

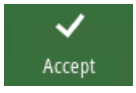


   Press the "Coils" softkey and select the desired operation in the list that opens.

   - OR -



   Press the "Boxes" softkey and select the desired operation in the list that opens.



5. Press the "Accept" softkey to confirm the respective action.
   The changes will now be saved.
   Note:
   The changes only take effect when the user program is loaded to the CPU.

Note

Save changes

If you make changes in the program, you must save the project before you change from the PLC area to another operating area. You can transfer the project to the PLC using the "Load to CPU" softkey. If this is not done, all of the changes will be lost and must be re-entered.

Please note the corresponding program information:

Loading the program to the CPU





1. Press the "PLC-CPU" and "Load to CPU" softkeys.



2. Press the "OK" softkey to start the load operation.
   After the error-free compilation of the program, the PLC is switched to the STOP state and loaded into the PLC.

17.5.8.3 Deleting a program block

You can delete program blocks.

Procedure

1. The relevant block is selected and the "Program Block" window is open.



2. Select the block and press "Delete".



5. Press "OK" to delete the block.

   - OR -



   Press "Cancel" to abort the action.



Machine manufacturer
Please observe the information provided by the machine manufacturer.

17.5.8.4 Inserting and editing networks

You can create a new network and then insert operations (bit operation, assignment, etc.) at the selected cursor position.



Machine manufacturer
Please observe the information provided by the machine manufacturer.

The bit combinations comprise one or several logical operations and the assignment to an output / bit memory.

If the cursor is moved further to the left with the arrow key, the type of assignment or a logic operation can be selected. A further logic operation cannot be placed to the right of an assignment. A network must always be terminated with an assignment.

Procedure

1. A block is selected.



2. Press the "Edit" softkey.

3. Position the cursor on a network.



4. Press the "Insert network" softkey.

   - OR -



   Press the <INSERT> key.

   If the cursor is positioned on "Network x", a new, empty network is inserted behind this network.



5. Position the cursor on the desired element below the network title and press the "Insert operation" softkey.

   The "Insert Operation" window appears.



6. Select a bit operation and press the "OK" softkey.



7. Press the "Insert operand" softkey.



8. Enter the logic operation or the command and press the <INPUT> key to complete the entry.



9. Position the cursor on the operation that you want to delete and press the "Delete operation" softkey.

   - OR -



   Position the cursor on the title of the network that you want to delete and press the "Delete network" softkey.

   - OR -



   Press the <DEL> key.

   The network, including all the logic operations and operands and/or the selected operation is deleted.

17.5.8.5 Editing network properties

You can edit the network properties of a block.

Network title and network comment

The title can have a maximum of 3 lines and 128 characters. The comment must not contain more than 100 lines and 4096 characters.

Procedure



1. The ladder diagram display (LAD) is open.





2. Use the cursor keys to select the network that you want to edit.





3. Press the <SELECT> key.

   The "Network Title / Comment" window opens and shows the title and a possibly assigned comment for the selected network.



5. Press the "Change" softkey.

   The fields can be edited.

   Note:

   To insert a line break in comment, use the key combination <ALT> + <INPUT>.



6. Enter the changes and press the "OK" softkey to transfer the data to the user program.

17.5.9 Displaying the network symbol information table

All of the symbolic identifiers used in the selected network are displayed in the "Network Symbol Information Table" window.

The following information is listed:

- Names
- Absolute addresses
- Comments

The symbol information table remains empty for networks that do not contain any global symbols.

Procedure

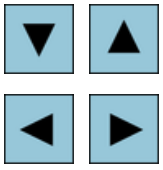




1. The ladder diagram display (LAD) is open.



2. Select the desired network and press the "Symbol info" softkey.
   The "Network Symbol Information Table" window appears.



3. Use the cursor keys to move within the table.

17.6 Displaying symbol tables

You can display the symbol tables that are used to obtain an overview of the global operands available in the project.

The name, address and possibly also a comment is displayed for each entry.

Procedure



1. The Ladder editor is open.

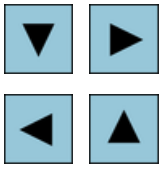




2. Press the "Symbol table" and "Sym. tab. selection" softkeys.
   The list with the symbol table entries is displayed.



3. Select the desired table and press the "Open" softkey.
   The table is displayed.



4. Use the cursor keys to select the desired entry.

17.7 Displaying cross references

You can display all the operands used in the PLC user project and their use in the list of cross-references.

This list indicates in which networks an input, output, bit memory, etc. is used.

The list of cross references contains the following information:

- Block
- Address in the network
- Context (command ID)

Symbolic and absolute address

You can select between specification in absolute or symbolic address.

Elements for which there are no symbolic identifiers are automatically displayed with absolute identifiers.

Opening program blocks in the ladder diagram

From the cross references, you can go directly to the position in the program where the operand is used. The corresponding block is opened in window 1 or 2 and the cursor is set on the corresponding element.

Procedure



1. The Ladder editor is open.



2. Press the "Cross refs." softkey.
   The list of cross references opens and the operands are displayed sorted according to absolute address.



3. Press the "Symbol. address" softkey.
   The list of operands is displayed sorted according to symbolic address.



4. Press the "Absolute address" softkey to return to the display showing the absolute addresses.





5. Select the desired cross reference and press the "Open in window 1" or "Open in window 2" softkey.
   The ladder diagram is opened and the selected operand is marked.



6. Press the "Find" softkey.
   The "Find / Go To" window opens.



7. Select "Find operand" or "Go to" and enter the sought element or the desired line and select the search order (e.g. search up).



8. Press the "OK" softkey to start the search.



9. If an element is found that corresponds to the sought element, but is not at the appropriate position, press the "Find next" softkey to find where the search term occurs next.

17.8 Searching for operands

Use the search function in very large PLC user programs to go directly to a position that you wish to edit, for example.

Restricting the search

- "Window 1" / "Window 2"
  With "Go to", you can jump directly to the desired network.
- "Cross references", "Symbol table"
  With "Go to", you can jump directly to the desired line.

Requirement

Window 1 / window 2, the symbol tables or the list of cross references is open.

Procedure



1. Press the "Find" softkey.
   A new vertical softkey bar appears. The "Find / Go To" window opens at the same time.



2. Select the "Find operand" entry in the first input field if you are searching for a specific operand and enter the search term in the "Find" input field.



3. Select the search range (e.g. Find all).



4. Select the "In this program unit" or "In all program units" entry if you are in "Window 1" or "Window 2" or in the symbol table in order to restrict the search.



5. Press the "OK" softkey to start the search.
   If the operand you are searching for is found, the corresponding line is highlighted.



Press the "Continue search" softkey if the operand found during the search does not correspond to the element you are looking for.

- OR -



Press the "Cancel" softkey if you want to cancel the search.

Service and diagnostics

18

18.1 Alarm, error, and system messages

18.1.1 Displaying alarms

If the machine develops a fault in operation, an alarm is generated and machining is possibly interrupted.

The error text that is displayed together with the alarm number gives you more detailed information on the error cause.

You have the possibility of saving all of the relevant diagnostic data to a ZIP file, which you can subsequently send to the hotline for analysis.

⚠ CAUTION
Dangers for persons and machines Carefully check the system, based on the description of the active alarm(s). Resolve the cause of the alarms. Then acknowledge the alarms in the specified way. Failure to observe this warning will place your machine, workpiece, saved settings and possibly even your own safety at risk.

Note

Alarm filtering by the machine OEM

If a filter icon is displayed in the "Alarms" window at the top right, then the machine OEM has activated alarm filtering.

Alarm overview

You can display all upcoming alarms and acknowledge them.

The alarm overview contains the following information:

- Date and time
- Cancel criterion
  The delete criterion specifies the key or softkey that can be used to acknowledge the alarm.
- Alarm number
- Alarm text

Procedure



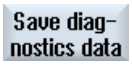
1. Select the "Diagnostics" operating area.



2. Press the "Alarm list" softkey.
   The "Alarms" window appears.
   All pending alarms are displayed.
   The "Hide SI alarms" softkey is displayed if safety alarms are pending.



3. Press the "Hide SI alarms" softkey if you do not wish to display SI alarms.



4. Press the "Save diag. data" softkey if the cause of the alarm is unknown.
   The function collects all available LOG files of the operating software and saves them to the following directory:
   \user\sinumerik\didac\out_<Date-Time>.7z
5. If there is a problem with the system, you can send the ZIP file to the SINUMERIK hotline to help with the analysis of the problem.

Cancel alarms

In the "Cancel" column it is symbolized how you delete the pending alarms from the alarm list.

6. Position the cursor on an alarm.
7. If an NCK-POWER ON alarm is displayed, turn the unit off and back on (main switch), or press NCK-POWER ON.
   - OR -
   If an NC-Start alarm is displayed, press the <NC-Start> key.
   - OR -
   If a RESET alarm is displayed, press the <RESET> key.
   - OR -



If a Cancel alarm is displayed, press the <ALARM CANCEL> key or press the "Cancel Alarm delete" softkey.

- OR -



- OR -



If an HMI alarm is displayed, press the "Delete HMI alarm" softkey.

- OR -
If a dialog alarm of the HMI is displayed, press the <RECALL> key.
- OR -
If a PLC alarm is displayed, press the key provided by the machine manufacturer.

- OR -



If a PLC alarm of the type SQ is displayed, press the "Acknowl. alarm" softkey.

The softkeys are activated when the cursor is on the corresponding alarm.

Acknowledgement symbols

Symbol	Meaning
	NCK POWER ON
	NC start
	RESET alarm
	Cancel alarm
	HMI alarm
	Dialog alarms of the HMI
	PLC alarm
	PLC alarm of the SQ type (alarm number from 800000)
	Safety alarms



Machine manufacturer

Observe the information provided by the machine OEM.

18.1.2 Displaying an alarm log

A list of all the alarms and messages that have occurred so far are listed in the "Alarm Log" window.

Up to 500 administered, incoming and outgoing events are displayed in chronological order.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure



1. Select the "Diagnostics" operating area.



2. Press the "Alarm log" softkey.

   The "Alarm Log" window opens.

   All of the coming and going events - that have occurred since the HMI was started - are listed.



3. Press the "Display new" softkey to update the list of displayed alarms/ messages.



4. Press the "Save Log" softkey.

   The log that is currently displayed is stored as text file alarmlog.txt in the system data in directory `/user/sinumerik/hmi/log/alarm_log`.

18.1.3 Displaying messages

PLC and part program messages may be issued during machining.

These message will not interrupt the program execution. Messages provide information with regard to a certain behavior of the cycles and with regard to the progress of machining and are usually kept beyond a machining step or until the end of the cycle.

Overview of messages

You can display all issued messages.

The message overview contains the following information:

- Date
- Message number
  is only displayed for PLC messages
- Message text

Note

Message filtering by the machine OEM

If a filter icon is displayed in the "Messages" window at the top right, then the machine OEM has activated message filtering.

Procedure



1. Select the "Diagnostics" operating area.



2. Press the "Messages" softkey.
   The "Messages" window appears.

18.1.4 Sorting, alarms, faults and messages

If a large number of alarms, messages or alarm logs are displayed, you have the option of sorting these in an ascending or descending order according to the following criteria:

- Date (alarm list, messages, alarm log)
- Number (alarm list, messages)

As a consequence, for every extensive lists, you can obtain the required information faster.

Procedure



1. Select the "Diagnostics" operating area.



2. Press the "Alarm list", "Messages" or "Alarm log" softkey to display the requested messages and interrupts.

...





3. Press the "Sort" softkey.



The list of entries is sorted in descending order according to date, i.e. the most recent information is at the beginning of the list.





4. Press the softkey "Ascending" to sort the list in the ascending order.
   The most recent event is shown at the end of the list.



5. Press the "Number" softkey if you wish to sort the alarm list or the list with messages according to numbers.



6. Press the "Descending" softkey to display the list in decreasing/descending order again.

18.2 PLC and NC variables

18.2.1 Displaying and editing PLC and NC variables

Changes can only be made to the NC/PLC variables with the appropriate password.

⚠ WARNING

Incorrect parameterization

Changes in the states of NC/PLC variables have a considerable influence on the machine. Incorrect configuration of the parameters can endanger life and cause damage to the machine.

In the "NC/PLC Variables" window, enter the NC system variables and PLC variables that you want to monitor or change in the list:

- Variable
  Address for NC/PLC variable.
  Incorrect variables have a red background and are displayed with a # character in the value column.
- Comment
  Any comment on the variables.
  The columns can be displayed and hidden.
- Format
  Specify the format in which the variable is to be displayed.
  The format can be specified (e.g. floating point).
- Value
  Displays the actual value of the NC/PLC variables

Note

Syntax for symbolic addressing

In addition to the icon, the area (IArea, QArea, MArea) must always be specified. The area for data blocks is the block name.

The subsequent table explains the differences between absolute and symbolic addressing.

PLC variables	Absolute addressing	Symbolic addressing
Inputs	• Input bit (Ix), input byte (IBx), input word (IWx), input double word (IDx) Example: I2.0, IB2, IW2, ID2 • Input bit (Ex), input byte (EBx), input word (EWx), input double word (EDx) Example: E2.0, EB2, EW2, ED2	IArea.<variable name> Example: IArea.VarInput2, IArea.McpIn.KeypadAxis.feedrateStart
Outputs	• Output bit (Qx), output byte (QBx), output word (QWx), output double word (QDx) Example: Q2.0, QB2, QW2, QD2 • Output bit (Ex), output byte (EBx), output word (EWx), output double word (EDx) Example: A2.0, AB2, AW2, AD2	QArea.<variable name> Example: QArea.VarOutput2, QArea.McpOut.KeypadAxis.feedrateStart
Bit memory	• Memory bit (Mx), memory byte (MBx), memory word (MWx), memory double word (MDx) Example: M2.0, MB2, MW2, MD2	MArea.<variable name> Example: MArea.VarNote2
Data	• Data block (DBx): Data bit (DBXx), data byte (DBBx), data word (DBWx), data double word (DBDx) Example: DB2.DBX2.0, DB2.DBB2, DB2.DBW2, DB2.DBD2	<DBname>.<variable name> Example: LBP_NC.E_NCKready, LBP_Axis1.A_DriveEnable, ModMcp.fromMcp.KeyPadModeGroup.reset
Times	-	Times can only be accessed with symbolic address
Counters	-	Counters can only be accessed with symbolic address

Formats	
B	Binary
H	Hexadecimal
D	Decimal without sign
+/-D	Decimal with sign
F	Floating point (for double words)
A	ASCII character

Notation examples

Permissible notation for variables:

- PLC variables:
  - Addresses: EB2, A1.2, DB2.DBW2
  - Names: LBP_NC.E_NCKready, LBP_HMI.A_MCPChan, IArea.VarInput1, QArea.VarOutput1

  Note

  With symbolic addressing, the operand must be set in quotation marks (" ") depending on the characters and digits it contains.

  More information can be found in the FAQ entry "When must identifiers or operands be place in "quotation marks" in STEP 7 (TIA Portal)? (https://support.industry.siemens.com/cs/document/109477857)"

- NC variables:
  - NC system variables: Notation $AA_IM[1]
  - User variables / GUD: Notation GUD/MyVariable[1,3]
  - OPI notation: /CHANNEL/PARAMETER/R[u1,2]

Note

If the PLC user program writes a string in an NC/PLC variable, the string will only be displayed correctly if the variable is parameterized as a field variable of the type "A" (ASCII) on the NC side.

Example of a field variable

Variable	Format
DBx.DBBy[<number>]	A
LBP_ModeGroup.MG[2].A_MGReset	B

Inserting variables

The start value for "Filter/Search" of variables differs. For example, to insert the variable $R[0], enter the following start value:

- The start value is 0 if you filter according to "System variables".
- The start value is 1 if you filter according to "All (no filter)". In this case, all signals are displayed and shown in the OPI notation.

The GUD from the machine data is only displayed in the Search window for the variable selection when the associated definition file has been activated. Otherwise, the sought variables must be entered manually, e.g. GUD/SYG_RM[1]

The following machine data is representative for all variable types (INT, BOOL, AXIS, CHAR, STRING): MD18660 $MN_MM_NUM_SYNACT_GUD_REAL[1].

Note

Display of NC/PLC variables

- System variables can be dependent on the channel. When the channel is switched over, the values from the selected channel are displayed.
  You have the option of having the variable displayed for a specific channel, e.g. $R1:CHAN1 and $R1:CHAN2. The values of channel 1 and channel 2 are displayed regardless of the channel you are in.
- For user variables (GUD), it is not necessary to make a specification according to global or channel-specific GUD. The first element of a GUD array starts with index 0 as for NC variables.
- Using the tooltip, you can display the OPI notation for NC variables (except for GUD).

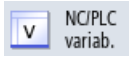
Servo variables

Servo variables can only be selected and displayed at "Diagnostics" → "Trace".

Changing and deleting values



1. Select the "Diagnostics" operating area.



2. Press the "NC/PLC variables" softkey.

   The "NC/PLC Variables" window opens.

3. Position the cursor in the "Variable" column and enter the required variable.



4. Press the <INPUT> key.

   The operand is displayed with the value.



5. Press the "Details" softkey.

   The "NC/PLC Variables: Details" window opens. The information for "Variable", "Comment" and "Value" is displayed in full length.



6. Position the cursor in the "Format" field and select the required format with <SELECT>.



7. Press the "Display comments" softkey.

   The "Comments" column is displayed. You have the option of creating comments or editing existing comments.

   Press the "Display comments" softkey once again to hide the column.





8. Press the "Change" softkey if you would like to edit the value.

   The "Value" column can be edited.



9. Press the "Insert variable" softkey if you wish to select a variable from a list of all existing variables and insert this.

   The "Select variable" window opens.



10. Press the "Filter/search" softkey to restrict the display of variables (e.g. to mode group variables) using the "Filter" selection box and/or select the desired variable using the "Search" input box.



11. Press the "Delete all" softkey if you would like to delete all the entries for the operands.



12. Press the "OK" softkey to confirm the changes or the deletion.

    - OR -



    Press the "Cancel" softkey to cancel the changes.

Editing a variable list

You can edit the variable list using the "Insert line" and "Delete line" softkeys.



If you press the softkey, a new line inserted before the line marked by the cursor.

You can only use the "Insert line" softkey if there is at least one empty line at the end of the variable list.

The softkey is deactivated if there is no empty line.



If you press the "Delete line" softkey, the line marked by the cursor is deleted.

An empty line will be added at the bottom of the variable list.

Changing operands

Depending on the type of operand, you can increment or decrement the address by 1 place at a time using the "Operand +" and "Operand -" softkeys.

Note

Axis names as index

For axis names, the "Operand +" and "Operand -" softkeys do not act as index, e.g. for $AA_IM[X1].



Examples

DB97.DBX2.5

Result: DB97.DBX2.6

$AA_IM[1]

Result: $AA_IM[2]



MB201

Result: MB200

/Channel/Parameter/R[u1,3]

Result: /Channel/Parameter/R[u1,2]

18.2.2 Saving and loading screen forms

You have the option of saving the configurations of the variables made in the "NC/PLC variables" window in a screen form that you reload again when required.

Editing screen forms

If you change a screen form that has been loaded, then this is marked using with * after the screen form name.

The name of a screen form is kept in the display after switching-off.

Procedure

1. You have entered values for the desired variables in the "NC/PLC variables" window.



2. Press the ">>" softkey.



3. Press the "Save screen" softkey.
   The "Save screen: Select archiving" window opens.



4. Position the cursor on the template folder for variable screen forms in which your actual screen form should be saved and press the "OK" softkey.
   The "Save screen: Name" window opens.



5. Enter the name for the file and press the "OK" softkey.
   A message in the status line informs you that the screen form was saved in the specified folder.
   If a file with the same name already exists, they you will receive a prompt.



6. Press the "Load screen" softkey.
   The "Load screen" window opens and displays the sample folder for the variable screen forms.
7. Select the desired file and press the "OK" softkey.
   You return to the variable view. The list of all of the predefined NC and PLC variables is displayed.

18.3 Accessing PLC data

Folder "PLC data" contains files that are saved in the internal PLC memory of the PLC. You can read access these files in SINUMERIK Operate via the data tree of the system data. You can open individual files. You can also copy files from folder "PLC data" to other folders under "System data".

The following subdirectories are available in folder "PLC data" (assuming that these subdirectories contain data):

- User data
- Recipes
- Log files

Note

To read access folder "PLC data", as a minimum you must have access level 1 (machine manufacturer).

18.4 Creating screenshots

You can create screenshots of the current user interface.

Each screenshot is saved as a file and stored in the following folder:

/**user**/sinumerik/hmi/log/screenshot

Procedure

Ctrl + P Press the "Ctrl + P" key combination.

A screenshot of the current user interface is created in .png format.

The file names assigned by the system are in ascending order from "SCR_SAVE_0001.png" to "SCR_SAVE_9999.png". You can create up to 9,999 screenshots.

Copy file



1. Select the "Commissioning" operating area.



2. Press the "System data" softkey.
3. Open the folder specified above, and select the required screenshots.



4. Press the "Copy" softkey.

   - OR -



   Press the "Cut" softkey.



5. Open the required archive directory, e.g. on a USB flash drive and press the "Paste" softkey.

Note

WinSCP

You can also copy the screenshots to a Windows PC using "WinSCP".

Note

Open files

Open the files in SINUMERIK Operate to view the screenshots. On a Windows PC, you can open the files using a graphic program such as "Office Picture Manager".

18.5 Version

18.5.1 Displaying version data

The following components with the associated version data are specified in the "Version data" window:

- System software
- PLC basic program
- PLC user program
- System expansions
- OEM applications
- Hardware
- Engineering tools used

Information is provided in the "Nominal version" column as to whether the versions of the components differ from the version supplied on the memory card.



The version displayed in the "Actual version" column matches the version of the memory card.



The version displayed in the "Actual version" column does not match the version of the memory card.

Note

Under "Engineering Tools used" you can find information about the application, which was used to generate the installed software.

You may save the version data. Version data saved as text files can be further processed as required or sent to the hotline in the event of an error.

Procedure



1. Select the "Diagnostics" operating area.



2. Press the "Version" softkey.
   The "Version Data" window appears.
   Data of the available components is displayed.



3. Select the component for which you would like more information.





4. Press the "Details" softkey, in order to receive more exact information on the components displayed.

18.5.2 Save information

All the machine-specific information of the control is combined in a configuration via the user interface. You then have the option of saving the machine-specific information on the drives that have been set up.

Procedure



1. Select the "Diagnostics" operating area.



2. Press the "Version" softkey.
   It takes some time to call the version display. While the version data is being determined a progress message box and the appropriate text are displayed in the dialog line.



3. Press the "Save" softkey.
   The "Save version information: Select Archive" window opens. The following storage locations are offered depending on the configuration:
   - Local drive
   - Network drives
   - USB
   - Version data (archive: Data tree in the "HMI data" directory)



4. Then press the "New directory" softkey if you wish to create your own directory.



5. Press the "OK" softkey. The directory is created.



6. Press the "OK" softkey again to confirm the storage location.
   The "Save version information: Name" window opens.

7. Specify the desired settings.
   - "Name:" input field
     The file name is preassigned <Machine name/no.>+<Memory card number>. "_config.xml" or "_version.txt" is automatically attached to the file names.
   - "Comment:" input field
     You can enter a comment that is stored with the configuration data.

- Version data (.TXT)
  Activate the checkbox if you wish to output the pure version data in the text format.
- Configuration data (.XML)
  Activate the checkbox if you wish to output the configuration data in the XML format.
  The configuration file contains the data you entered under Machine identity, the license requirements, the version information and the logbook entries.



8. Press the "OK" softkey to start the data transfer.

18.6 Logbook

The logbook provides you with the machine history in an electronic form.

If service is carried out on the machine, this can be electronically saved. This means that it is possible to obtain a picture about the "History" of the control and to optimize service.

Editing the logbook

You can edit the following information:

- Editing information on the machine identity
  - Machine name/No.
  - Machine type
  - Address data
- Make logbook entries (e.g. "filter replaced")
- Deleting logbook entries

Note

Deleting logbook entries

Up to the 2nd commissioning, you have the option to delete all of the entered data up to the time of the first commissioning.

Output of the logbook

You have the possibility of exporting the logbook by generating a file using the "Save version" function in which the logbook is contained as section.

See also

Save information (Page 469)

18.6.1 Displaying and editing the logbook

Procedure



1. Select the "Diagnostics" operating area.



2. Press the "Version" softkey.



3. Press the "Logbook" softkey.
   The "Machine logbook" window opens.

Editing end customer data



4. You have the option of changing the address data of the end customer using the "Change" softkey.

   - OR -



   Using the "Clear" softkey, you can delete all logbook entries.



   All entries, except the date of the first commissioning, are deleted. Softkey "Clear" is deactivated.

Note

Deleting logbook entries

As soon as the 2nd commissioning has been completed, the "Clear" softkey to delete the logbook data is no longer available.

18.6.2 Making a logbook entry

Using the "New logbook entry" window to make a new entry into the logbook.

Enter your name, company and department and a brief description of the measure taken or a description of the fault.

Note

Setting line breaks

If you wish to make line breaks in the "Fault diagnostics/measure" field, use the key combination <ALT> + <INPUT>.

The date and entry number are automatically added.

Sorting the entries

The logbook entries are displayed numbered in the "machine logbook" window.

More recent entries are always added at the top in the display.

Procedure

1. The logbook is opened.



2. Press the "New entry" softkey.

   The "New logbook entry" window opens.



3. Enter the required data and press the "OK" softkey.

   You return to the "Machine logbook" window and the entry is displayed below the machine identity data.

Note

Deleting logbook entries

Up to the completion of the 2nd commissioning, you have the option to delete the logbook entries up to the time of the first commissioning using the "Clear" softkey.

Searching for a logbook entry

You have the option for searching for specific entries using the search function.

1. The "Machine logbook" window is opened.
2. 

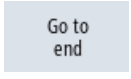
Press the "Find" softkey.
3. Enter the desired term in the search form. You can make a search according to date/time, company name/department or according to fault diagnostics/measure.
   The cursor is positioned on the first entry that corresponds to the search term.
4. 

Press the "Continue search" softkey if the entry found is not the one that you are looking for.

Additional search option



Press the "Go to Beginning" softkey to start the search at the latest entry.



Press the "Go to End" softkey to start the search at the oldest entry.

18.7 Remote diagnostics

18.7.1 Setting remote access

You can influence the remote access to your control in the "Remote diagnostics (RCS)" window.

You set the rights for all remote operating types in this window. The selected rights are defined from the PLC and using the setting at the HMI.

The HMI can restrict the rights specified from the PLC, but however, cannot extend the rights beyond the PLC rights.

If the settings made permit access from outside, access also requires manual or automatic confirmation.

Rights for remote access

The "Specified by PLC" field shows the access rights for remote access or remote monitoring specified from the PLC.



Machine manufacturer
Follow the machine manufacturer's instructions.

In the "Selected in the HMI" selection box, you have the possibility of setting rights for remote control:

- Do not permit remote access
- Permit remote monitoring
- Permit remote control

Depending on the combination of the settings in the HMI and in the PLC, the valid status as to whether access is permitted or not is shown in the "Resulting from this" line.

Settings for the confirmation dialog box

However, if the settings made for "Specified from the PLC" and "Selected in the HMI" permit access from outside, access also requires manual or automatic confirmation.

As soon as a remote access is permitted, at all of the active operating stations, a query dialog box is displayed for the operator at the active operating station to either confirm or reject an access.

If there is no local operation, the control behavior can be set for this particular scenario. You define for how long this window will be displayed and whether remote access will be automatically rejected or accepted after the confirmation has expired.

Display of the state



Remote monitoring active



Remote control active

If remote access is active, these icons on the status line inform you whether remote access is currently active or whether only monitoring is permitted.

Procedure



1. Select the "Diagnostics" operating area.



2. Press the "Remote diag." softkey.
   The "Remote diagnostics (RCS)" window opens.



3. Press the "Change" softkey.
   The "Selected in the HMI" is activated.



4. If you want to use remote control, select the "Permit remote control" item.
   Remote control is only possible if the "Allow remote operation" item is specified in the fields "Specified by PLC" and "Selected in HMI".
5. Enter new values in the group "Behavior for remote access confirmation" if you would like to change the behavior relating to confirming remote access.



6. Press the "OK" softkey.
   The settings are accepted and saved.

Further information

You will find further information on configuration in the SINUMERIK Operate Commissioning Manual.

18.7.2 Permit modem

You can permit remote access to your control via a teleservice adapter IE connected at X127.



Machine manufacturer
Follow the machine manufacturer's instructions.



Software option
You need the "Access MyMachine /P2P" option to display the "Allow modem" softkey.

Procedure



1. The "Remote diagnostics (RCS)" window is open.



2. Press the "Allow modem" softkey.
   Access to the control via modem is enabled so that a connection is established.



3. To block access again, press the "Allow modem" softkey again.

18.7.3 Request remote diagnostics

Using the "Request remote diagnostics" softkey, you have the option from your control of actively requesting remote diagnostics from your machine OEM.

Access via modem must be enabled if access is to be possible via a modem.



Machine manufacturer
Follow the machine manufacturer's instructions.

When you request remote diagnostics, a window opens with the relevant pre-assigned data and values of the ping service. If required, you can ask your machine manufacturer for this data.

Data	Meaning
IP address	IP address of the remote PC
Port	Standard port that is intended for remote diagnostics
Send duration	Duration of the request in minutes
Send interval	Cycle in which the message is sent to the remote PC in seconds
Ping send data	Message for the remote PC

Procedure



1. The "Remote diagnostics (RCS)" window is open.



2. Press the "Request remote diagnostics" softkey.
   The "Request remote diagnostics" window is displayed.



3. Press the "Change" softkey if you would like to edit the values.



4. Press the "OK" softkey.
   The request is sent to the remote PC.

See also

Permit modem (Page 475)

18.7.4 Exit remote diagnostics

Procedure



1. The "Remote diagnostics (RCS)" is opened and it is possible that remote monitoring or remote access is active.
2. Block the modem access if access via modem is to be blocked.

   - OR -

   In the "Remote Diagnostics (RCS)" window, reset the access rights to "Permit no remote access".

See also

Permit modem (Page 475)

Setting remote access (Page 474)

Index

"

"SINUMERIK Operate Gen. 2"
    Multi-touch panel, 29
"SINUMERIK Operate Generation 2" user interface, 28
    Function key block, 70
    Touch operator controls, 71
    Virtual keyboard, 71

A

ABC keyboard, 80
Access protection
    Program block, 443
Acknowledgment button
    HT 10, 396
Actual-value display, 37
Adapter-transformed view, 321
Advanced Surface, 179
Alarm log
    display, 457
    Sorting, 459
Alarms
    Backing up log data, 455
    cancel, 456
    Displaying, 455
    Sorting, 459
Any file, 336
Archive
    Binary format, 365
    Generate in the program manager, 365
    generating in the system data, 366
    read in archive from system data, 369
    reading-in in the Program Manager, 368
Auxiliary functions
    H functions, 180
    M functions, 180
Axes
    Defined increment, 122
    Direct positioning, 125
    Referencing, 88
    Repositioning, 139
    Traversing, 122
    Variable increment, 123

B

Backing up
    Data - in the Program Manager, 365
    Data - via the system data, 366
    Parameters, 379
    Setup data, 371
Base offset, 99
Basic blocks, 135
Binary format, 365
Blank input
    Function, 244
    Parameter, 246
Block
    Search, 141
    Searching - interruption point, 144
    Searching - search pointer, 144
Block search
    Mode, 146
    Program interruption, 144
    Search pointer, 144
    Search target parameters, 145, 146
    Search target specification, 143
    Using, 141

C

Camera
    Streaming, 61
Channel switchover, 93
Checking the loading, 377
Cleaning mode, 60
Coarse and fine offsets, 99
Code carrier connection, 284
Collision avoidance, 255
    Displaying the machine model (kinematic chains), 257
    Machine operating area, 258
    Settings, 258
Configured stop, 149
Configuring
    Energy-saving profiles, 416, 419
Context-sensitive online help, 62
Coordinate system
    Switching over, 95

Creating
Multitool, 313
Program block, 160
Cross-references, 452
displaying operands, 452
Ctrl-Energy
Comparing usage values, 413
Displaying consumption values, 412
Displaying measurement curves, 411
Energy analysis, 405, 406
Energy-saving profiles, 422
Functions, 403
Measuring the energy consumption, 409
Saved measuring curves, 413
Cycles
Current levels, 241
Screen forms, 241

D

Data block (SB2), 132
Delete
Multitool, 317
Deleting
Directory, 348
Program, 348
Determine demand, 375
Dictionary
Importing, 54
Directory
Copying, 346
Creating, 333
Deleting, 348
Highlight, 344
Inserting, 346
Naming convention, 333
Properties, 349
Selecting, 344
Display Manager
Operator controls, 83
DRF (handwheel offset), 148
Drives
Logical drive, 351
Setting up, 351
DRY (dry run feed), 148
Duplo number, (See replacement tool number)
DXF file
Changing the section, 193
Cleaning, 191
close, 191
Delete Element, 197
Delete range, 197
Increasing / decreasing the section, 192
Machining plane, 196
Open, 191
Rotating the drawing, 193
Saving, 197
Select contour and accept, 199
Select the machining range, 196
Snap radius, 195
Specifying a reference point, 198
Zero point, 198
DXF-Reader, 190

E

Edges
Managing, 280
Editor
Calling, 154
Settings, 162
EES (execution from external storage), 354
Energy analysis
Details, 406
Displaying, 406
Long time measurement, 413
Energy consumption
Displaying, 405
Measuring, 409
Energy-saving profiles
Edit, 416, 419
operating, 422
EXTCALL call, 361

F

Feed data
Actual value window, 40
Finger gestures, 66
FTP drive, 328
Function
REF POINT, 91
REPOS, 91
Single block, 92
TEACH IN, 92
Function key block
"SINUMERIK Operate Generation 2" user interface, 70

G

G code program
Blank input, 244

G Code program
Creating, 335
G functions
Display all G groups, 179
Displaying selected G groups, 177
Mold making, 179
Global R parameters, 168
Global user variables, 171
Gloves, 65

H

Handheld terminal 10, 395
Handwheel
Assigning, 110
Highlight
Directory, 344
Program, 344
HT 10
Acknowledgment button, 396
Overview, 395
Touch Panel, 401
User menu, 398
Virtual keyboard, 400
HT 8
Touch Panel, 401

I

IME
Chinese characters, 51
Korean characters, 55
Interruption point
Approach, 144

J

Job list, 337

K

Key combinations - Simulation
Changing the section, 232
Enlarging/reducing a graphic, 230
Feedrate, 229
Moving the graphic, 231
Override, 228
Rotating the view, 231
Single block mode, 229

L

Ladder editor
Editing the PLC user program, 425
Layer selection, 191
Loading
Multitool, 317
Local drive, 326
Creating an NC directory, 326
Local variable tables, 441
Logbook
Delete entries, 472
Displaying, 471
Edit the address data, 471
Entry search, 473
Making an entry, 472
Output, 469
Overview, 471
Long time measurement
Energy analysis, 413

M

Machine control panel
in the sidescreen, 79
Operator controls, 29
Machine model, 255
Machine-specific information, 469
Machining times
Delete, 166
Display, 237
Display in block display, 41, 134
Magazine
Deleting tools, 298
Loading tools, 298
Open, 294
Positioning, 296
Relocating tools, 298
Selecting, 283
Unloading tools, 298
Magazine management, 263
Manual mode, 117
Positioning axes, 125
Settings, 127
Spindle, 120
T, S, M windows, 118
Tool, 119
Traversing axes, 122
Unit of measurement, 118
Manual retraction, 126

MDI
Deleting a program, 115
Executing a program, 114
Loading a program, 112
Saving a program, 113
Messages
Displaying, 458
Sorting, 459
Mode groups, 93
Mold making view
adapting, 204
Changing a graphic, 206
Changing the section, 208
Editing a program block, 205
Programs, 202
Searching for program blocks, 206
Starting, 204
MRD (Measuring Result Display), 149
Multi-channel view, 249
"Machine" operating area, 250
Settings, 253
Multiple selection, 311
Multitool, 312
Creating, 313
Delete, 317
Equipping tools, 315
Loading, 317
Parameters in the tool list, 312
Positioning, 320
Reactivating, 318
Relocating, 319
Removing tools, 316
Unloading, 318
Multitouch panel
Widescreen format, 73
Multi-touch panel
SINUMERIK Operate Gen. 2, 29

N

Navigation bar
Sidescreen, 73
NC memory, 325
NC/PLC variables
Changing, 431, 463
Displaying, 428, 460
Network
displaying symbol identifiers, 449
editing, 447
Network properties, 448
Number of teeth, 276

O

Online help
Context-sensitive, 62
Opening tool data, 376
Operands
Cross-references, 452
inserting, 447
Operating areas, 27
Changing, 43
Selecting, 43
Operating mode
AUTO, 92
Changing, 43
JOG, 91, 117
MDI, 92
REPOS, 91
Operation
deleting, 447
inserting, 447
Operator controls
Display Manager, 83
Machine control panel, 29

P

Pages, 73
Panel
cleaning, 60
Parameter
Calculating, 45
Changing, 45
Entering, 45
Parameters
Backing up, 379
Pitch, 275
PLC properties, 426
PLC signals
displaying, 434
editing, 434
PLC user program
Absolute address, 439
Diagnostics, 425, 441
Enlarging/reducing the ladder diagram, 439
Ladder editor, 425
loading, 426
Resetting processing times, 426
Search function, 454
Shortcuts, 437

Symbolic address, 439
User interface, 436
PLC user programs
editing, 444
Positioning
Multitool, 320
Preview
Program, 343
Processing times
reset in the PLC user program, 426
Program
Closing, 329
Copying, 346
Correcting, 138
Creating with cycle support, 243
Deleting, 348
Editing, 154
Executing, 331
Highlight, 344
Inserting, 346
Managing, 323
Naming convention, 333
Open, 329
opening additional programs, 161
Preview, 343
Properties, 349
Renumbering blocks, 159
Replacing text, 156
Running-in, 132
Searching for a program position, 154
Selecting, 131, 344
Teach-in, 385
Program block
Access protection, 443
adding, 442
Copying and inserting, 158
creating, 440, 442
Current, 41, 134
Delete, 158
deleting, 447
editing, 440, 442, 444, 445
Information, 435, 440
insert, 157
Numbering, 158, 159
opening in the window, 443
Search, 154
Selecting, 158
Program blocks, 160
Program control
activating, 149
Modes of operation, 148
Program correction, 138
Program level, 136
Program list, 338
Program Manager, 323
Displaying HTML documents, 358
Displaying PDF documents, 358
Searching for directories and files, 341
Program runtime, 185
Program status, 438
Program views
G code, 236
Programmed stop 1, 148
Programmed stop 2, 148
Properties
Directory, 349
Program, 349
Protection levels
Softkeys, 58
PRT (no axis motion), 148

Q

Quantity, 290

R

R parameters, 169
Reactivating
Multitool, 318
Record tools, 375
Reference, 88
Relocating
Multitool, 319
Remote access
Allowing, 475
Setting, 474
Remote diagnostics, 474
Exit, 477
Requesting, 476
Replacement tool number, 275
Repositioning, 139
Retract, 126
RG0 (reduced rapid traverse), 148
R-parameters
Backing up, 379

S

Save
Parameters, 379
Setup data, 371
SB (single blocks), 149

SB1, 132
SB2, 132
SB3, 132
Screen layout, 69
Screenshots
  Copying, 467
  Creating, 467
  Open, 467
Search
  in the Program Manager, 341
Search function
  PLC user program, 454
Search mode, 146
Search pointer, 144
Searching
  Logbook entry, 473
Select, 311
Selecting
  Directory, 344
  Program, 344
Setting actual values, (See setting work offsets)
Settings
  Collision avoidance, 258
  Editor, 162
  For automatic operation, 187
  For manual operation, 127
  Multi-channel view, 253
  Teach-in, 394
  Tool lists, 321
Setup data
  Backing up, 371
  reading in, 373
Sidescreen
  ABC keyboard, 79
  MCP, 79
  Navigation bar, 73
  Operator controls, 73
  Overview, 73
  Pages, 79
  Requirements, 73
  Show, 75
  Standard widgets, 75
  Widgets, 73
Simulation, 211
  Alarm display, 234
  Blank, 227
  Canceling, 219
  Changing the feedrate, 228
  Changing the graphic section, 232
  Moving the graphic, 231
  Non-modal, 229
  Program control, 228
  Rotating the graphic, 231
  Showing and hiding the path display, 227
  Start, 219
  stopping, 219
  Views, 224
Simultaneous recording, 213
  Before machining, 221
  Changing the graphic section, 232
  Moving the graphic, 231
  Rotating the graphic, 231
Single block
  Coarse (SB1), 132
  Fine (SB3), 132
SINUMERIK Operate Gen. 2
  Screen layout, 69
Skip blocks, 151
SKP (skip blocks), 149
Spindle data
  Actual value window, 40
Spindle speed limitation, 108
Standard widget
  Actual values, 76
  Alarms, 76
  Axle load, 77
  Camera, 79
  Service life, 78
  Tool, 77
  Variables, 77
  Zero point, 76
Startup, 87
Status display, 35
Switching over
  Coordinate system, 95
Symbol tables, 451
Symbolic identifiers
  displaying in the network, 449
Synchronization commands
  Display, 238
Synchronized actions
  Displaying status, 183
System data
  Displaying HTML documents, 358
  Displaying PDF documents, 358

T

Teach-in, 385
  Changing blocks, 388
  Circle intermediate position CIP, 391
  Continuous-path mode, 393
  Deleting blocks, 389
  General sequence, 385

Inserting a position, 391
Inserting blocks, 388
Motion type, 392
Parameter, 392
Rapid traverse G0, 391
Selecting, 387
Selecting a block, 389
Settings, 394
Traversing block G1, 391
Templates
Creating, 340
Storage locations, 340
Tip angle, 275
Tool
Change type, 304
Creating, 277
delete, 281
Details, 299
Dimensioning, 267
Loading, 282
Multiple edges, 280
Reactivating, 291
Relocating, 296
Unloading, 282
Tool data
Actual value window, 39
Backing up, 371
reading in, 373
Tool life, 290
Tool list, 275
Tool lists
Settings, 321
Tool management, 261
List filtering, 307
Sorting lists, 305
Tool parameters, 267
Tool types, 264
Tool wear, 289
Tool wear list
Open, 289
Touch operator controls
Cancel alarms, 71
Changing the channel, 71
Touch Panel
calibration, 401
Transformed view, 321
TTD file
Open, 376

U

Unit of measurement
switching, 95
Unloading
Multitool, 318
USB drive, 327
User agreement, 89
User variables, 167
Activating, 175
Backing up, 379
Channel GUD, 172
Defining, 175
Global GUD, 171, 175
Global R parameters, 168
Local LUD, 173
Program PUD, 174
R parameters, 169
Searching, 174

V

Variable screen forms, 465
Virtual keyboard
"SINUMERIK Operate Generation 2" user interface, 71
HT 10, 400
Virtual keys
ABC keyboard, 73
MCP keys, 73

W

Wear, 290
Widgets, 73
Work offsets
Delete, 105
Overview, 99, 101
Setting, 97
Working area limitation, 107
Workpiece
creating, 334
Starting machining, 129
Stopping machining, 129, 130
Workpiece counter, 185

Z

Zero offsets
Active ZO, 100
Displaying details, 103
Settable ZO, 103
Zero point
DXF file, 198
Zero point settings
Backing up, 371
reading in, 373