SIEMENS

SINUMERIK

SINUMERIK 840D sl Grinding

Operating Manual

Preface
Fundamental safety instructions 1
Introduction 2
Multitouch operation with SINUMERIK Operate 3
Setting up the machine 4
Working in the manual mode 5
Machining the workpiece 6
Simulating machining 7
Generating a G-code program 8
Programming technology functions 9
Grinding with a B axis (only for cylindrical grinding machines) 10
Collision avoidance 11
Multi-channel view 12
Manage tools 13
Managing programs 14
Alarm, fault, and system messages 15
Teaching in a program 16
Ctrl-Energy 17
Easy XML 18
Handheld terminals for multi-touch operation 19
Appendix 20

Valid for:
SINUMERIK 840D sl / 840DE sl
Software version
CNC system software for 840D sl / 840DE sl V4.93
SINUMERIK Operate for PCU/PC V4.93

12/2019
6FC5398-0EP40-6BA2

Промэлектроника, sales@prom-elec.com

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 personnel are those who, based on their training and experience, are capable of identifying risks and avoiding potential hazards when working with these products/systems.

Proper use of Siemens products

Note the following:

⚠ WARNING
Siemens products may only be used for the applications described in the catalog and in the relevant technical documentation. 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 AG. 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 AG
Digital Industries
Postfach 48 48
90026 NÜRNBERG
GERMANY

Document order number: 6FC5398-0EP40-6BA2
℗ 12/2019 Subject to change

Copyright © Siemens AG 2015 - 2019.
All rights reserved

Preface

SINUMERIK documentation

The SINUMERIK documentation is organized into the following categories:

- General documentation/catalogs
- User documentation
- Manufacturer/service documentation

Additional information

You can find information on the following topics at the following address (https://support.industry.siemens.com/cs/de/en/view/108464614):

- Ordering documentation/overview of documentation
- Additional links to download documents
- Using documentation online (find and search in manuals/information)

If you have any questions regarding the technical documentation (e.g. suggestions, corrections), please send an e-mail to the following address (mailto:docu.motioncontrol@siemens.com).

mySupport/Documentation

At the following address (https://support.industry.siemens.com/My/ww/en/documentation), you can find information on how to create your own individual documentation based on Siemens' content, and adapt it for your own machine documentation.

Training

At the following address (http://www.siemens.com/sitrain), you can find information about SITRAIN (Siemens training on products, systems and solutions for automation and drives).

FAQs

You can find Frequently Asked Questions in the Service&Support pages under Product Support (https://support.industry.siemens.com/cs/de/en/ps/faq).

SINUMERIK

You can find information about SINUMERIK at the following address (http://www.siemens.com/sinumerik).

Target group

Target group

This documentation is intended for users of cylindrical grinding machines and grinding 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 describes the functionality of the standard scope. Extensions or changes made by the machine manufacturer are documented by the machine manufacturer.

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

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

Note regarding the General Data Protection Regulation

Siemens observes standard data protection principles, in particular the principle of privacy by design. That means that

this product does not process / store any personal data, only technical functional data (e.g. time stamps). If a user links this data with other data (e.g. a shift schedule) or stores personal data on the same storage medium (e.g. hard drive) and thus establishes a link to a person or persons, then the user is responsible for ensuring compliance with the relevant data protection regulations.

Technical Support

Country-specific telephone numbers for technical support are provided on the Internet at the following address (https://support.industry.siemens.com/sc/ww/en/sc/2090) in the "Contact" area.

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

Table of contents

Preface **3**

1 Fundamental safety instructions **15**

1.1 General safety instructions 15

1.2 Warranty and liability for application examples 16

1.3 Industrial security 17

2 Introduction **19**

2.1 Product overview 19

2.2 Operator panel fronts 20
2.2.1 Overview 20
2.2.2 Keys of the operator panel 22

2.3 Machine control panels 30
2.3.1 Overview 30
2.3.2 Controls on the machine control panel 30

2.4 User interface 34
2.4.1 Screen layout 34
2.4.2 Status display 35
2.4.3 Actual value window 37
2.4.4 T,F,S window 39
2.4.5 Current block display 40
2.4.6 Operation via softkeys and buttons 42
2.4.7 Entering or selecting parameters 43
2.4.8 Pocket calculator 45
2.4.9 Pocket calculator functions 46
2.4.10 Context menu 48
2.4.11 Changing the user interface language 48
2.4.12 Entering Chinese characters 49
2.4.12.1 Entering Asian characters 51
2.4.12.2 Editing the dictionary 52
2.4.13 Entering Korean characters 53
2.4.14 Protection levels 56
2.4.15 Cleaning mode 57
2.4.16 Display live image from a camera 58
2.4.17 Display live image from a camera 59
2.4.18 Online help in SINUMERIK Operate 60

3 Multitouch operation with SINUMERIK Operate **63**

3.1 Multitouch panels 63

3.2 Touch-sensitive user interface 64

3.3 Finger gestures 65

3.4 Multitouch user interface 68

3.4.1 Screen layout 68
3.4.2 Function key block 69
3.4.3 Further operator touch controls 69
3.4.4 Virtual keyboard 70
3.4.5 Special "tilde" character 71

3.5 Expansion with side screen 72
3.5.1 Overview 72
3.5.2 Sidescreen with standard windows 72
3.5.3 Standard widgets 74
3.5.4 "Actual value" widget 74
3.5.5 "Zero point" widget 75
3.5.6 "Alarms" widget 75
3.5.7 "NC/PLC variables" widget 75
3.5.8 "Axle load" widget 76
3.5.9 "Tool" widget 76
3.5.10 "Service life" widget 77
3.5.11 "Program runtime" widget 77
3.5.12 Widget "Camera 1" and "Camera 2" 77
3.5.13 Sidescreen with pages for the ABC keyboard and/or machine control panel 78
3.5.14 Example 1: ABC keyboard in the sidescreen 79
3.5.15 Example 2: Machine control panel in the sidescreen 80

3.6 SINUMERIK Operate Display Manager 81
3.6.1 Overview 81
3.6.2 Screen layout 82
3.6.3 Operator controls 82

4 Setting up the machine 87

4.1 Switching on and switching off 87

4.2 Approaching a reference point 88
4.2.1 Referencing axes 88
4.2.2 User agreement 89

4.3 Operating modes 91
4.3.1 Operating modes 91
4.3.2 Modes groups and channels 93
4.3.3 Channel switchover 93

4.4 Settings for the machine 95
4.4.1 Switching over the coordinate system (MCS/WCS) 95
4.4.2 Switching the unit of measurement 95
4.4.3 Setting the zero offset 97

4.5 Zero offsets 99
4.5.1 Zero offsets 99
4.5.2 Display active zero offset 100
4.5.3 Displaying the zero offset "overview" 101
4.5.4 Displaying and editing base zero offset 102
4.5.5 Displaying and editing settable zero offset 103
4.5.6 Displaying and editing seat-related fine offset 103
4.5.7 Displaying and editing details of the zero offsets 104
4.5.8 Deleting a zero offset 106
4.5.9 Deleting seat-related fine offsets 106

4.6 Measuring a tool 108
4.6.1 Cylindrical grinding 108
4.6.1.1 Overview 108
4.6.1.2 Measure grinding tool manually with the work piece reference point 108
4.6.1.3 Measure grinding tool manually with the dresser reference point 110
4.6.1.4 Measuring the dressing tool manually with the grinding tool reference point 111
4.6.2 Surface grinding 112
4.6.2.1 Overview 112
4.6.2.2 Measure grinding tool manually with the work piece reference point 113
4.6.2.3 Measure grinding tool manually with the dresser reference point 114
4.6.2.4 Measuring the dressing tool manually with the grinding tool reference point 115

4.7 Measuring the workpiece zero 118
4.7.1 Cylindrical grinding 118
4.7.1.1 Measuring the workpiece zero 118
4.7.2 Surface grinding 119
4.7.2.1 Overview 119
4.7.2.2 Setting the edge 119

4.8 Monitoring axis and spindle data 121
4.8.1 Specify working area limitations 121
4.8.2 Editing spindle data 121
4.8.3 Spindle chuck data 122
4.8.3.1 Defining spindle chuck data 122
4.8.3.2 Parameters, spindle chuck data 124
4.8.4 Entering cylinder error compensation (only rotary grinding machine) 125

4.9 Displaying setting data lists 127

4.10 Handwheel assignment 128

4.11 MDI 130
4.11.1 Loading an MDA program from the Program Manager 130
4.11.2 Saving an MDA program 131
4.11.3 Editing/executing a MDI program 132
4.11.4 Deleting an MDA program 132

5 Working in the manual mode 135

5.1 Overview 135

5.2 Selecting a tool and spindle 136
5.2.1 T,S,M window 136
5.2.2 Selecting a tool 137
5.2.3 Starting and stopping the spindle manually 138
5.2.4 Positioning the spindle 139

5.3 Traversing axes 140
5.3.1 Traversing the axes 140
5.3.2 Traverse axes by a defined increment 140
5.3.3 Traversing axes by a variable increment 141

5.4 Positioning axes 142

5.5 Default settings for manual mode 143

6 Machining the workpiece 145

6.1 Starting and stopping machining 145
6.2 Selecting a program 147
6.3 Executing a trial program run 148
6.4 Displaying the current program block 150
6.4.1 Current block display 150
6.4.2 Displaying a basic block 151
6.4.3 Display program level 152
6.5 Correcting a program 154
6.6 Repositioning axes 155
6.7 Starting execution at a specific point 156
6.7.1 Use block search 156
6.7.2 Continuing program from search target 158
6.7.3 Simple search target definition 158
6.7.4 Defining an interruption point as search target 159
6.7.5 Parameters for block search in the search pointer 159
6.7.6 Block search mode 160
6.8 Controlling the program run 162
6.8.1 Program control 162
6.8.2 Skip blocks 163
6.9 Overstore 165
6.10 Editing a program 167
6.10.1 Processing a program (editor) 167
6.10.2 Searching in programs 167
6.10.3 Replacing program text 169
6.10.4 Copying/pasting/deleting program blocks 170
6.10.5 Renumbering a program 171
6.10.6 Creating a program block 172
6.10.7 Editor settings 174
6.11 Working with DXF files 178
6.11.1 Overview 178
6.11.2 Displaying CAD drawings 179
6.11.2.1 Open a DXF file 179
6.11.2.2 Cleaning a DXF file 179
6.11.2.3 Enlarging or reducing the CAD drawing 180
6.11.2.4 Changing the section 181
6.11.2.5 Rotating the view 181
6.11.2.6 Displaying/editing information for the geometric data 182
6.11.3 Importing and editing a DXF file 183
6.11.3.1 General procedure 183
6.11.3.2 Setting the tolerance 183
6.11.3.3 Assigning the machining plane 183
6.11.3.4 Selecting the machining range / deleting the range and element 184
6.11.3.5 Saving the DXF file 185
6.11.3.6 Specifying a reference point 186
6.11.3.7 Accepting contours 186

6.12 Display and edit user variables 189
6.12.1 Global R parameters 190
6.12.2 R parameters 191
6.12.3 Displaying global user data (GUD) 192
6.12.4 Displaying channel GUDs 194
6.12.5 Displaying local user data (LUD) 195
6.12.6 Displaying program user data (PUD) 196
6.12.7 Searching for user variables 196

6.13 Display G and auxiliary functions 198
6.13.1 Selected G functions 198
6.13.2 All G functions 200
6.13.3 G functions for mold making 200
6.13.4 Auxiliary functions 201

6.14 Displaying superimpositions 203

6.15 Displaying status of synchronized actions 204

6.16 Displaying the program runtime and counting workpieces 206

6.17 Settings for the automatic mode 208

7 Simulating machining 211

7.1 Overview 211

7.2 Simulation before machining of the workpiece 215
7.2.1 Simulation before machining of the workpiece 215
7.2.2 Start simulation 215

7.3 Simultaneous recording before machining of the workpiece 217
7.3.1 Overview 217
7.3.2 Starting simultaneous recording 217

7.4 Simultaneous recording during machining of the workpiece 218

7.5 Program control during simulation 219
7.5.1 Changing the feedrate 219
7.5.2 Simulating the program block by block 220

7.6 Different views of a workpiece 221
7.6.1 Overview 221
7.6.2 Side view 221
7.6.3 Dressing view 221
7.6.4 3D view 222
7.6.5 Face view - for cylindrical grinding 222
7.6.6 Further side views - for surface grinding 222
7.6.7 Machine space (with "collision avoidance" option) 223

7.7 Editing the simulation display 224
7.7.1 Deleting the tool path 224

7.8 Editing and adapting a simulation graphic 225
7.8.1 Enlarging or reducing the graphical representation 225
7.8.2 Panning a graphical representation 226
7.8.3 Rotating the graphical representation 226
7.8.4 Modifying the viewport 227

8 Generating a G-code program 229

8.1 Graphical programming 229

8.2 Program views 230

8.3 Program structure 232

8.4 Basic principles 233
8.4.1 Machining planes 233
8.4.2 Programming a tool (T) 233

8.5 Generating a G code program 235

8.6 Selection of the cycles via softkey 236

9 Programming technology functions 237

9.1 Programming contours 237
9.1.1 Representation of the contour 237
9.1.2 Creating a new contour 238
9.1.3 Creating contour elements 240
9.1.3.1 Entering a contour element 242
9.1.3.2 Cylindrical grinding 243
9.1.3.3 Surface grinding 244
9.1.4 Changing the contour 247
9.1.4.1 Overview 247
9.1.4.2 Modifying a contour element 247
9.1.5 Contour call (CYCLE62) 248
9.1.5.1 Function 248
9.1.5.2 Calling the cycle 249
9.1.5.3 Parameter 249

9.2 Set dresser coordinate system - CYCLE435 250
9.2.1 Note regarding the dressing tool position 250
9.2.2 Function 250

9.3 Form-truing (CYCLE495) 252

9.4 Oscillating cycles (CYCLE4071 ... CYCLE4079) 256
9.4.1 Note regarding oscillating cycles 256
9.4.2 CYCLE4071 - longitudinal grinding with infeed at the reversal point 256
9.4.3 CYCLE4072 - longitudinal grinding with infeed at the reversal point and cancel signal 258
9.4.4 CYCLE4073 - longitudinal grinding with continuous infeed 262
9.4.5 CYCLE4074 - longitudinal grinding with continuous infeed and cancel signal 263
9.4.6 CYCLE4075 - surface grinding with infeed at the reversal point 266
9.4.7 CYCLE4077 - surface grinding with infeed at the reversal point and cancel signal 269
9.4.8 CYCLE4078 - surface grinding with continuous infeed 273
9.4.9 CYCLE4079 - surface grinding with intermittent infeed 275

9.5 Aligning a grinding wheel (CYCLE400) 279
9.5.1 Function 279
9.5.2 Calling the cycle 279

10 Grinding with a B axis (only for cylindrical grinding machines) 281

10.1 Overview 281

10.2 T, S, M window for set-up B axis 283

10.3 Measuring in JOG 285
10.3.1 Grinding wheel alignment for grinding 285
10.3.2 Manually measuring the grinding tool (with B axis) 285
10.3.3 Manually measuring the dresser (with B axis) 287
10.3.4 Calibrating the swivel axis 288

11 Collision avoidance 291

11.1 Activating collision avoidance 293
11.2 Set collision avoidance 294

12 Multi-channel view 297

12.1 Multi-channel view 297
12.2 Multi-channel view in the "Machine" operating area 298
12.3 Multi-channel view for large operator panels 301
12.4 Setting the multi-channel view 303

13 Manage tools 305

13.1 Lists for the tool management 305
13.2 Magazine management 307
13.3 Tool types 308
13.4 Tool dimensioning 310
13.5 Tool list 312
13.5.1 Tool list 312
13.5.2 Additional data 314
13.5.3 Creating a new tool 315
13.5.4 Measuring a tool - tool list 317
13.5.5 Managing several cutting edges 317
13.5.6 Deleting a tool 318
13.5.7 Loading and unloading tools 319
13.5.8 Selecting a magazine 320
13.5.9 Code carrier connection 321
13.5.9.1 Overview 321
13.5.9.2 Managing tools on a code carrier 322
13.5.10 Managing a tool in a file 323
13.6 Tool wear 326
13.6.1 Tool wear 326
13.6.2 Reactivating a tool 329
13.7 Tool data OEM 331
13.8 Magazine 333
13.8.1 Positioning a magazine 334
13.8.2 Relocating a tool 335
13.8.3 Deleting/unloading/loading/relocating all tools 336
13.9 Sorting tool management lists 337
13.10 Filtering the tool management lists 338
13.11 Specific search in the tool management lists 340

13.12 Tool details 342
13.12.1 Displaying tool details 342
13.12.2 Tool data 342
13.12.3 Grinding data 343
13.12.4 Cutting edge data 344
13.12.5 Monitoring data 345
13.13 Changing a tool type 346
13.14 Working with Multitool 347
13.14.1 Tool list for multitool 347
13.14.2 Create multitool 348
13.14.3 Equipping multitool with tools 349
13.14.4 Removing a tool from multitool 350
13.14.5 Deleting multitool 351
13.14.6 Loading and unloading multitool 351
13.14.7 Positioning multitool 352
13.14.8 Relocating a multitool 352
13.14.9 Reactivating a multitool 353
13.15 Settings for tool lists 355
14 Managing programs 357
14.1 Overview 357
14.1.1 NC memory 359
14.1.2 Local drive 360
14.1.3 Creating an NC directory on the local drive 360
14.1.4 USB drives 361
14.1.5 FTP drive 362
14.2 Opening and closing the program 363
14.3 Executing a program 365
14.4 Creating a directory / program / job list 367
14.4.1 File and directory names 367
14.4.2 Creating a new directory 367
14.4.3 Creating a new workpiece 368
14.4.4 Creating a new G code program 369
14.4.5 Creating a new dressing program 369
14.4.6 Storing any new file 370
14.4.7 Creating a job list 371
14.4.8 Creating a program list 373
14.5 Creating templates 374
14.6 Searching directories and files 375
14.7 Displaying the program in the Preview 377
14.8 Selecting several directories/programs 378
14.9 Copying and pasting a directory/program 380
14.10 Deleting a program/directory 382
14.11 Changing file and directory properties 383
14.12 Creating drives 384

14.12.1 Overview 384
14.12.2 Setting up drives 384
14.13 Viewing PDF documents 391
14.14 EXTCALL 393
14.15 Execution from external storage (EES) 395
14.16 Backing up data 396
14.16.1 Generating an archive in the Program Manager 396
14.16.2 Generating an archive via the system data 397
14.16.3 Reading in an archive in the Program Manager 399
14.16.4 Read in archive from system data 400
14.17 Setup data 401
14.17.1 Backing up setup data 401
14.17.2 Reading-in set-up data 402
14.18 Backing up parameters 405
14.19 V24 408
14.19.1 Reading-in and reading-out archives via a serial interface 408
14.19.2 Setting V24 in the program manager 410
15 Alarm, fault, and system messages 413
15.1 Displaying messages 413
15.2 Displaying alarms 414
15.3 Displaying an alarm log 417
15.4 Sorting, alarms, faults and messages 418
15.5 Creating screenshots 419
15.6 PLC and NC variables 420
15.6.1 Displaying and editing PLC and NC variables 420
15.6.2 Saving and loading screen forms 424
15.7 Version 425
15.7.1 Displaying version data 425
15.7.2 Save information 426
15.8 Logbook 427
15.8.1 Overview 427
15.8.2 Displaying and editing the logbook 427
15.8.3 Making a logbook entry 428
15.9 Remote diagnostics 430
15.9.1 Setting remote access 430
15.9.2 Permit modem 431
15.9.3 Request remote diagnostics 432
15.9.4 Exit remote diagnostics 433
16 Teaching in a program 435
16.1 Overview 435
16.2 Select teach in mode 437
16.3 Processing a program 438

16.3.1 Inserting a block 438
16.3.2 Editing a block 438
16.3.3 Selecting a block 439
16.3.4 Deleting a block 439
16.4 Teach sets 441
16.4.1 Input parameters for teach-in blocks 442
16.5 Settings for teach-in 444
17 Ctrl-Energy 445
17.1 Functions 445
17.2 Ctrl-E analysis 446
17.2.1 Displaying energy consumption 446
17.2.2 Displaying the energy analyses 447
17.2.3 Measuring and saving the energy consumption 448
17.2.4 Tracking measurements 449
17.2.5 Tracking usage values 449
17.2.6 Comparing usage values 450
17.2.7 Long-term measurement of the energy consumption 451
17.3 Ctrl-E profiles 452
17.3.1 Using the energy-saving profile 452
18 Easy XML 455
18.1 Easy XML 455
18.2 SINUMERIK Integrate Run MyScreens 457
19 Handheld terminals for multi-touch operation 459
19.1 HT 8 459
19.1.1 Overview 459
19.1.2 Traversing keys 461
19.1.3 Machine control panel menu 462
19.1.4 Virtual keyboard 464
19.2 HT 10 466
19.2.1 HT 10: Overview 466
19.2.2 Machine control panel menu 468
19.2.3 Virtual keyboard 470
19.3 Calibrating the touch panel 472
20 Appendix 473
20.1 Documentation overview SINUMERIK 840D sl 473
Index 475

Fundamental safety instructions

1

1.1 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.

1.2 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 specific customer solutions, but are only intended to provide support for 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.

1.3 Industrial security

Note

Industrial security

Siemens provides products and solutions with industrial security 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 security concept. Products and solutions from Siemens 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. using firewalls and/or network segmentation) are in place.

For additional information on industrial security measures that can be implemented, please visit:

Industrial security (https://www.siemens.com/industrialsecurity)

Siemens' products and solutions undergo continuous development to make them more secure. Siemens strongly recommends that product updates are applied as soon as they become available, and that only 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 Security RSS Feed at:

Industrial security (https://www.siemens.com/industrialsecurity)

Further information is provided on the Internet:

Industrial Security Configuration Manual (https://support.industry.siemens.com/cs/ww/en/view/108862708)

⚠ 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 holistic, state-of-the-art industrial security concept for the installation or machine.
- Make sure that you include all installed products into the holistic industrial security concept.
- Protect files stored on exchangeable storage media from malicious software by with suitable protection measures, e.g. virus scanners.
- On completion of commissioning, check all security-related settings.
- Protect the drive against unauthorized changes by activating the "Know-how protection" converter function.

Introduction

2

2.1 Product overview

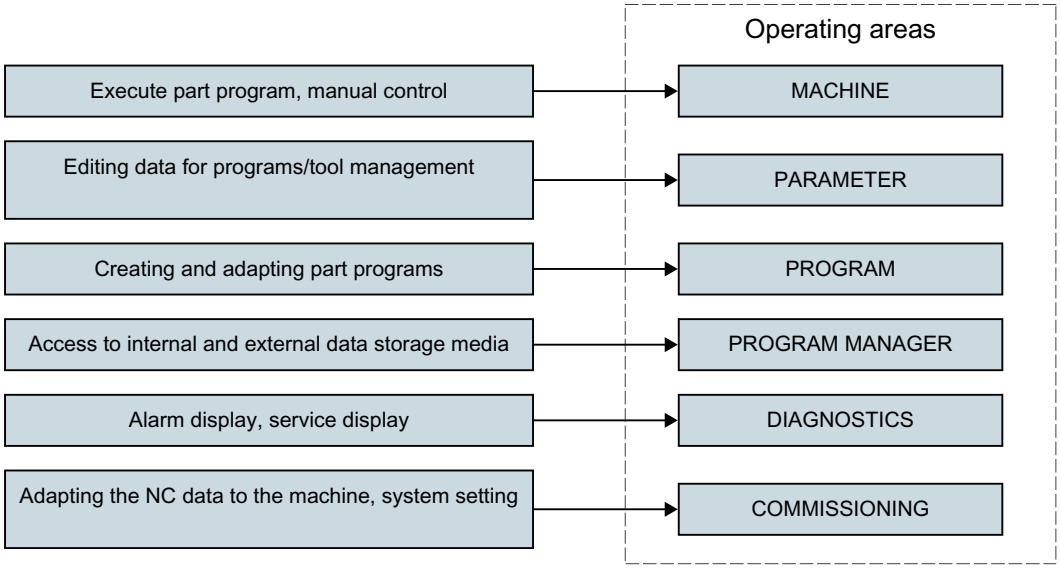
The SINUMERIK control system is a CNC (Computerized Numerical Control) for machine tools.

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:



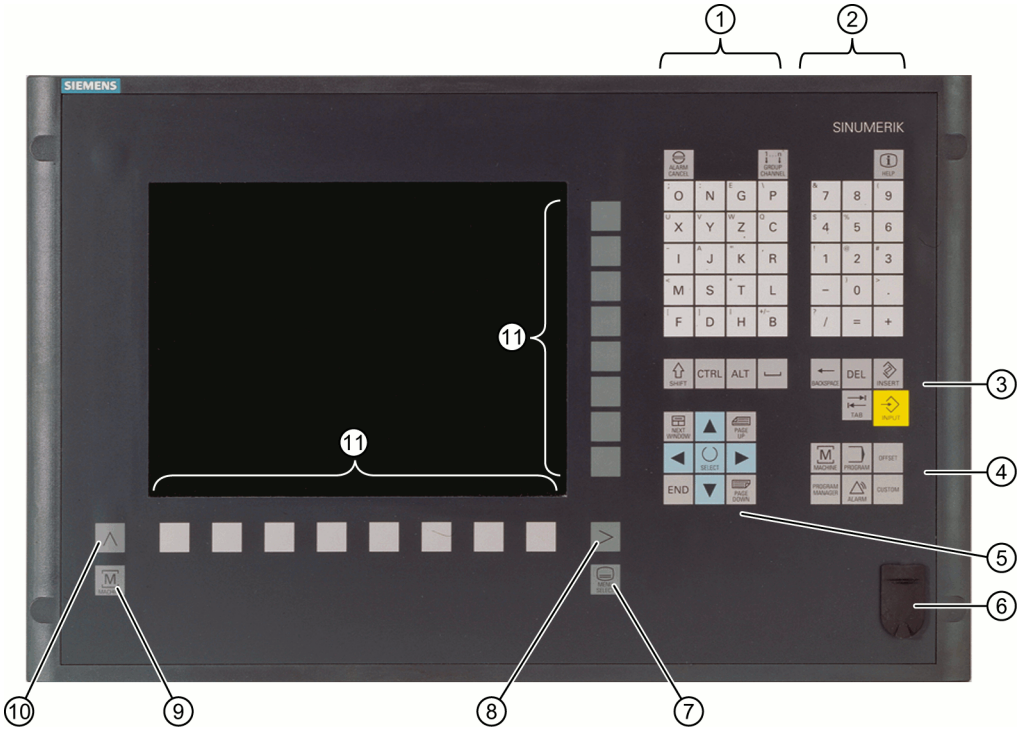
2.2 Operator panel fronts

2.2.1 Overview

The display (screen) and operation (e.g. hardkeys and softkeys) of the SINUMERIK Operate user interface are via the operator panel front.

Operator controls and indicators

In this example, the OP 010 operator panel front is used to illustrate the components that are available for operating the controller and machine tool.



① Alphabetic key group

With the <Shift> key pressed, you activate the special characters on keys with double assignments, and write in the uppercase.

Note: Depending on the particular configuration of your control system, uppercase letters are always written

② Numerical key group

With the <Shift> key pressed, you activate the special characters on keys with double assignments.

③ Control key group

④ Hotkey group

⑤ Cursor key group

⑥ USB interface

⑦ Menu select key

⑧ Menu forward button

⑨ Machine area button

⑩ Menu back key

⑪ Softkeys

Further information

Further information about the OP 010 and other usable operator panel fronts can be found at:

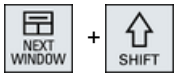
- Operating Components Equipment Manual - Handheld Units (https://support.industry.siemens.com/cs/ww/en/view/109736210)
- Equipment Manual OP 010 (https://support.industry.siemens.com/cs/ww/en/view/109759204)
- Equipment Manual OP 012 (https://support.industry.siemens.com/cs/ww/en/view/109741627)
- Equipment Manual OP 015A (https://support.industry.siemens.com/cs/ww/en/view/109748600)
- Operating Components Equipment Manual - TCU 30.3 (https://support.industry.siemens.com/cs/ww/en/view/109749929)
- HT 8 (https://support.industry.siemens.com/cs/ww/en/view/109763514)

2.2.2 Keys of the operator panel

The following keys and key combinations are available for operation of the control and the machine tool.

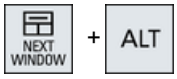
Keys and key combinations

Key	Function
	<ALARM CANCEL> Cancels alarms and messages that are marked with this symbol.
	<CHANNEL> Advances for several channels.
	<HELP> Calls the context-sensitive online help for the selected window.
	<NEXT WINDOW> * • Toggles between the windows. • For a multi-channel view or for a multi-channel functionality, switches within a channel gap between the upper and lower window. • Selects the first entry in selection lists and in selection fields. • Moves the cursor to the beginning of a text. * on USB keyboards use the <Home> or <Pos 1> key



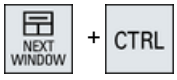
<NEXT WINDOW> + <SHIFT>

- Selects the first entry in selection lists and in selection fields.
- Moves the cursor to the beginning of a text.
- Selects a contiguous selection from the current cursor position up to the target position.
- Selects a contiguous selection from the current cursor position up to the beginning of a program block.



<NEXT WINDOW> + <ALT>

- Moves the cursor to the first object.
- Moves the cursor to the first column of a table row.
- Moves the cursor to the beginning of a program block.



<NEXT WINDOW> + <CTRL>

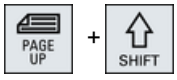
- Moves the cursor to the beginning of a program.
- Moves the cursor to the first row of the current column.

<NEXT WINDOW> + <CTRL> + <SHIFT>

- Moves the cursor to the beginning of a program.
- Moves the cursor to the first row of the current column.
- Selects a contiguous selection from the current cursor position up to the target position.
- Selects a contiguous selection from the current cursor position up to the beginning of the program.

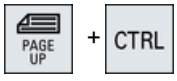
<PAGE UP>

Scrolls upwards by one page in a window.



<PAGE UP> + <SHIFT>

In the program manager and in the program editor from the cursor position, selects directories or program blocks up to the beginning of the window.



<PAGE UP> + <CTRL>

Positions the cursor to the topmost line of a window.

<PAGE DOWN>

Scrolls downwards by one page in a window.

<PAGE DOWN> + <SHIFT>

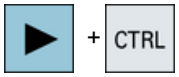
In the program manager and in the program editor, from the cursor position, selects directories or program blocks up to the end of the window.

<PAGE DOWN> + <CTRL>

Positions the cursor to the lowest line of a window.

<Cursor right>

- Editing box
  Opens a directory or program (e.g. cycle) in the editor.
- Navigation
  Moves the cursor further to the right by one character.

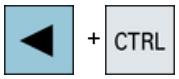


<Cursor right> + <CTRL>

- Editing box
  Moves the cursor further to the right by one word.
- Navigation
  Moves the cursor in a table to the next cell to the right.

<Cursor left>

- Editing box
  Closes a directory or program (e.g. cycle) in the program editor. If you have made changes, then these are accepted.
- Navigation
  Moves the cursor further to the left by one character.

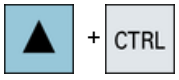


<Cursor left> + <CTRL>

- Editing box
  Moves the cursor further to the left by one word.
- Navigation
  Moves the cursor in a table to the next cell to the left.

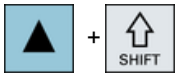
<Cursor up>

- Editing box
  Moves the cursor into the next upper field.
- Navigation
  - Moves the cursor in a table to the next cell upwards.
  - Moves the cursor upwards in a menu screen.



<Cursor up> + <Ctrl>

- Moves the cursor in a table to the beginning of the table.
- Moves the cursor to the beginning of a window.

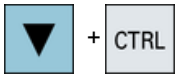


<Cursor up> + <SHIFT>

In the program manager and in the program editor, selects a contiguous selection of directories and program blocks.

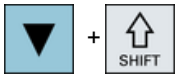
<Cursor down>

- Editing box
  Moves the cursor downwards.
- Navigation
  - Moves the cursor in a table to the next cell downwards.
  - Moves the cursor in a window downwards.



<Cursor down> + <CTRL>

- Navigation
  - Moves the cursor in a table to the end of the table.
  - Moves the cursor to the end of a window.
- Simulation
  Reduces the override.



<Cursor down> + <SHIFT>

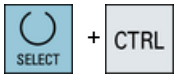
In the program manager and in the program editor, selects a contiguous selection of directories and program blocks.

<SELECT>

Switches between several specified options in selection lists and in selection boxes.

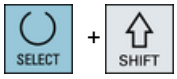
Activates checkboxes.

In the program editor and in the program manager, selects a program block or a program.



<SELECT> + <CTRL>

When selecting table rows, switches between selected and not selected.



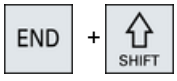
<SELECT> + <SHIFT>

Selects in selection lists and in selection boxes the previous entry or the last entry.

<END>

Moves the cursor to the last entry field in a window, to the end of a table or a program block.

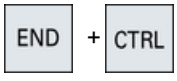
Selects the last entry in selection lists and in selection boxes.



<END> + <SHIFT>

Moves the cursor to the last entry.

Selects a contiguous selection from the cursor position up to the end of a program block.



<END> + <CTRL>

Moves the cursor to the last entry in the last line of the actual column or to the end of a program.

<END> + <CTRL> + <SHIFT>

Moves the cursor to the last entry in the last line of the actual column or to the end of a program.

Selects a contiguous selection from the cursor position up to the end of a program block.

<BACKSPACE>

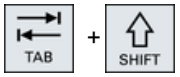
- Editing box
  Deletes a character selected to the left of the cursor.
- Navigation
  Deletes all of the selected characters to the left of the cursor.

<BACKSPACE> + <CTRL>

- Editing box
  Deletes a word selected to the left of the cursor.
- Navigation
  Deletes all of the selected characters to the left of the cursor.

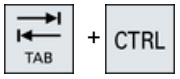
<TAB>

- In the program editor, indents the cursor by one character.
- In the program manager, moves the cursor to the next entry to the right.



<TAB> + <SHIFT>

- In the program editor, indents the cursor by one character.
- In the program manager, moves the cursor to the next entry to the left.



<TAB> + <CTRL>

- In the program editor, indents the cursor by one character.
- In the program manager, moves the cursor to the next entry to the right.

<Tab> + <Ctrl> + <Shift>

- In the program editor, indents the cursor by one character.
- In the program manager, moves the cursor to the next entry to the left.

<CTRL> + <A>

In the actual window, selects all entries (only in the program editor and program manager).

<CTRL> + <C>

Copies the selected content.

<CTRL> + <E>

Calls the "Ctrl Energy" function.

<CTRL> + <F>

Opens the search dialog in the machine data and setting data lists, when loading and saving in the MDI editor as well as in the program manager and in the system data.

<CTRL> + <G>

- Switches in the program editor for ShopMill or ShopTurn programs between the work plan and the graphic view.
- Switches in the parameter screen between the help display and the graphic view.

<CTRL> + <I>

Calculates the program runtime up to or from the selected set/block and displays a graphic representation of the times.

<CTRL> + <L>

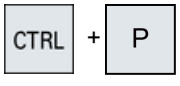
Scrolls the actual user interface through all installed languages one after the other.

<CTRL> + <SHIFT> + <L>

Scrolls the actual user interface through all installed languages in the inverse sequence.

<CTRL> + <M>

Selects the maximum feedrate of 120% during the simulation.



<CTRL> + <P>

Generates a screenshot from the actual user interface and saves it as file.

CTRL + S

<CTRL> + <S>

Switches the single block in or out in the simulation.

CTRL + V

<CTRL> + <V>

- Pastes text from the clipboard at the actual cursor position.
- Pastes text from the clipboard at the position of a selected text.

CTRL + X

<CTRL> + <X>

Cuts out the selected text. The text is located in the clipboard.

CTRL + Y

<CTRL> + <Y>

Reactivates changes that were undone (only in the program editor).

CTRL + Z

<CTRL> + <Z>

Undoes the last action (only in the program editor).

CTRL + ALT + C

<CTRL> + <ALT> + <C>

Creates a complete standard archive (.ARC) on an external data carrier (USB-FlashDrive)

Note:

The complete backup via this key combination is only suitable for diagnostic purposes.

Note:

Please observe the information provided by the machine manufacturer.

CTRL + ALT + S

<CTRL> + <ALT> + <S>

Creates a complete Easy Archive (.ARD) on an external data carrier (USB-FlashDrive)

Note:

The complete backup (.ARC) via this key combination is only suitable for diagnostic purposes.

Note:

Please observe the information provided by the machine manufacturer.

CTRL + ALT + D

<CTRL> + <ALT> + <D>

Backs up the log files on the USB-FlashDrive. If a USB-FlashDrive is not inserted, then the files are backed-up in the manufacturer's area of the CF card.

SHIFT + ALT + D

<SHIFT> + <ALT> + <D>

Backs up the log files on the USB-FlashDrive. If a USB-FlashDrive is not inserted, then the files are backed-up in the manufacturer's area of the CF card.

SHIFT + ALT + T

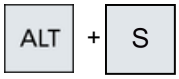
<SHIFT> + <ALT> + <T>

Starts "HMI Trace".

SHIFT + ALT + T

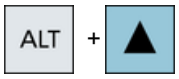
<SHIFT> + <ALT> + <T>

Exits "HMI Trace".



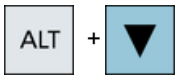
<ALT> + <S>

Opens the editor to enter Asian characters.



<ALT> + <Cursor up>

Moves the block start or block end up in the editor.

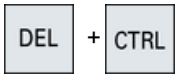


<ALT> + <Cursor down>

Moves the block start or block end down in the editor.

<DEL>

- Editing box
  Deletes the first character to the right of the cursor.
- Navigation
  Deletes all characters.



<DEL> + <CTRL>

- Editing box
  Deletes the first word to the right of the cursor.
- Navigation
  Deletes all characters.

<Spacebar>

- Editing box
  Inserts a space.
- Switches between several specified options in selection lists and in selection boxes.

<Plus>

- Opens a directory which contains the element.
- Increases the size of the graphic view for simulation and traces.

<Minus>

- Closes a directory which contains the element.
- Reduces the size of the graphic view for simulation and traces.

<Equals>

Opens the calculator in the entry fields.

<Asterisk>

Opens a directory with all of the subdirectories.

<Tilde>

Changes the sign of a number between plus and minus.

<INSERT>

- Opens an editing window in the insert mode. Pressing the key again, exits the window and the entries are undone.
- Opens a selection box and shows the selection possibilities.
- In the machining step program, enters an empty line for G code.
- Changes into the double editor or into the multi-channel view from the edit mode into the operating mode. You can return to the edit mode by pressing the key again.

<INSERT> + <SHIFT>

For G code programming, for a cycle call activates or deactivates the edit mode.

<INPUT>

- Completes input of a value in the entry field.
- Opens a directory or a program.
- Inserts an empty program block if the cursor is positioned at the end of a program block.
- Inserts a character to select a new line and the program block is split up into two parts.
- In the G code, inserts a new line after the program block.
- In the machining step program, inserts a new line for G code e
- Changes into the double editor or into the multi-channel view from the edit mode into the operating mode. You can return to the edit mode by pressing the key again.

<ALARM> - only OP 010 and OP 010C

Calls the "Diagnosis" operating area.

<PROGRAM> - only OP 010 and OP 010C

Calls the "Program Manager" operating area.

<OFFSET> - only OP 010 and OP 010C

Calls the "Parameter" operating area.

<PROGRAM MANAGER> - only OP 010 and OP 010C

Calls the "Program Manager" operating area.

Menu forward key

Advances in the extended horizontal softkey bar.

Menu back key

Returns to the higher-level menu.

<MACHINE>

Calls the "Machine" operating area.

<MENU SELECT>

Calls the main menu to select the operating area.

2.3 Machine control panels

2.3.1 Overview

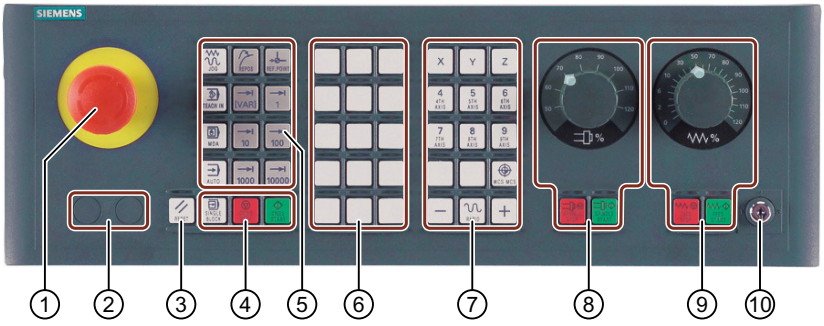
The machine tool can be equipped with a machine control panel by Siemens or with a specific machine control panel from the machine manufacturer.

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.

2.3.2 Controls on the machine control panel

In this example, the MCP 483C IE machine control panel is used to illustrate the operator controls and displays of a Siemens machine control panel.

Overview



① EMERGENCY STOP button
② Installation locations for control devices (d = 16 mm)
③ RESET
④ Program control
⑤ Operating modes, machine functions
⑥ User keys T1 to T15
⑦ Traversing axes with rapid traverse override and coordinate switchover
⑧ Spindle control with override switch
⑨ Feed control with override switch
⑩ Keyswitch (four positions)

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.

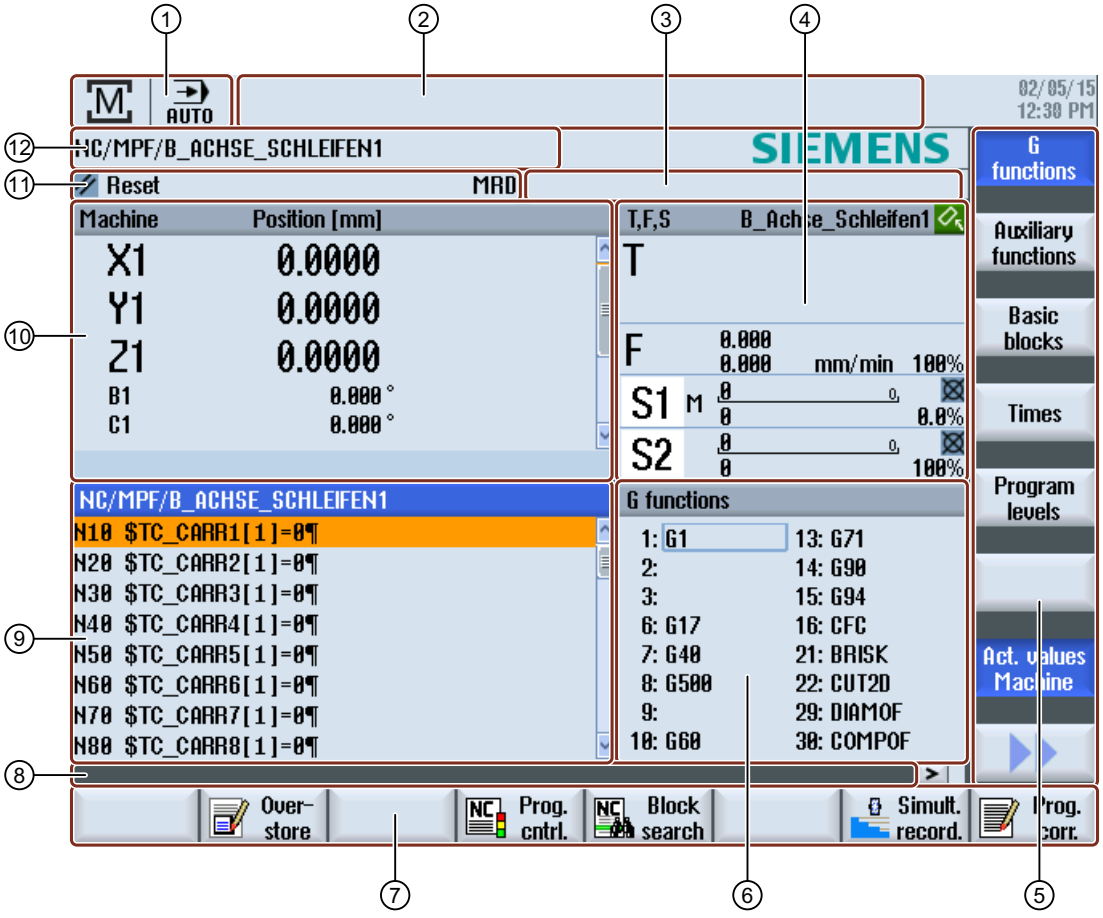
<FEED START>

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

2.4 User interface

2.4.1 Screen layout

Overview



① Active operating area and mode
② Alarm/message line
③ Channel operational messages
④ Display for
- Active tool T
- Current feedrate F
- Active spindle with current state (S)
- Spindle utilization rate in percent

⑤ Vertical softkey bar
⑥ Display active G functions, all G functions, H functions and input window for different functions (e.g. skip blocks, program control)
⑦ Horizontal softkey bar

⑧ Dialog line to provide additional user notes
⑨ Operating window with program block display
⑩ Axis position display in actual value window
⑪ Channel state and program control
⑫ Program name

2.4.2 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
	"Program manager" operating area

Display	Meaning
	"Diagnostics" operating area
	"Startup" operating area

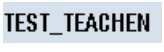
Active mode or function

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.

2.4.3 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
Please observe the information provided by the machine manufacturer.

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.	
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.	
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

Set collision avoidance (Page 294)

2.4.4 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.

The "T, S, F" window displays several spindles with a maximum of two utilization indicators. The grinding power display is integrated in the spindle speed display. The power bar is located in the Z plane behind the speed value.

The following applies to the spindle display:

- The master spindle is always displayed
- The PLC specifies which tool spindle should be displayed
- The spindle number entered in the tool data is also the active tool spindle if the value is not zero

Tool data

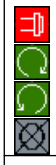
Display	Meaning
T	
Tool name	Name of the current tool.
Location	Location number of the current tool.
D	Cutting edge of the current tool. The tool is displayed with the associated tool type icon corresponding to the actual coordinate system in the selected cutting edge position. If the tool is swiveled, 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.
Ø	Diameter of the current tool.
R	Radius of the current 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, it is displayed for: • "JOG" mode: Axis feed for the traversing axis • "MDA" and "AUTO" mode: Programmed axis feed
Rapid traverse	G0 is active

Display	Meaning
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)
Icon  Spindle not enabled Spindle is turning clockwise Spindle is turning counterclockwise Spindle is stationary	Spindle status
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.

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 refer to the machine manufacturer's specifications.

2.4.5 Current block display

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

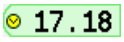
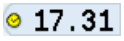
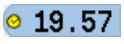
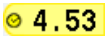
Display of current program

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	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.

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 208)

2.4.6 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.

2.4.7 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.

2.4.8 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.

2.4.9 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.

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

   - OR -

   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.

2.4.10 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.

2.4.11 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>.

2.4.12 Entering Chinese characters

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.

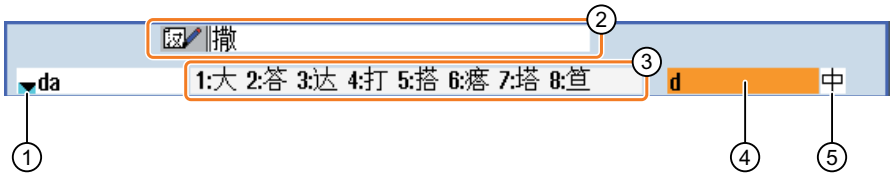
The editor is available for the following Asian languages:

- Simplified Chinese
- Traditional Chinese

Input types

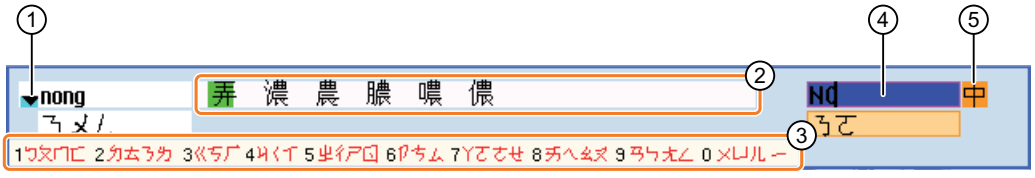
Input type	Description
Pinyin input	Latin letters are combined phonetically to denote the sound of the character. The editor lists all of the characters from the dictionary that can be selected.
Zhuyin input (only traditional Chinese)	Non-Latin letters are combined phonetically to denote the sound of 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 2-1 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 2-2 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.

2.4.12.1 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.

2.4.12.2 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

2.4.13 Entering Korean characters

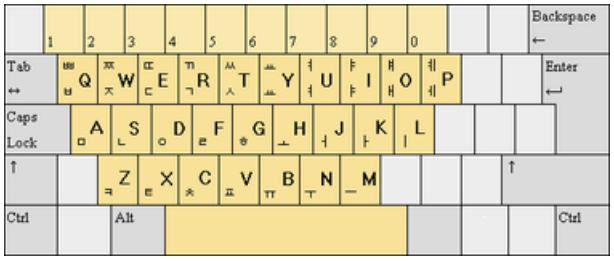
You can enter Korean characters in the input fields on classic panels (without touch operation) using the input editor IME (Input Method Editor).

Note

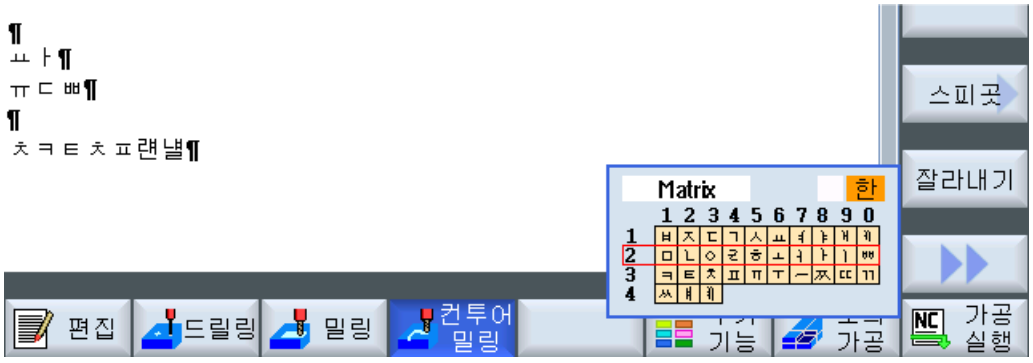
You require a special keyboard to enter Korean characters. If this is not available, then you can enter the characters using a matrix.

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.

2.4.14 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.

2.4.15 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 "Start-up" operating area.

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

2.4.16 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 side screen
- 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

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 side screen.

2.4.17 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.

Note

The camera is not available for operating SINUMERIK Operate on NCU ("Embedded HMI") when using an NCU 710.

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 side screen
- 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

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 side screen.

2.4.18 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.

Multitouch operation with SINUMERIK Operate 3

3.1 Multitouch panels

The "SINUMERIK Operate Generation 2" user interface has been optimized for multitouch 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:

- OP 015 black
- PPU 290.4
- HT 8
- HT 10
- SIMATIC ITC V3
- SIMATIC IFP
- SIMATIC panel IPC

Additional information

You can find further information on configuring the user user interface in the SINUMERIK Operate Commissioning Manual.

You can find additional information on Multitouch Panels at:

- Operator Panels Equipment Manual (OP 015 black / 019 black)

3.2 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

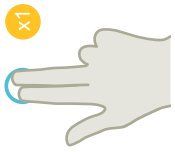
- Thermoplus KCL type 955
- KCL Men at Work type 301
- Camapur Comfort type 619
- Comasec PU (4342)

3.3 Finger gestures

Finger gestures

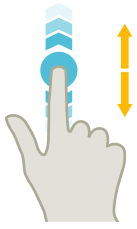
Tap

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



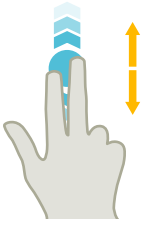
Tap with 2 fingers

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



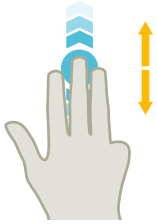
Flick vertically with one finger

- 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

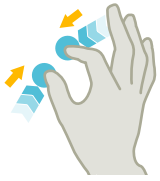
- Scroll to the start or end of lists
- Scroll to the start or end of files

Flick horizontally with one finger

- Scroll in lists with many columns

Spread

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

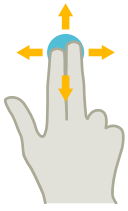


Pinch

- 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

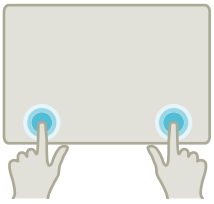
- Rotate graphic contents (e.g. simulation, mold making view)

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

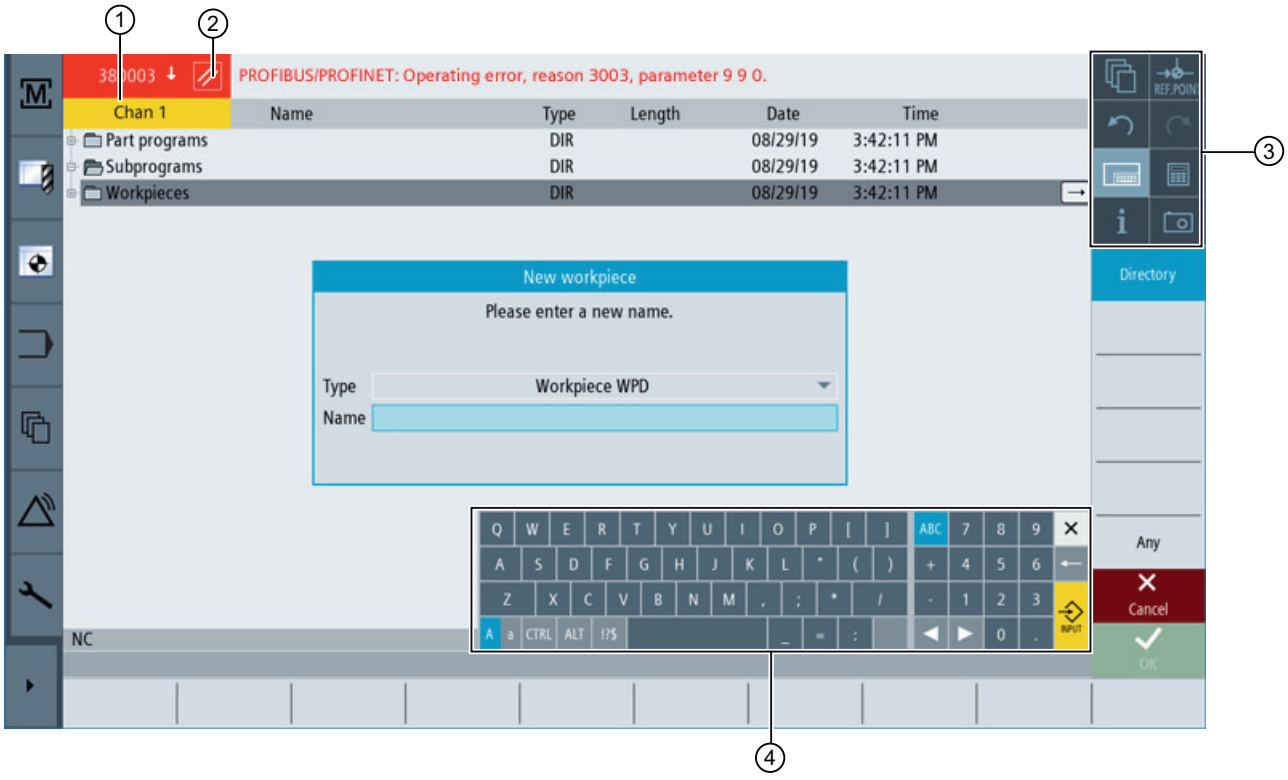
Flicking gestures with several fingers

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

3.4 Multitouch user interface

3.4.1 Screen layout

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



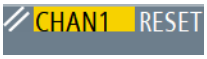
① Changing the channel
② Cancel alarms
③ Function key block
④ Virtual keyboard

3.4.2 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).
	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.

3.4.3 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.

Operator control	Function
	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.

3.4.4 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.

3.4.5 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.

3.5 Expansion with side screen

3.5.1 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
Please observe the information provided by the machine manufacturer.

Requirements

- A widescreen format multitouch panel (e.g. OP 015 black) 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.

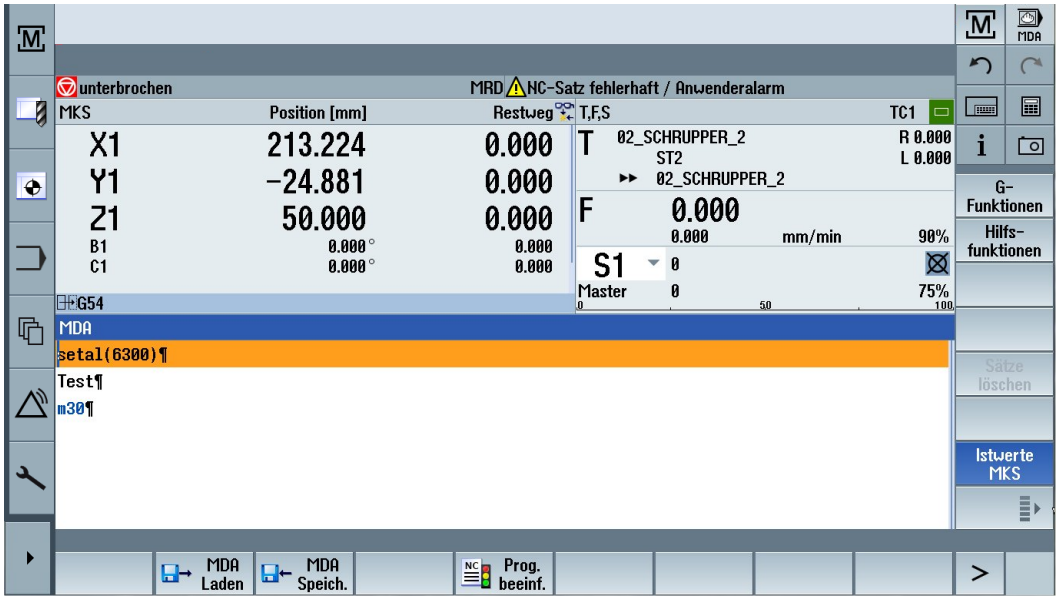
Further information

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

3.5.2 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.



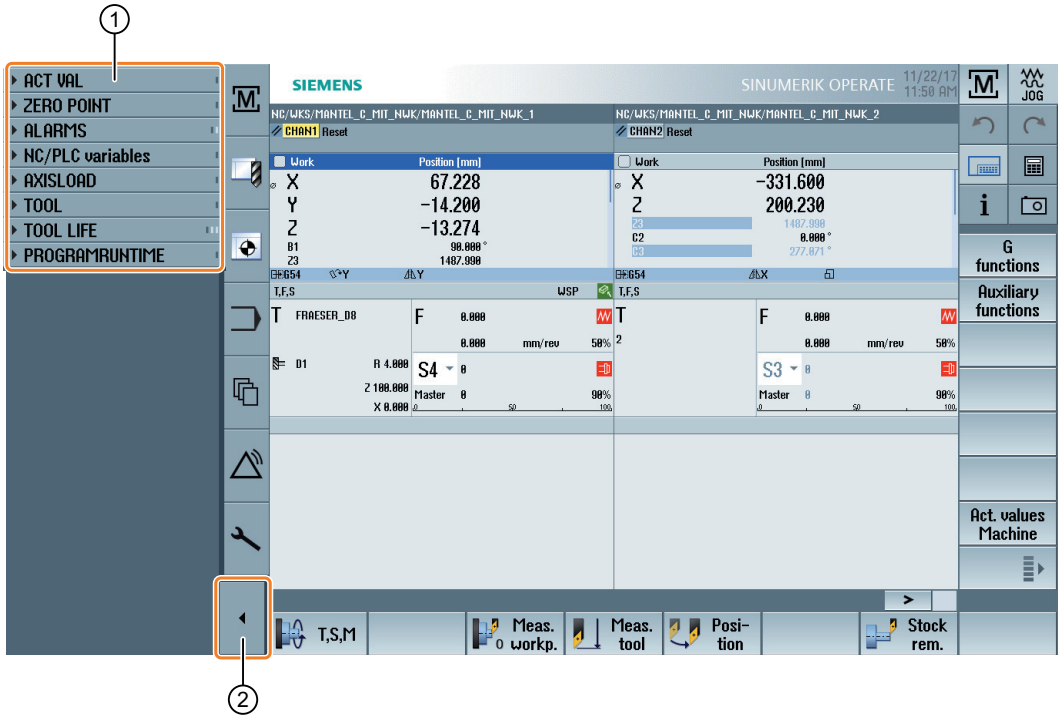
Navigation bar

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.
	Hides the sidescreen.
	Shows the sidescreen.

3.5.3 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.

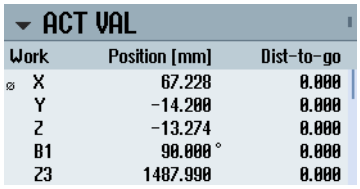
Open widgets

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

3.5.4 "Actual value" widget

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

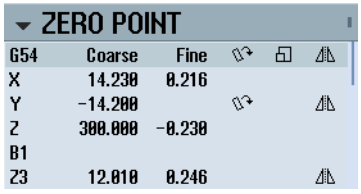
The distance-to-go for the current NC block is displayed while a program is running.



3.5.5 "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.



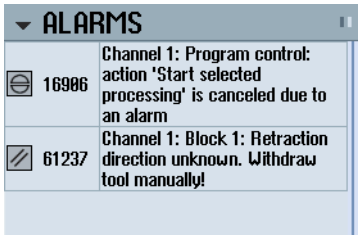
3.5.6 "Alarms" widget

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.

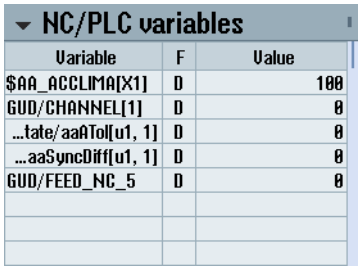
Wipe horizontally to switch between alarms and messages.



3.5.7 "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.



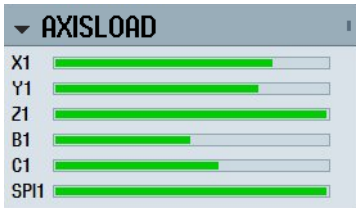
Only those variables that are currently displayed in the "NC/PLC variables" screen in the "Diagnostics" operating area are shown. To update the list in the "NC/PLC variables" widget following a change in the "NC/PLC variables" screen in the "Diagnostics" operating area, collapse and expand the widget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

Vertical scrolling is possible.

3.5.8 "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.

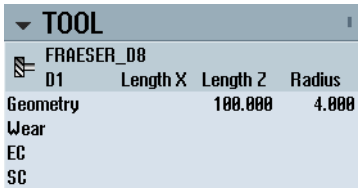


3.5.9 "Tool" widget

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



3.5.10 "Service life" widget

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

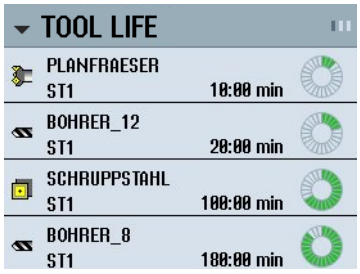
- Operating time of tool (standard time monitoring)
- Finished workpieces (quantity monitoring)
- Tool wear (wear monitoring)

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.



3.5.11 "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.

3.5.12 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.

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

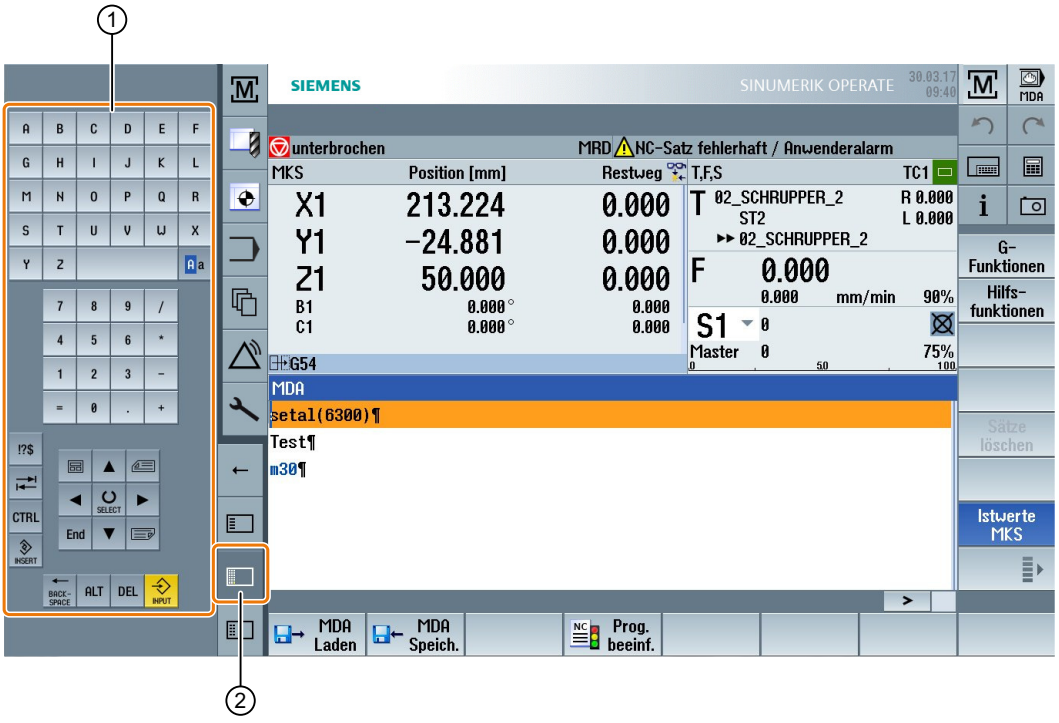
Not only standard widgets but also pages with ABC keyboards and machine control panels can be configured in the sidescreen of a multitouch panel.

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

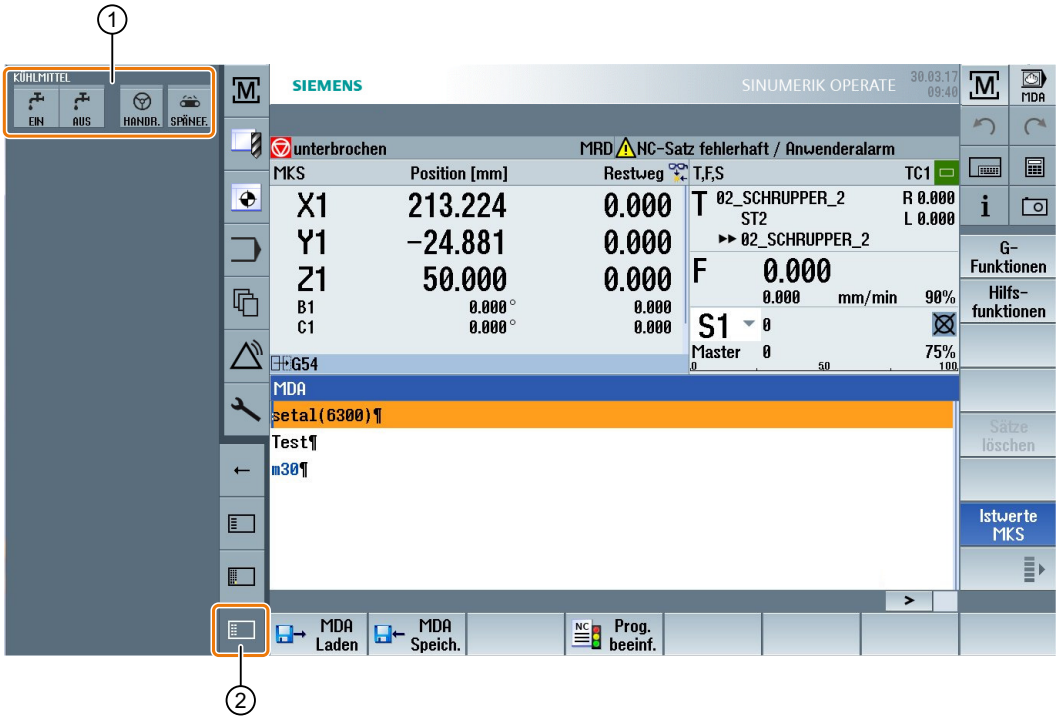
3.5.14 Example 1: ABC keyboard in the sidescreen



① ABC keyboard

② Key to display the keyboard

3.5.15 Example 2: Machine control panel in the sidescreen



① Machine control panel

② Key to display the machine control panel

3.6 SINUMERIK Operate Display Manager

3.6.1 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 720 and NCU 730.

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.

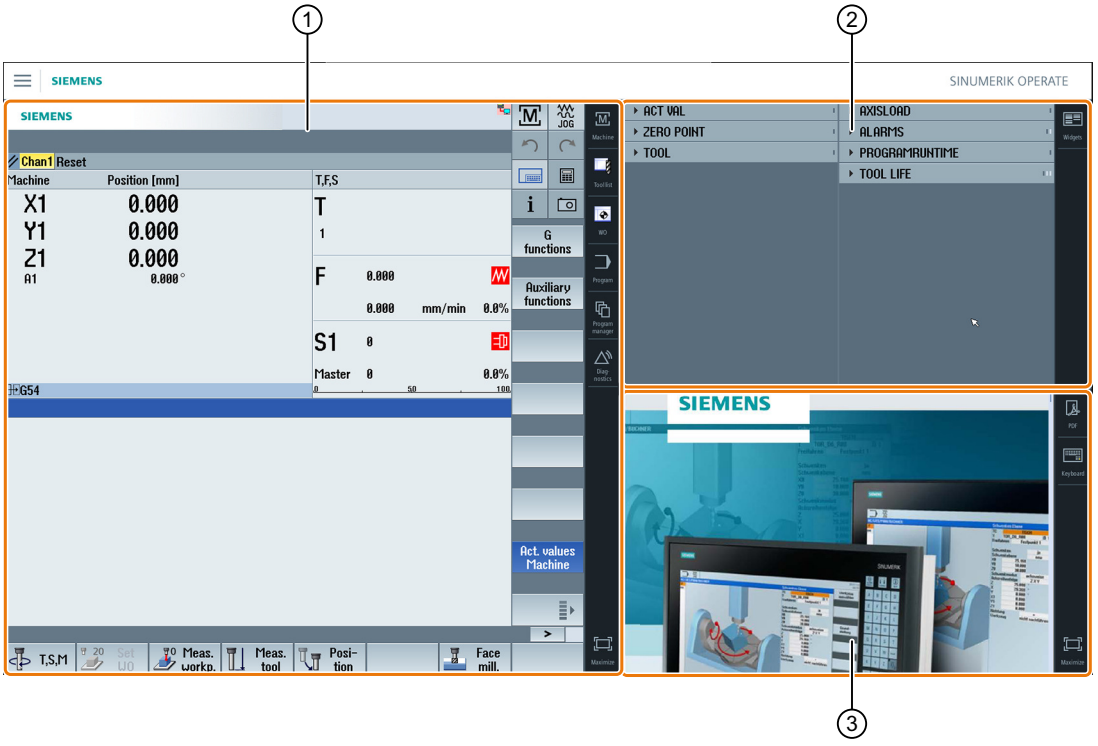
Additional information

For further 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.

3.6.2 Screen layout

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



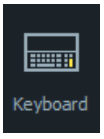
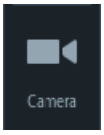
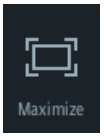
① SINUMERIK Operate with navigation bar for switchover of the operating area
② Display area for standard widgets
③ Display area for applications (e.g. PDF)

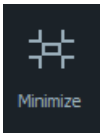
3.6.3 Operator controls

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 74) • Zero point (Page 75) • Tool (Page 76) • Axle load (Page 76) • Alarms (Page 75) • Program runtime (Page 77) • Service life (Page 77) • NC/PLC variables (Page 75)

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>) Alternatively, use the PDF display via the animated toolbar at the top left. 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.
	**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.

Operator control	Function
	**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

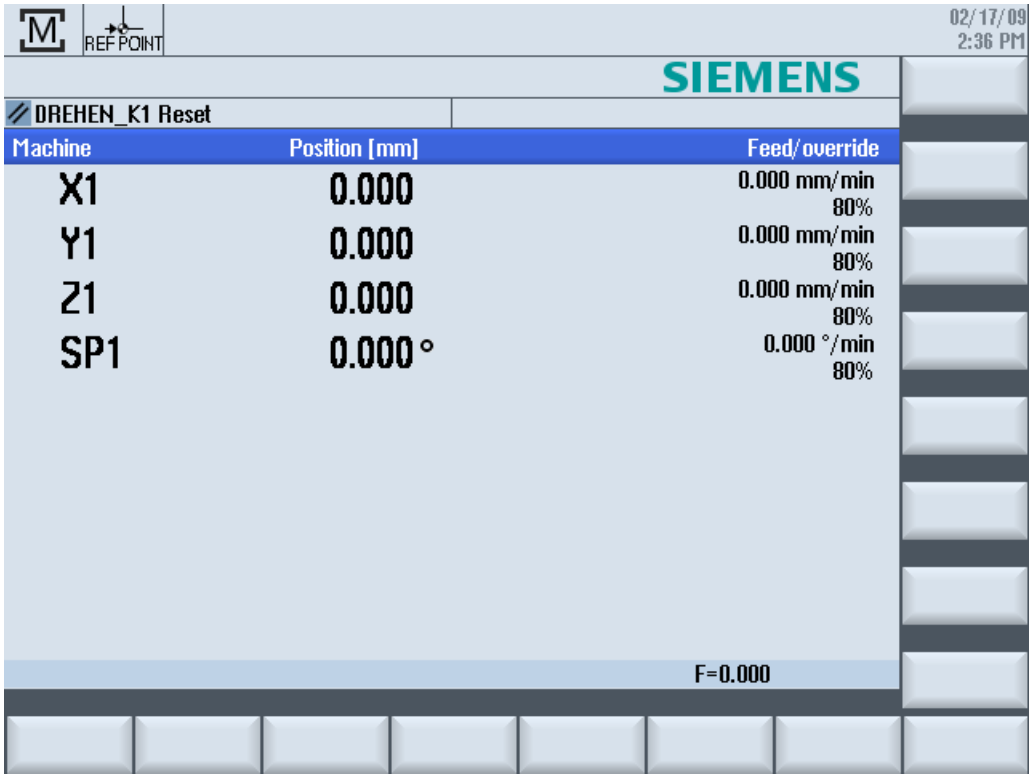
Widget "Camera 1" and "Camera 2" (Page 77)

Setting up the machine

4

4.1 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.

4.2 Approaching a reference point

4.2.1 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".

4.2.2 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.

4.3 Operating modes

4.3.1 Operating modes

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.

4.3.2 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.

4.3.3 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.

4.4 Settings for the machine

4.4.1 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.

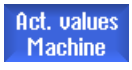
The machine coordinate system (MCS), in contrast to the workpiece coordinate system (WCS), does not take into account any zero offsets, tool offsets and coordinate rotation.

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.

4.4.2 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.

See also

Default settings for manual mode (Page 143)

4.4.3 Setting the zero offset

You can enter a new position value in the actual value display for individual axes when a settable zero 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 saved permanently in the currently active zero offset (e.g. G54).

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 zero 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 specifications.

Precondition

The controller is in the workpiece coordinate system.

The actual value is set in the reset state.

Note

Setting the ZO in the Stop state

If you enter the new actual value in the Stop state, the changes made are only visible and only take effect when the program is continued.

Procedure

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

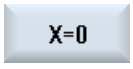
2. Press the "Set ZO" softkey.

   - 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.

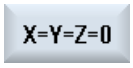
   - OR -



   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 ZO" softkey.

The offset is deleted permanently.

Note

Irreversible active zero offset

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

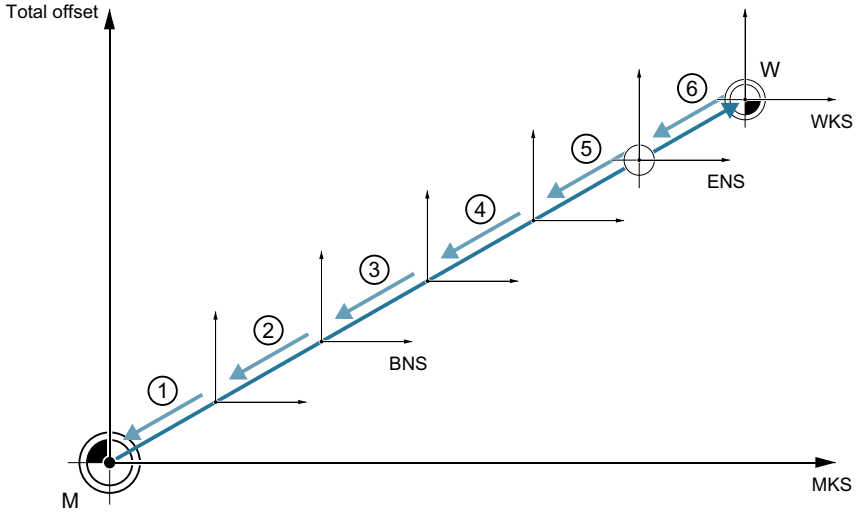
4.5 Zero offsets

4.5.1 Zero offsets

After 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.

After 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 $P_GFRAME
$P_IFRAME
⑤ $P_TOOLFRAME
$P_WPFRAME
⑥ Coordinate transformation

Figure 4-1 Zero 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.

Settable Zero System (SZS)

The SZS (**S**settable **Z**ero **S**ystem) corresponds to the WCS transformed by the programmable frame (e.g. $P_PFRAME, $PCYCFRAME, $P_TOOLFRAME and $P_WPFRAME).

Basic zero system (BZS)

The BZS (**B**asic **Z**ero **S**ystem) includes not only the frames of the SZS, but also the current settable frame ($P_IFRAME and $P_GFRAME).

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

Deselecting fine offset

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

4.5.2 Display active zero offset

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

- Work offsets, for which active offsets are included, or for which values are entered.
- Settable work offsets
- Seat-related fine offsets
- Total work offset

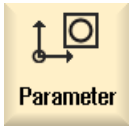
This window is generally used only for monitoring.

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 "Work offset" softkey.
   The "Work Offset - Active" window is opened.

Note

Further details on work 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.

4.5.3 Displaying the zero offset "overview"

The active offsets or system offsets are displayed for all set-up axes in the "Work Offset - Overview" window.

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

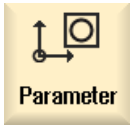
This window is generally used only for monitoring.

Display of active work offsets

Work offsets	
DRF	Displays the handwheel axis offset.
Rotary table reference	Displays the additional work offsets programmed with $ P_PART-FRAME.
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.
Programmed WO	Displays the additional work offsets programmed with $P_PFRAME.
Cycle reference	Displays the additional work offsets programmed with $P_CYCFRAME.

Work offsets	
GFRAME0 (seat-related work offset)	Displays the additional work offsets programmed with $P_GFRAME.
Total WO	Displays the active work offset, resulting from the total of all work offsets.

Procedure



1. Select the "Parameter" operating area.

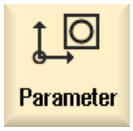
2. Press the "Work offset" and "Overview" softkeys.
   The "Work offset - Overview" window opens.

4.5.4 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

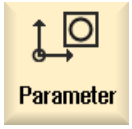
The offsets specified here are immediately active.

4.5.5 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.

4.5.6 Displaying and editing seat-related fine offset

In the window "Work offset - GFrame1 ... GFrame..." all position-related offset values (seat offsets) are displayed.

Translatory and rotary offsets are displayed.

Procedure

1. Select the "Parameter" operating area.

2. Press the "Work offset" softkey.

3. Press softkey "GFrame1...GFrame2".
   Window "Work offset - GFrame1 ... GFrame2" is opened.
   Note:
   The labeling of the softkeys for the seat-related fine offsets varies, i.e. the seat-dependent work offsets configured at the machine are displayed (examples: GFRAME1…GFRAME1, GFRAME1…GFRAME2, GFRAME1…GFRAME100).
   Here, please observe the information provided by the machine manufacturer.
4. You can edit the values directly in the table.

Note

Activating seat-related fine offsets

The seat-related work offsets must first be selected in the program before they are activated.

4.5.7 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

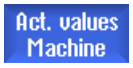
Settings for rotation, scaling and mirroring are specified here and can only be changed here.

Tool details

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

- TC
- Adapter dimension
- Length / length wear

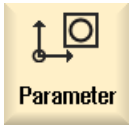
- EC setup correction
- SC sum correction
- Total length
- Radius / radius wear



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.

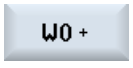
4. Place the cursor on the desired zero offset to view its details.

5. Press the "Details" softkey.

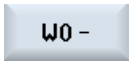
   A window opens, depending on the selected zero offset, e.g. "Zero Offset - Details: G54 to G599".

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".

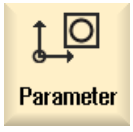
Machine manufacturer
Please refer to the machine manufacturer's specifications.

Press the "Back" softkey to close the window.

4.5.8 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.

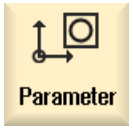
6. Press the "Clear offset" softkey.
   A confirmation prompt is displayed as to whether you really want to delete the work offset.

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

4.5.9 Deleting seat-related fine offsets

You have the option of deleting seat-dependent fine offsets after you have changed a tool. The values that have been entered are then set to zero.

Procedure



1. Select the "Parameter" operating area.

2. Press the "Work offset" softkey.

3. Press softkey "GFrame1...G-Frame2".

   Note:

   The labeling of the softkeys for the seat-related fine offsets varies, i.e. the seat-dependent work offsets configured at the machine are displayed (examples: GFRAME1…GFRAME1, GFRAME1…GFRAME2, GFRAME1…GFRAME100).
   Here, please observe the information provided by the machine manufacturer.

4. In the "Work offset - GFrame1 ... GFrame2" window in the table, set all values of the seat-related fine offsets to zero.

   - OR -

   Press the "Delete all" softkey.

   A confirmation prompt is displayed as to whether you really want to delete the seat-related fine offsets.

   Note:

   The "Delete all" softkey is only available if seat-related work offsets have been set up.

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

4.6 Measuring a tool

4.6.1 Cylindrical grinding

4.6.1.1 Overview

The geometries of the machining tool must be taken into consideration when executing a part program. These are stored as tool offset data in the tool list. Each time the tool is called, the control considers the tool offset data.

When programming the part program, you only need to enter the workpiece dimensions from the production drawing. The control then independently calculates the individual tool path.

Measuring grinding wheels and dressers

Tool offset data, i.e. the lengths or positions of the tools, are determined by taking manual measurements (scratching).

When measuring manually, traverse the tool manually to a selected reference point in order to determine the tool dimensions and positions in the X and Z directions. The control then calculates the tool offset data from the position of the tool carrier reference point and the approached reference point.

Measuring the reference points for the grinding wheel

For manual measurements of a grinding tool, you have the option of selecting the following reference points:

- Workpiece (with zero offset)
- Dresser (with zero offset as dressing tool)

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

Measuring the reference points for the dresser

Use the grinding wheel as reference point for manually measuring a dresser.

4.6.1.2 Measure grinding tool manually with the work piece reference point

Reference point

The workpiece edge serves as reference point when measuring length X and length Z.

You specify the position of the workpiece edge during the measurement.

Procedure

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

2. Press the "Meas. tool" softkey.

3. Press the "Measure wheel" softkey.

4. Press the "Select tool" softkey.
   The "Tool Selection" window opens.

5. In the "Tool Selection" window, select the grinding tool that you want to measure and press the "OK" softkey.
   The cutting edge location must be entered in the tool list.
   - OR -

   Press the "Tool list" softkey, select in the tool list the grinding tool that you want to measure and press the "In manual" softkey.
   The tool is transferred to the "Measure: "Grinding Wheel" window.

6. Select the "Workpiece" entry in the "Reference point" selection field.

7. Press the "X" or "Z" softkey, depending on which tool length you want to measure.

8. Enter the position of the workpiece edge in X0 and Z0.
   If no value is entered for X0 or Z0, the value is taken from the actual value display.
9. Scratch the required edge using the tool.

10. Press the "Set length" softkey.
    The tool length is calculated automatically and entered in the tool list. The cutting edge location is considered automatically.

Note

Active grinding tool

Tool measurement is possible only with an active grinding tool.

4.6.1.3 Measure grinding tool manually with the dresser reference point

Reference point

A dresser serves as reference point for the measurement of length X or Z.

The reference point of the dresser can be represented by a zero offset or a dressing tool. This setting is permanently stored in the machine data and specified by the machine manufacturer.

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

Procedure

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

2. Press the "Meas. tool" softkey.

3. Press the "Measure wheel" softkey.

4. Press the "Select tool" softkey.
   The "Tool Selection" window opens.

5. In the "Tool Selection" window, select the grinding tool that you want to measure and press the "OK" softkey.
   The cutting edge location must be entered in the tool list.
   - OR -

   Press the "Tool list" softkey, select in the tool list the grinding tool that you want to measure and press the "In manual" softkey.
   The tool is transferred to the "Measure: "Grinding Wheel" window.

6. Select the "Dresser" entry in the "Reference point" selection field.

7. Position the cursor in the "TR" field, press the "Select dresser" softkey, select the dresser for measuring the tool length and press the "OK" softkey.
   - OR -

   Position the cursor in the "Zero offset" field and press the "Select ZO" softkey.

Select in the "Zero Offset - G54 ... G509" window the desired zero offset and press the "In manual" softkey.

8. Press the "X" or "Z" softkey, depending on which tool length you want to measure.

9. Scratch the dresser using the tool.
10. Press the "Set length" softkey.

The tool length is calculated automatically and entered in the tool list.

The cutting edge location is considered automatically.

Note

Active grinding tool

Tool measurement is possible only with an active grinding tool.

4.6.1.4 Measuring the dressing tool manually with the grinding tool reference point

Reference point

A grinding wheel serves as reference point for the measurement of length X or Z.

Procedure

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

2. Press the "Meas. tool" softkey.

3. Press the "Measure dresser" softkey.

4. Press the "Select dresser" softkey.

   The "Tool Selection" window opens.

5. In the "Tool Selection" window, select the dressing tool that you want to measure and press the "OK" softkey.

   The cutting edge location must be entered in the tool list.

   - OR -

Tool list

Press the "Tool list" softkey, select in the tool list the dressing tool that you want to measure and press the "In manual" softkey.

In manual

The tool is transferred to the "Measure: Dresser" window.

- OR -

Select work offs.

Position the cursor to the "Zero offset" field and press the "Select ZO" softkey.

In manual

Select in the "Zero Offset - G54 ... G509" window the desired zero offset and press the "In manual" softkey.

Select grind. wheel

6. Position the cursor in the "TR" field, press the "Select grinding wheel" softkey, select the grinding wheel to be used as reference point and press the "OK" softkey.

X

7. Press the "X" or "Z" softkey, depending on which tool length you want to measure.

Z

8. Scratch the required edge using the tool.

Set length

9. Press the "Set length" softkey.

The tool length is calculated automatically and entered in the tool list. The cutting edge location is considered automatically.

4.6.2 Surface grinding

4.6.2.1 Overview

The geometries of the machining tool must be taken into consideration when executing a part program. These are stored as tool offset data in the tool list. Each time the tool is called, the control considers the tool offset data.

When programming the part program, you only need to enter the workpiece dimensions from the production drawing. The control then independently calculates the individual tool path.

Measuring grinding wheels and dressers

Tool offset data, i.e. the lengths or positions of the tools, are determined by taking manual measurements (scratching).

When measuring manually, traverse the tool manually to a defined reference point in order to determine the tool dimensions and positions in the Y and Z directions. The control then calculates the tool offset data from the position of the tool carrier reference point and the reference point.

Reference points for grinding wheels

For manual measurements of a grinding tool, you have the option of selecting the following reference points:

- Workpiece (with zero offset)
- Dresser (with zero offset as dressing tool)

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

Reference points for dressers

Use the grinding wheel as reference point for manually measuring a dresser.

4.6.2.2 Measure grinding tool manually with the work piece reference point

Reference point

The workpiece edge serves as reference point when measuring length X and length Z.

You specify the position of the workpiece edge during the measurement.

Procedure

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

2. Press the "Meas. tool" softkey.

3. Press the "Measure wheel" softkey.

4. Press the "Select tool" softkey.
   The "Tool Selection" window opens.

5. In the "Tool Selection" window, select the grinding tool that you want to measure and press the "OK" softkey.
   The cutting edge location must be entered in the tool list.
   - OR -

   Press the "Tool list" softkey, select in the tool list the grinding tool that you want to measure and press the "In manual" softkey.

The tool is transferred to the "Measure: Grinding Wheel" window.

6. Select the "Workpiece" entry in the "Reference point" selection field.

7. Press the "Y" or "Z" softkey, depending on which tool length you want to measure.
8. Enter the position of the workpiece edge in Y0 or Z0.

   If no value is entered for Y0 or Z0, the value is taken from the actual value display.
9. Scratch the required edge using the tool.

10. Press the "Set length" softkey.

    The tool length is calculated automatically and entered in the tool list. The cutting edge location is considered automatically.

Note

Active grinding tool

Tool measurement is possible only with an active grinding tool.

4.6.2.3 Measure grinding tool manually with the dresser reference point

Reference point

A dresser serves as reference point for the measurement of length Y or Z.

The reference point of the dresser can be represented by a zero offset or a dressing tool. This setting is permanently stored in the machine data and specified by the machine manufacturer.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure

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

2. Press the "Meas. tool" softkey.

3. Press the "Measure wheel" softkey.

4. Press the "Select tool" softkey.
   The "Tool Selection" window opens.

5. In the "Tool Selection" window, select the grinding tool that you want to measure and press the "OK" softkey.
   The cutting edge location must be entered in the tool list.
   - OR -

   Press the "Tool list" softkey, select in the tool list the grinding tool that you want to measure and press the "In manual" softkey.
   The tool is transferred to the "Measure: "Grinding Wheel" window.

6. Select the "Dresser" entry in the "Reference point" selection field.

7. Position the cursor in the "TR" field, press the "Select dresser" softkey, select the dresser for measuring the tool length and press the "OK" softkey.
   The tool is transferred to the "Measure: Length Manual" window.
   - OR -

   Press the "Tool list" softkey, select in the tool list the dressing tool for measuring the tool length and press the "In manual" softkey.
   The tool is transferred to the "Measure: "Grinding Wheel" window.

8. Press the "Y" or "Z" softkey, depending on which tool length of the dresser tool to be measured.

9. Scratch the dresser using the tool.

10. Press the "Set length" softkey.
    The tool length is calculated automatically and entered in the tool list.
    The cutting edge location is considered automatically.

Note

Active grinding tool

Tool measurement is possible only with an active grinding tool.

4.6.2.4 Measuring the dressing tool manually with the grinding tool reference point

Reference point

A grinding wheel serves as reference point for the measurement of length X, Y or Z.

Procedure

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

2. Press the "Meas. tool" softkey.

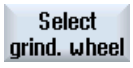
3. Press the "Measure dresser" softkey.

4. Press the "Select dresser" softkey.
   The "Tool Selection" window opens.

5. In the "Tool Selection" window, select the dressing tool to be measured and press the "OK" softkey.
   The cutting edge location must be entered in the tool list.
   - OR -

   Press the "Tool list" softkey, select in the tool list the dressing tool to be measured and press the "In manual" softkey.

   The tool is transferred to the "Measure: Dresser" window.



6. Position the cursor in the "TR" field, press the "Select grinding wheel" softkey.

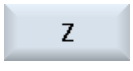
   Select the grinding wheel for measuring the tool length and press the "OK" softkey
   - OR -

   Press the "Tool list" softkey, select in the tool list the grinding wheel for measuring the tool length and press the "In manual" softkey.

   The tool is transferred into the "Measure: Dresser" window.

7. Press the "X", "Y" or "Z" softkey, depending on which tool length of the dresser tool to be measured.

...



8. Scratch the dresser using the tool.

9. Press the "Set length" softkey.
   The tool length is calculated automatically and entered in the tool list.
   The cutting edge location is considered automatically.

Note

Active dressing tool

Tool measurement is possible only with an active dressing tool.

4.7 Measuring the workpiece zero

4.7.1 Cylindrical grinding

4.7.1.1 Measuring the workpiece zero

The reference point for programming a workpiece is always the workpiece zero. To determine this zero point, measure the length or the diameter of the workpiece and save it in a zero offset. This means that the position is stored in the coarse offset and existing values in the fine offset are deleted.

Calculation

When the workpiece zero / zero offset is calculated, the tool length is automatically taken into account.

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

Requirement

The requirement for measuring the workpiece is that a tool with known lengths is in the machining position.

Procedure

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

2. Press the "Workpiece zero" softkey.
   The "Measure: Edge" window opens.

3. Select "Measuring only" if you only wish to display the measured values.

   - OR -

   Select in the "Zero offset" field the desired zero offset in which you want to store the zero point (e.g. basis reference).

   - OR -

Press the "Select ZO" softkey and select the zero offset in which the zero point is to be saved in the "Zero offset – G54 … G599" window and press the "In manual" softkey.

You return to the "Measure: Edge" window.

4. Traverse the tool in the X or Z direction and scratch the workpiece.
5. Enter the position setpoint of the workpiece edge X0 or Z0 and press the "Set ZO" softkey.

Note

Settable zero offsets

The labeling of the softkeys for the settable zero offsets varies, i.e. the settable zero offsets configured on the machine are displayed (examples: G54…G57, G54…G505, G54…G599).

Please refer to the machine manufacturer's specifications.

4.7.2 Surface grinding

4.7.2.1 Overview

The reference point for programming a workpiece is always the workpiece zero. The workpiece zero is determined on the workpiece edge.

Manual measurement

To measure the zero point manually, you need to traverse your tool manually up to the workpiece. Alternatively, you can also deploy a grinding tool with known length.

4.7.2.2 Setting the edge

The workpiece lies parallel to the coordinate system on the work table. You measure one reference point in one of the axes (X, Y, Z).

Requirement

A grinding tool is deployed for scratching when you measure the workpiece zero manually.

Procedure

1. Select the "Machine" operating area and press the <JOG> key.

2. Press the "Workpiece zero" and "Set edge" softkeys.
   The "Measure: Edge" window opens.

3. Select "Measuring only" if you only wish to display the measured values.

   - OR -

4. In the selection box, select the desired zero offset in which you want to store the zero point.

   - OR -

   Press the "Select ZO" softkey to select a settable zero offset.

   In the "Zero Offset – G54 ... G599" window, select a zero offset, in which the zero point should be saved.

   Press the "In manual" softkey.

   You return to the measurement window.

5. Use the softkeys to select in which axis direction you want to approach the workpiece first.

...

6. In X0, Y0, or Z0, specify the setpoint position of the workpiece edge.
   The setpoint position corresponds, e.g. to the dimension specifications of the workpiece edge from the workpiece drawing.

Note

Settable zero offsets

The labeling of the softkeys for the settable zero offsets varies, i.e. the settable zero offsets configured on the machine are displayed (examples: G54…G57, G54…G505, G54…G599).

Please refer to the machine manufacturer's specifications.

4.8 Monitoring axis and spindle data

4.8.1 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.

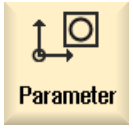
In this way, you are able to restrict the traversing range of the axes in addition to the limit switches.

Requirements

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

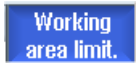
You can make changes in "JOG" mode at any time. These changes, however, only become active at the start of a new motion.

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.

4.8.2 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.

4.8.3 Spindle chuck data

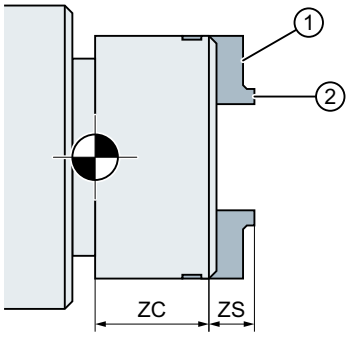
4.8.3.1 Defining spindle chuck data

You store the chuck dimensions of the spindles at your machine in the "Spindle Chuck Data" window.

Manually measuring a tool

If you want to use the chuck of the main or counter-spindle as a reference point during manual measuring, specify the chuck dimension ZC.

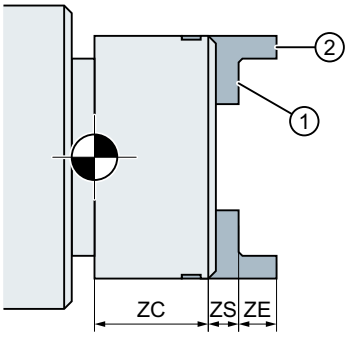
Main spindle



Dimensioning, main spindle jaw type 1

① Stop edge

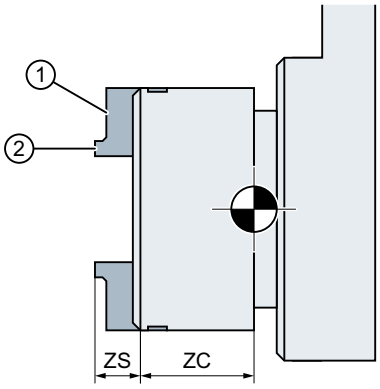
② Front edge



Dimensioning, main spindle jaw type 2

Counter-spindle

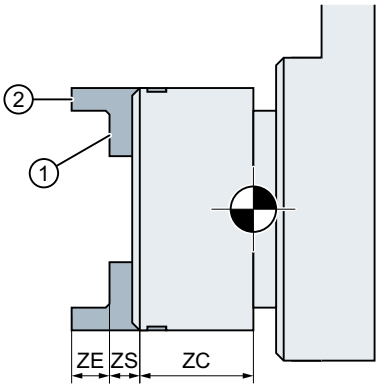
You can measure either the forward edge or stop edge of the counter-spindle. The forward edge or stop edge automatically serves as the valid reference point when traversing the counter-spindle. This is especially important when gripping the workpiece using the counter-spindle.



Dimensioning, counter-spindle jaw type 1

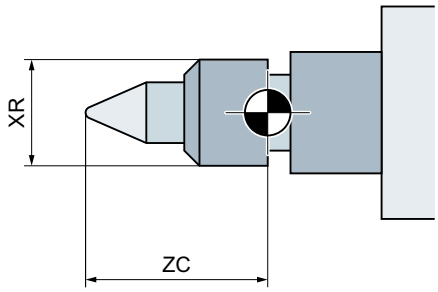
① Stop edge

② Front edge

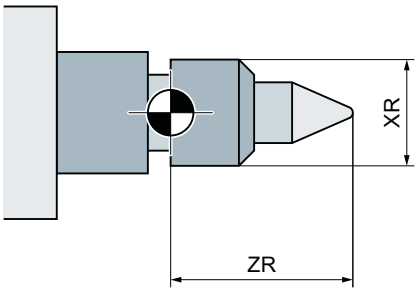


Dimensioning, counter-spindle jaw type 2

Tailstock

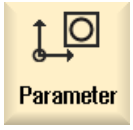


Dimensioning tailstock main spindle



Dimensioning tailstock counter-spindle

Procedure



1. Select the "Parameter" operating area.

2. Press the "Setting data" and "Spindle chuck data" softkeys.
   The "Spindle Chuck Data" window opens.

3. Enter the desired parameter.
   The settings become active immediately.

4.8.3.2 Parameters, spindle chuck data

Parameter	Description	Unit
Main spindle		
	Dimensions of the forward edge or stop edge • Jaw type 1 • Jaw type 2	
ZC1	Main spindle chuck dimensions (inc)	mm
ZS1	Main spindle stop dimensions (inc)	mm
ZE1	Jaw dimension, main spindle (inc) - only for "Jaw type 2"	mm
XR	Tailstock diameter - only for tailstock that has been set-up	mm
ZR	Tailstock length - only for tailstock that has been set-up	mm
Counter-spindle		
	Dimensions of the forward edge or stop edge • Jaw type 1 • Jaw type 2	
ZC3	Chuck dimension, counter-spindle (inc) - only for a counter-spindle that has been set-up	mm

Parameter	Description	Unit
ZS3	Stop dimension, counter-spindle (inc) - only for a counter-spindle that has been set-up	mm
ZE3	Jaw dimension, counter-spindle (inc) - only for a counter-spindle that has been set-up and "Jaw type 2"	mm
XR	Tailstock diameter - only for tailstock that has been set-up	mm
ZR	Tailstock length - only for tailstock that has been set-up	mm

4.8.4 Entering cylinder error compensation (only rotary grinding machine)

The "Cylinder error compensation" function allows you to correct cylinder errors that have occurred when clamping the workpiece. The maximum compensation value is 10 mm.

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

Software option
In order to use this function, you need the software option: "SINUMERIK Grinding Advanced".

Further information

You can find further information on cylinder error compensation in the Monitoring and Compensation Function Manual.

Precondition

- The option is set.
- A compensation table for sag compensation (CEC) has been set up.

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

Procedure

1. Select the "Parameter" operating area.

2. Press the "Setting data" and "Cylinder error compensation" softkeys.
   The "Cylinder Error compensation" window opens.

3. Select the desired data set in the "Compensation" selection field.

4. For measuring points P1 and P2, enter the basis value (ZM) and compensation value (XM).

   The "Set comp." softkey can be used after entering the compensation values.

5. Press the "Set comp." softkey.

   The compensation is applied.

   - OR -

   Press the "Delete comp." softkey if you wish to delete the compensation values.

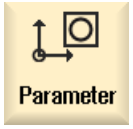
   The cylinder error compensation is deleted.

4.9 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.

4.10 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).

4.11 MDI

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.

4.11.1 Loading an MDA program from the Program Manager

Procedure

1. Select the "Machine" operating area.

2. Press the <MDI> key.
   The MDI editor opens.

3. Press the "Load MDI" softkey.
   The "Load in 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.

4.11.2 Saving an MDA program

Procedure

1. Select the "Machine" operating area.

2. Press the <MDI> key.

   The MDI editor opens.

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

4. Press the "Store 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.

4.11.3 Editing/executing a MDI program

Procedure

1. Select the "Machine" operating area.

2. Press the <MDI> 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

4.11.4 Deleting an MDA program

Precondition

The MDA 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 the manual mode

5

5.1 Overview

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.

5.2 Selecting a tool and spindle

5.2.1 T,S,M window

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

In addition to the main spindle (S1), there is another tool spindle (S2) for powered tools.

Your turning machine can also be equipped with a counter-spindle (S3).

In manual mode, you can select a tool on the basis of either its name or its revolver 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 revolver location number, you can also 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.

Display	Meaning
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)
DL	Number for additive and setup offset
ST	Sister tool (for replacement tool strategy)
Spindles 1 and 2 (e.g. S1)	Spindle selection for master spindle and 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.

Display	Meaning
G zero offset	Selection of the zero offset (basic reference, G54 - 57) You can select zero offsets from the tool list of settable zero offsets via the "Zero offset" softkey.
Measurement unit	Selection of the unit of measurement (inch, mm). The setting made here has an effect on the programming.
Machining plane	Selection of the machining plane (G17(XY), G18 (ZX), G19 (YZ))
Gear stage	Specification of the gear stage (auto, I - V)
Stop position	Input of the spindle position in 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.

5.2.2 Selecting a tool

Procedure

1. Select the "JOG" operating mode.

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

- OR -

3. Enter the name or the number of the tool T.

   - OR -

   Press the "Select tool" softkey to open the tool list, position the cursor on the desired tool and press the "In Manual" 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. Press the "CYCLE START" key.

   The tool is loaded into the spindle.

5.2.3 Starting and stopping the spindle manually

Procedure

1. Select the "T,S,M" softkey in the "JOG" mode.

- OR -

2. Select the desired spindle (e.g. S1) and enter the desired spindle speed or cutting speed in the right-hand entry field.
3. If the machine has a gearbox for the spindle, set the gearing step.

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

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

6. 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.

5.2.4 Positioning the spindle

Procedure

1. Select the "T,S,M" softkey in the "JOG" mode.

- OR -

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

4. 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.

5.3 Traversing axes

5.3.1 Traversing the 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.

5.3.2 Traverse axes by a defined increment

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

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 traverse path 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 specifications.

5.3.3 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.

5.4 Positioning axes

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

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. Enter the target position or target angle for the axis or axes to be traversed.
5. Specify the desired value for the feedrate F.

   - OR -

   Press the "Rapid traverse" softkey.

   The rapid traverse is displayed in field "F".
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.

5.5 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.

See also

Switching the unit of measurement (Page 95)

Machining the workpiece 6

6.1 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.

6.2 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 changeover to the "Machine" operating area occurs.

6.3 Executing a trial program run

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.

6.4 Displaying the current program block

6.4.1 Current block display

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

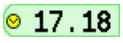
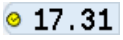
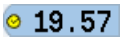
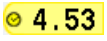
Display of current program

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
Green font 	"G1" motion command

Display	Meaning
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 208)

6.4.2 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.

6.4.3 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

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

6.5 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.

6.6 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.

6.7 Starting execution at a specific point

6.7.1 Use block search

If you only want to perform 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
- Specifying 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:
    During the block search, you must ensure that the correct tool is in the working position before beginning program execution.
  - 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)
    The level can only be changed if an interruption point in a subprogram is selected. You can then change the program level up 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. It is possible to continue the cascading any number of times after every search target found.

Note

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

Preconditions

- 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 version.

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

You can find further information on the block search function in the Basic Functions Function Manual.

6.7.2 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

6.7.3 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.

6.7.4 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.

6.7.5 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

6.7.6 Block search mode

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

The set mode is retained when the the controller is shut down. When you activate the "Search" function after restarting the controller, 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. Information such as this 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	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 (after REPOS motion) 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.

Further information

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

6.8 Controlling the program run

6.8.1 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. Please observe the information provided by the machine manufacturer.
DRF Handwheel offset	Enables an additional incremental work offset while processing in automatic mode with an electronic handwheel. This function can be used to compensate for tool wear within a programmed block.
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.
GCC	When executing a jobshop program, it is converted into a G-code program.

Abbreviation/program control	Mode of operation
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 can 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 You can also define NC functions (auxiliary functions, cycle calls, T-preselection) as NC function transitions. **Note**: Please observe 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.

6.8.2 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.

6.9 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.

6.10 Editing a program

6.10.1 Processing a program (editor)

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.

6.10.2 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
	The cursor is set to the first character in the program.
	The cursor is set to the last character in the program.

6.10.3 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.

6.10.4 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.

6.10.5 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.

6.10.6 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.

6.10.7 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".
Display 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 of the program with 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.

Setting	Meaning
Cut only after selecting	• 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.
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.
Saving 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.
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 "Delete mach. 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.

6.11 Working with DXF files

6.11.1 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"

6.11.2 Displaying CAD drawings

6.11.2.1 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.

6.11.2.2 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.

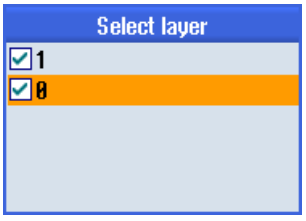
Requirement

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.

6.11.2.3 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.

6.11.2.4 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

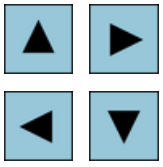
1. Press the "Details" and "Magnifying glass" softkeys.
   A magnifying glass in the shape of a rectangular frame appears.

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.

6.11.2.5 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.

...

6.11.2.6 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".

6.11.3 Importing and editing a DXF file

6.11.3.1 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

6.11.3.2 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.

6.11.3.3 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.

6.11.3.4 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".

6.11.3.5 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.

6.11.3.6 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.

6.11.3.7 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 -

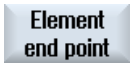
   Press "Only to 1st cut" if you do not want to accept the complete contour elements at once.
   The contour will be followed to the first cut of the contour element.

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.

6.12 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.

6.12.1 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

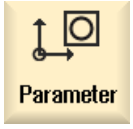
The number of global R parameters is defined in a machine data element.

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.

6.12.2 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.

6.12.3 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)

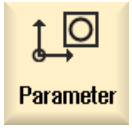
Example

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.

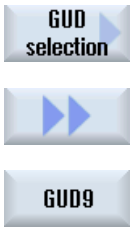
3. Press the "Global GUD" softkeys.

   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.

6.12.4 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

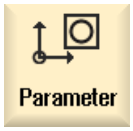
A channel-specific GUD variable is defined with the following:

- 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.

6.12.5 Displaying local user data (LUD)

Local user variables

LUDs are only valid in the program or subprogram in which they were defined.

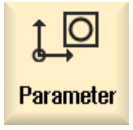
The controller displays the LUDs after the start of program processing. The display is available until the end of program processing.

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.

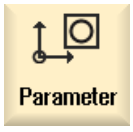
6.12.6 Displaying program user data (PUD)

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.

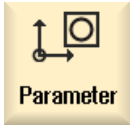
2. Press the "User variable" softkey.

3. Press the "Program PUD" softkey.

6.12.7 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.

6.13 Display G and auxiliary functions

6.13.1 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.

6.13.2 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

6.13.3 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).

Software option

You require the "Advanced Surface" software option in order to use this function.

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.

Further information

Additional 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.

6.13.4 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.

6.14 Displaying superimpositions

You can display handwheel axis offsets or programmed superimposed movements in the "Superimpositions" window.

Input field	Meaning
Tool	Current superimposition in the tool direction
Min	Minimum value for superimposition in the tool direction
Max	Maximum value for superimposition in the tool direction
DRF	Displays the handwheel axis offset

The selection of values displayed in the "Superimposition" 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 "Superimposition" softkeys.
   The "Superimposition" window opens.

4. Enter the required new minimum and maximum values for superimposition and press the <INPUT> key to confirm your entries.
   Note:
   You can only change the superimposition values in "JOG" mode.

5. Press the "Superimposition" softkey again to hide the window.

6.15 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

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.

...

6.16 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.

6.17 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.

In addition, you have the option of additionally limiting the traversing speed for rapid traverse so that when running-in a new program with rapid traverse, no undesirable high traversing speeds occur.

Dry run feedrate

The feedrate defined here replaces the programmed feedrate during execution if you have selected "DRY run feedrate" under program control.

Reduced rapid traverse

The value entered here reduces the rapid traverse to the entered percentage value if you selected "RG0 reduced rapid traverse" under program control.

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 determination function is activated while the workpiece is being machined.

- Off
  The time determination function is deactivated while machining a workpiece, i.e. the machining times are not determined.
- Non-modal
  The machining times are determined for each traversing block of a main program.
- 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 as with the block-by-block setting.

Saving machining times

Here, you specify how the determined machining times are processed.

- Yes
  A subdirectory with the name "GEN_DATA.WPD" is created in the directory of the part program. The determined machining times are saved there in an ini file together with the name of the program.
- No
  The determined machining times are displayed only in the program block display.

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 not effect if you do not change the specified amount of 100%.

6. Select the required entry in the "Record machining times" and where relevant, in the "Save machining times" field.

See also

Current block display (Page 40)

Simulating machining 7

7.1 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 moving the machine axes. Incorrectly programmed machining steps are detected at an early stage and incorrect machining on the workpiece is prevented.

Graphical display

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 grinding wheel or a dressing tool is set to a value proportional to the tool radius so that cutting can be simulated.

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 that was valid when 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 zero offset.

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.

Machine coordinate system references

The simulation is implemented as workpiece simulation. This means that it is not assumed that the zero offset has already been precisely scratched or is known.

In spite of this, there are unavoidable machine coordinate system 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 coordinate system references can mean that collisions are shown in the simulation that would not occur with a realistic work offset, or conversely that collisions are not shown that could occur with a realistic work offset.

Programmable frames

All frames and zero 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

For the simulation or simultaneous recording, the traversing paths, i.e. the programmed tool paths, are displayed.

Display variants

You can choose between three variants of the graphical display:

- Simulation before machining of the workpiece
  Before you machine the workpiece on the machine, you will graphically display a quick run-through of 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 motion".
- 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 of simultaneous recording and simulation

The following views are available for all three variants:

- Side view
- Dressing view
- 3D view (with option)
- Face view - for cylindrical grinding
- Further side views - surface grinding
- Machine space (with "collision avoidance" option)

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 option "Simultaneous recording (real-time simulation)" for the "Simultaneous recording" function.

Determining the program runtime

The program runtime is determined when executing the simulation. The program runtime is displayed in the editor untíl the next program change at the end of the program.

Properties of simultaneous recording and simulation

Traversing paths

In simulation, the traversing paths displayed 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 such cases, the following message is output: "High-resolution image cannot be generated".

Working zone limitation

No working area limitations and software limit switches are effective in the tool simulation.

Start position for simulation and simultaneous recording

During simulation, the start position is converted to the workpiece coordinate system by application of the zero offset.

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 (Toolcarrier / TRAORI, TRACYL) are evaluated and must be correctly commissioned for correct simulation.
- Transformations with swiveled linear axis (TRAORI 64 - 69) as well as OEM transformations (TRAORI 4096 - 4098) are not supported.

- Changes to the toolcarrier 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.

7.2 Simulation before machining of the workpiece

7.2.1 Simulation before 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 0 to 120%.

7.2.2 Start simulation

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.

- OR -

5. Press "Start" softkey.

The program execution is displayed graphically on the screen. The machine axes do not move.

6. To stop simulation, press the "Stop" softkey.

- OR -

Press the "Reset" softkey to cancel the simulation.

7. 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.

7.3 Simultaneous recording before machining of the workpiece

7.3.1 Overview

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.

7.3.2 Starting simultaneous recording

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.

- OR -

4. Press the <CYCLE START> key.

   The program execution is displayed graphically.

5. Press the "Sim. rec." softkey again to stop the recording.

- OR -

7.4 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.

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.

- OR

3.

Press the <CYCLE START> key.
The machining of the workpiece is started and graphically displayed.

- OR

4.

Press the "Sim. rec." softkey again to stop the recording.

Note

Display of the traversing paths

- If you switch-off simultaneous recording during machining and then switch-on the function again at a later time, then the traversing paths generated in the intermediate time will not be displayed.

7.5 Program control during simulation

7.5.1 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

If you are working with the "Simultaneous recording" function, you use the rotary switch (override) on the control panel.

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 "<<" 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

Press the <Ctrl> and <M> keys simultaneously to select the maximum feedrate of 120%.

7.5.2 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

Press the <CTRL> and <S> keys simultaneously to enable and disable the single block mode.

7.6 Different views of a workpiece

7.6.1 Overview

The following views are available:

- Side view
- Dressing view
- 3D view (with option)
- Face view - for cylindrical grinding
- Further side views - for surface grinding
- Machine space (with option)

7.6.2 Side view

Displaying a side view

1. Simultaneous recording or simulation is started.
2. Press the "Side view" softkey.
   The side view shows the workpiece in the Z-X plane.

Changing the display

You can enlarge, reduce, pan, and rotate the simulation graphic and the segment.

7.6.3 Dressing view

Displaying the dressing view

1. Simultaneous recording or simulation is started.
2. Press the "Dressing view" softkey.
   In the dressing view, the contour is shown mirrored twice.

Changing the display

You can enlarge, reduce, pan, and rotate the simulation graphic and the segment.

7.6.4 3D view

Displaying the 3D view

1. Simultaneous recording or 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 enlarge, reduce, pan, and rotate the simulation graphic and the segment.

Displaying and moving cutting planes
You can display and move cutting planes X, Y, and Z.

7.6.5 Face view - for cylindrical grinding

Displaying the face view

1. Simultaneous recording or simulation is started.
2. Press the "Further views" and "Face view" softkeys.

The face view shows the workpiece in the X-Y plane.

Changing the display
You can enlarge, reduce, pan, and rotate the simulation graphic and the segment.

7.6.6 Further side views - for surface grinding

Displaying further side views

1. Simultaneous recording or simulation is started.
2. Press the "Further views" softkey.

3. Press the "From the front" softkey if you wish to view from the front.

- OR -

Press the "From the rear" softkey if you wish to view from the rear.

- OR -

Press the "From the left" softkey if you wish to view from the left.

- OR -

Press the "From the right" softkey if you wish to view from the right.

Changing the display

You can enlarge, reduce, pan, and rotate the simulation graphic and the segment.

7.6.7 Machine space (with "collision avoidance" option)

Displaying the machine space view

1. Simultaneous recording or the simulation is started.

4. Press the "Other views" and "Machine space" softkeys.

During simultaneous recording, an active machine model is displayed.

Changing the display

You have the capability of enlarging, reducing, moving, and rotating the simulation graphic and to change the section.

7.7 Editing the simulation display

7.7.1 Deleting 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.

Procedure

1. Simultaneous recording is started.
2.
   Press the ">>" softkey.
   The tool paths are displayed.
4.

   Press the "'Delete tool path" softkey.
   All tool paths recorded up until now are deleted.

7.8 Editing and adapting a simulation graphic

7.8.1 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

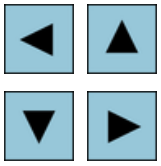
The selected sections and size changes are kept as long as the program is selected.

7.8.2 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.

7.8.3 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.

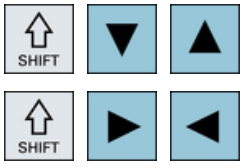
2. Press the "Rotate view" softkey.

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.

7.8.4 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.

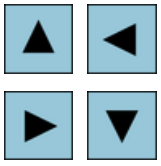
2. Press the "Magnifying glass" softkey.
   A magnifying glass in the shape of a rectangular frame appears.

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.

Generating a G-code program 8

8.1 Graphical programming

Functions

The following functionality is available:

- 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.

8.2 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.

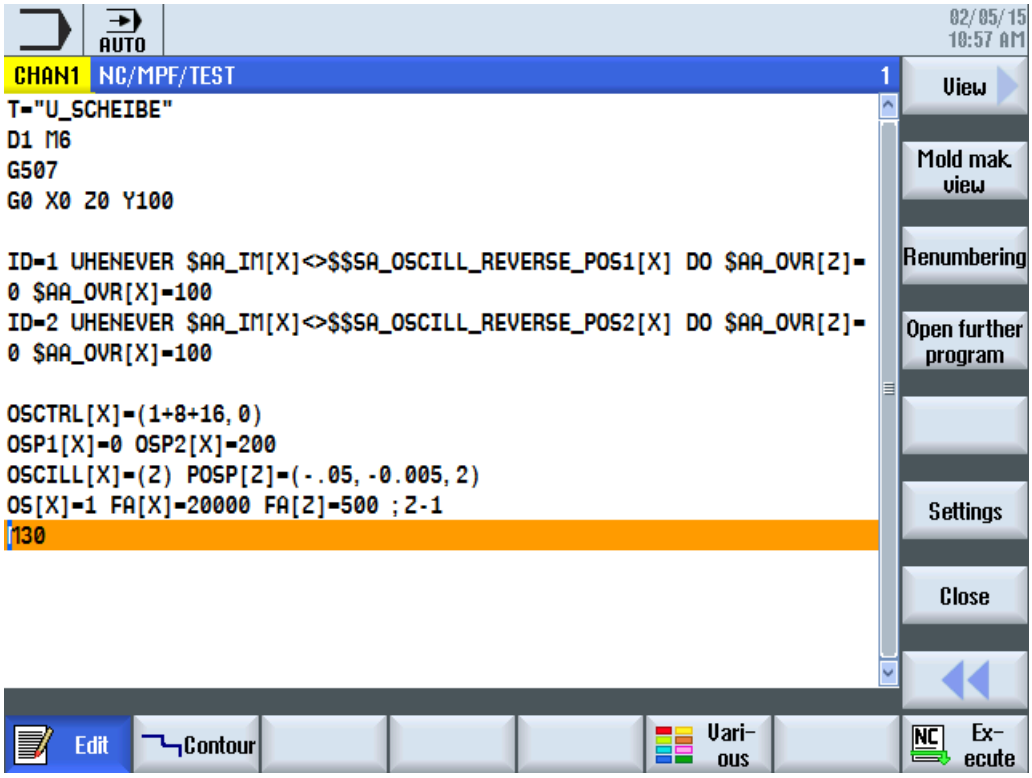
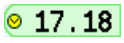
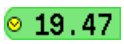
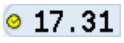
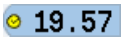
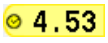


Figure 8-1 Program view of a G code program

Representation of the machining times

Representation	Meaning
Light green background 	Measured machining time of the program block (automatic mode)
Green background 	Measured machining time of the program group (automatic mode)

Representation	Meaning
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)

See also

Editor settings (Page 174)

8.3 Program structure

G-code programs can always be freely programmed. The most important commands that are normally included are:

- Set a machining plane
- Call a tool (T and D)
- Call a zero 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.

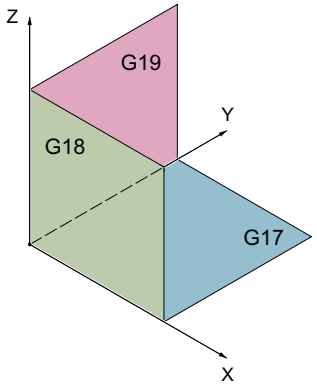
A blank can be specified for simultaneous recording.

8.4 Basic principles

8.4.1 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

8.4.2 Programming a tool (T)

Calling a tool

1. You are in a part program
2. Press the "Select tool" softkey.

   The "Tool Selection" window opens.
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="WHEEL100"

- 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 "To program" softkey.

   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.

8.5 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 "Workpieces" 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.

8.6 Selection of the cycles via softkey

Overview of machining steps

The following softkeys are available to insert machining steps.

⇒

Programming technology functions 9

9.1 Programming contours

Function

The free contour programming allows you to create simple or complex contours. You define open or closed contours.

A contour comprises separate contour elements, whereby at least two and up to 250 elements result in a defined contour. Radii, chamfers and tangential transitions are available as contour transition elements.

The integrated contour calculator calculates the intersection points of the individual contour elements taking into account the geometrical relationships, which allows you to enter incompletely dimensioned elements.

You must always program the geometry of the contour before you program the technology.

9.1.1 Representation of the contour

G-code program

In the editor, the contour is represented in a program section using individual program blocks. If you open an individual block, then the contour is opened.

The cycle represents a contour as a program block in the program. If you open this block, the individual contour elements are listed symbolically and displayed in broken-line graphics.

Symbolic representation

The individual contour elements are represented by icons adjacent to the graphics window. They appear in the order in which they were entered.

Contour element	Icon	Meaning
Starting point		Starting point of the contour
Straight line up Straight line down		Straight line in 90° grid Straight line in 90° grid
Straight line left Straight line right		Straight line in 90° grid Straight line in 90° grid

Contour element	Icon	Meaning
Straight line in any direction		Straight line with any gradient
Arc right Arc left		Circle Circle
Pole		Polar coordinates
Finish contour	END	End of contour definition

The different colors of the icons indicate their status:

Foreground	Background	Meaning
Black	Blue	Cursor on new element
Black	Orange	Cursor on current element
Black	White	Normal element
Red	White	Element not currently evaluated (element will only be evaluated when it is selected with the cursor)

Graphical display

The progress of contour programming is shown in broken-line graphics while the contour elements are being entered.

When the contour element has been created, it can be displayed in different line styles and colors:

- Black: Programmed contour
- Orange: Current contour element
- Green dashed: Alternative element
- Blue dotted: Partially defined element

The scaling of the coordinate system is adjusted automatically to match the complete contour.

The position of the coordinate system is displayed in the graphics window.

9.1.2 Creating a new contour

Function

You must create a new contour for each contour that you want to create.

The contours are saved at that position in the program where they are defined.

Note

Ensure that the contours are positioned after the program end identification!

The first step in creating a contour is to specify a starting point. Enter the contour element. The contour processor then automatically defines the end of the contour.

If you alter the machining plane, the cycle will automatically adjust the associated starting point axes. You can enter any additional commands (max. 40 characters) in G-code format for the starting point.

Additional commands

You can program feedrates and M commands, for example, using additional G-code commands. You can enter the additional commands (max. 40 characters) in the extended parameter screens ("All parameters" softkey). However, make sure that the additional commands do not collide with the generated G code of the contour. Therefore, do not use any G-code commands of group 1 (G0, G1, G2, G3), no coordinates in the plane and no G-code commands that have to be programmed in a separate block.

Procedure

1. The part program to be executed has been created and you are in the editor.
2.

Press the "Contour" and "New contour" softkeys.
The "New Contour" input window opens.
3. Enter a contour name.
4.

Press the "Accept" softkey.
The input screen for the starting point of the contour appears. You can enter Cartesian or Polar coordinates.

Cartesian starting point

1. Select the machining plane and enter the contour starting point.
2. Enter any additional commands in G-code format, as required.
3.

Press the "Accept" softkey.
4. Enter the individual contour elements.

Polar starting point

1. Select the machining plane and press the "Pole" softkey.
2. Enter the pole position in Cartesian coordinates.
3. Enter the starting point for the contour in Polar coordinates.
4. Enter any additional commands in G-code format, as required.

5. Press the "Accept" softkey.
6. Enter the individual contour elements.

Parameter	Description	Unit
PL	Machining plane	
	Cartesian:	
X	Starting point X (abs)	mm
Y	Starting point Y (abs)	mm
	Polar:	
X	Position pole (abs)	mm
Y	Position pole (abs)	Degrees
Starting point		
L1	Distance to pole, end point (abs)	mm
ϕ1	Polar angle to the pole, end point (abs)	Degrees
Additional commands	The contour is finished in continuous-path mode (G64). As a result, contour transitions such as corners, chamfers or radii may not be machined precisely. If you wish to avoid this, then it is possible to use additional commands when programming. Example: For a contour, first program the X parallel straight line and then enter "G9" (non-modal exact stop) for the additional command parameter. Then program the Y-parallel straight line. The corner will be machined exactly, as the feedrate at the end of the X-parallel straight line is briefly zero.	

9.1.3 Creating contour elements

After you have created a new contour and specified the starting point, you can define the individual elements that make up the contour.

The following contour elements are available for the definition of a contour:

- Straight vertical line
- Straight horizontal line
- Diagonal line
- Circle/arc
- Pole

For each contour element, you must parameterize a separate parameter screen.

The coordinates for a horizontal or vertical line are entered in Cartesian format; however, for the contour elements Diagonal line and Circle/arc you can choose between Cartesian and Polar coordinates. If you wish to enter Polar coordinates you must first define a pole. If you have already defined a pole for the starting point, you can also refer the Polar coordinates to this pole. Therefore, in this case, you do not have to define an additional pole.

Parameter entry

Parameter entry is supported by various help screens that explain the parameters.

If you leave certain fields blank, the geometry processor assumes that the values are unknown and attempts to calculate them from other parameters.

Conflicts may result if you enter more parameters than are absolutely necessary for a contour. In such a case, try to enter fewer parameters and allow the geometry processor to calculate as many parameters as possible.

Contour transition elements

As a transition between two contour elements, you can choose a radius or a chamfer. The transition element is always attached at the end of a contour element. The contour transition element is selected in the parameter screen of the respective contour element.

You can use a contour transition element whenever there is an intersection between two successive elements which can be calculated from the input values. Otherwise you must use the straight/circle contour elements.

The contour end is an exception. Although there is no intersection to another element, you can still define a radius or a chamfer as a transition element for the blank.

If the value of the transition element is "NULL", no transition element will be parameterized.

Additional functions

The following additional functions are available for programming a contour:

- Tangent to preceding element
  You can program the transition to the preceding element as tangent.
- Dialog box selection
  If two different possible contours result from the parameters entered previously, select the desired contour.

  Confirm the selection.

- Close contour
  From the current position, you can close the contour with a straight line to the starting point.

9.1.3.1 Entering a contour element

After you have created a new contour and specified the starting point, you can define the individual elements that make up the contour.

Creating contour elements

For each contour element, you must parameterize a separate parameter screen.

The coordinates for a horizontal or vertical line are entered in Cartesian format; however, for the contour elements Diagonal line and Circle/arc, you can choose between Cartesian and Polar coordinates.

If you want to enter Polar coordinates, first define a pole. If you have already defined a pole for the starting point, the Polar coordinates can also refer to this pole. You do not require any further pole in this case.

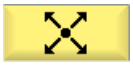
1. Open the part program and create a contour with the "Contour" and "New contour" softkeys.
2. Position the cursor at the desired entry position.
3. Press one of the softkeys to create a contour element.

For a horizontal straight line, the "Straight line (e.g. X or Y)" input window opens.

- OR -

For a vertical straight line, the "Straight line (e.g. Y or Z)" input window opens.

- OR -



For a diagonal straight line, the "Straight line (e.g. XY or ZX)" input window opens.

- OR -

For a circle/arc, the "Circle" input window opens.

- OR

For Polar coordinates, the "Pole input" input window opens.

4. Enter all the data available from the workpiece drawing in the appropriate input screens (e.g. length of straight line, target position, transition to next element, angle of lead, etc.).

5. Press the "Accept" softkey.

   The contour element is added to the contour.

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

   Press the "Tangent to prec. elem." softkey. The "tangential" selection appears in the parameter α2 entry field.

7. Repeat the procedure until the contour is complete.

8. Press the "Accept" softkey.
   The programmed contour is transferred to the program view.

9. If you want to display further parameters for certain contour elements, e.g. to enter additional commands, press the "All parameters" softkey.

9.1.3.2 Cylindrical grinding

Parameter	Description		Unit
X	End point X (abs or inc)		mm
α1	Starting angle, e.g. to the X axis (only for information)		Degrees
α2	Angle to the predecessor element (only for information)		Degrees
Transition to next element	Type of transition • Radius • Chamfer		
Radius	R	Transition to following element - radius	mm
Chamfer	FS	Transition to following element - chamfer	mm
Additional commands	Additional G-code commands		

Parameter	Description		Unit
Z	End point Z (abs or inc)		mm
α1	Starting angle, e.g. to the Z axis (only for information)		Degrees
Transition to next element	Type of transition • Radius • Chamfer		
Radius	R	Transition to following element - radius	mm
Chamfer	FS	Transition to following element - chamfer	mm
Additional commands	Additional G-code commands		

Contour element "Circle"

Parameter	Description	Unit
Direction of rotation	 • Clockwise direction of rotation • Counterclockwise direction of rotation	
R	Radius	mm
e.g. X	End point X (abs or inc)	mm
e.g. Z	End point Z (abs or inc)	mm
e.g. I	Circle center point I (abs or inc)	mm
e.g. J	Circle center point J (abs or inc)	mm

Parameter	Description		Unit
α1	Starting angle to X axis		Degrees
α2	Angle to the preceding element		Degrees
β1	End angle to Z axis		Degrees
β2	Opening angle		Degrees
Transition to next element	Type of transition • Radius • Chamfer		
Radius	R	Transition to following element - radius	mm
Chamfer	FS	Transition to following element - chamfer	mm
Additional commands	Additional G-code commands		

Parameter	Description		Unit
X	End point X (abs or inc)		mm
Z	End point Z (abs or inc)		mm
L	Length		mm
α1	Starting angle, e.g. to the X axis		Degrees
α2	Angle to the preceding element		Degrees
Transition to next element	Type of transition • Radius • Chamfer		
Radius	R	Transition to following element - radius	mm
Chamfer	FS	Transition to following element - chamfer	mm
Additional commands	Additional G-code commands		

Contour element "Pole"

Parameter	Description	Unit
X	Position pole (abs)	mm
Z	Position pole (abs)	mm

Contour element "End"

The data for the transition at the contour end of the previous contour element is displayed in the "End" parameter screen.

The values cannot be edited.

9.1.3.3 Surface grinding

Parameter	Description	Unit
Z	End point Z (abs or inc)	mm
α1	Starting angle to Z axis (only for information)	Degrees

Parameter	Description			Unit
α2	Angle to the preceding element			Degrees
Transition to next element	Type of transition • Radius • Undercut • Chamfer			
Radius	R	Transition to following element - radius		mm
Undercut	Form E	Undercut size e.g. E1.0x0.4		
	Form F	Undercut size e.g. F0.6x0.3		
	DIN thread	P α	Thread pitch Insertion angle	mm/rev Degrees
	Thread	Z1 Z2 R1 R2 T	Length Z1 Length Z2 Radius R1 Radius R2 Insertion depth	mm mm mm mm mm
Chamfer	FS	Transition to following element - chamfer		mm
CA	Grinding allowance • Grinding allowance to right of contour • Grinding allowance to left of contour			mm
Additional commands	Additional G-code commands			

Parameter	Description			Unit
Y	End point Y ∅ (abs or end point Y (inc))			mm
α1	Starting angle, e.g. to the Y axis (only for information)			Degrees
α2	Angle to the preceding element			Degrees
Transition to next element	Type of transition • Radius • Undercut • Chamfer			
Radius	R	Transition to following element - radius		mm
Undercut	Form E	Undercut size e.g. E1.0x0.4		
	Form F	Undercut size e.g. F0.6x0.3		
	DIN thread	P α	Thread pitch Insertion angle	mm/rev Degrees
	Thread	Z1 Z2 R1 R2 T	Length Z1 Length Z2 Radius R1 Radius R2 Insertion depth	mm mm mm mm mm
Chamfer	FS	Transition to following element - chamfer		mm

Parameter	Description	Unit
CA	Grinding allowance • Grinding allowance to right of contour • Grinding allowance to left of contour	mm
Additional commands	Additional G-code commands	

Parameter	Description		Unit
Z	End point Z (abs or inc)		mm
Y	End point Z ∅ (abs or end point Z (inc))		mm
α1	Starting angle, e.g. to the Z axis (only for information)		Degrees
α2	Angle to the preceding element		Degrees
Transition to next element	Type of transition • Radius • Chamfer		
Radius	R	Transition to following element - radius	mm
Chamfer	FS	Transition to following element - chamfer	mm
CA	Grinding allowance • Grinding allowance to right of contour • Grinding allowance to left of contour		mm
Additional commands	Additional G-code commands		

Contour element "Circle"

Parameter	Description		Unit
Direction of rotation	• Clockwise direction of rotation • Counterclockwise direction of rotation 		
Z	End point Z (abs or inc)		mm
X	End point Y ∅ (abs or end point Y (inc))		mm
K	Circle center point K (abs or inc)		mm
I	Circle center point I ∅ (abs or circle center point I (inc))		mm
α1	Starting angle to the Z axis (only for information)		Degrees
β1	End angle to the Z axis (only for information)		Degrees
β2	Opening angle		Degrees
Transition to next element	Type of transition • Radius • Chamfer		
Radius	R	Transition to following element - radius	mm
Chamfer	FS	Transition to following element - chamfer	mm

Parameter	Description	Unit
CA	Grinding allowance • Grinding allowance to right of contour • Grinding allowance to left of contour	mm
Additional commands	Additional G-code commands	

Contour element "Pole"

Parameters	Description	Unit
Z	Position pole (abs)	mm
Y	Position pole (abs)	Degrees

Contour element "End"

The data for the transition at the contour end of the previous contour element is displayed in the "End" parameter screen.

The values cannot be edited.

9.1.4 Changing the contour

9.1.4.1 Overview

Function

You can change a previously created contour later.

Individual contour elements can be

- added,
- changed,
- inserted or
- deleted.

9.1.4.2 Modifying a contour element

Procedure for changing a contour element

1. Open the part program to be machined.
2. Open the contour.

3. With the cursor, select the program block where you want to change the contour. Open the geometry processor.
   The individual contour elements are listed.
4. Position the cursor at the position where a contour element is to be inserted or changed.
5. Select the desired contour element with the cursor.
6. Enter the parameters in the input screen or delete the element and select a new element.

7. Press the "Accept" softkey.
   The desired contour element is inserted in the contour or changed.

Procedure for deleting a contour element

1. Open the part program to be machined.
2. Open the contour.
3. Position the cursor on the contour element that you want to delete.

4. Press the "Delete element" softkey.

5. Press the "Delete" softkey.

Note

Ensure that the complete contour and the transition to the following element remain.

9.1.5 Contour call (CYCLE62)

9.1.5.1 Function

Function

The input creates a reference to the selected contour.

There are four ways to call the contour:

1. Contour name
   The contour is in the calling main program.
2. Labels
   The contour is in the calling main program and is limited by the labels that have been entered.

3. Subprogram
   The contour is located in a subprogram in the same workpiece.
4. Labels in the subprogram
   The contour is in a subprogram and is limited by the labels that have been entered.

9.1.5.2 Calling the cycle

Procedure

1. The part program to be executed has been created and you are in the editor.
2.

   Press the "Contour" and "Contour call" softkeys.

   The "Contour Call" input window opens.
3. Assign parameters to the contour selection.

9.1.5.3 Parameter

Parameter	Description	Unit
Contour selection 	• Contour name • Labels • Subprogram • Labels in the subprogram	
Contour name	CON: Contour name	
Labels	• LAB1: Label 1 • LAB2: Label 2	
Subprogram	PRG: Subprogram	
Labels in the subpro-gram	• PRG: Subprogram • LAB1: Label 1 • LAB2: Label 2	

Note

Calling via EXTCALL

Calling a part program via EXTCALL without EES: via "contour name" or "labels." This behavior is monitored in the cycle.

Contour calls via "subprogram" or "labels in subprogram" are only possible if EES is active.

9.2 Set dresser coordinate system - CYCLE435

9.2.1 Note regarding the dressing tool position

The cycle to calculate the dressing tool position is not programmed using an input screen form in SINUMERIK Operate.

Syntax

```
CYCLE435(<_T>, <_DD>, <S_TA>, <S_DA>, <S_AD>, <S_AL>, <S_PVD>,
<S_PVL>, <S_PD>, <S_PL>, <_AMODE>)
```

Parameter

No.	Parameter mask	Parameter internal	Data type	Meaning		
1		<_T>	STRING[32]	Tool name of the grinding wheel		
2		<_DD>	INT	Cutting edge number of the grinding wheel		
3		<S_TA>	STRING[32]	Dressing tool reference point - dressing tool name		
4		<S_DA>	INT	Cutting edge number of the dressing tool		
5		<S_AD>	REAL	Dressing value, diameter		
6		<S_AL>	REAL	Dressing value, face		
7		<S_PVD>	REAL	Form-truing offset, diameter		
8		<S_PVL>	REAL	Form-truing offset, face		
9		<S_PD>	REAL	Form-truing allowance, diameter		
10		<S_PL>	REAL	Form-truing allowance, face		
11		<_AMODE>	INT	Alternative mode		
				UNITS:	active tool at the end of the cycle	
					0 =	dressing tool active
					1 =	wheel active

9.2.2 Function

The cycle is used to activate a coordinate system for dressing. A decision can be made as to whether, after the cycle has been called, the transferred dressing tool or the transferred grinding tool is active. The dimension offsets are already taken into account in the cycle so that the required form-truing contour can be machined.

After the dressing contour has been machined, the activated coordinate system must be cleared again by calling the cycle without transfer parameter.

Sequence

In the cycle, the tool data of the tool that is not active is transferred into the cycle frame. This means that the dressing contour can be subsequently called relative to the geometry parameters.

After machining the dressing contour, the cycle frame is cleared by calling the cycle without transfer parameter.

Example

```
T="WHEEL"D1
CYCLE435(„WHEEL“,1,”DRESSER”,1,0.01,0.01,10,10,0,0,0)
G01 G64 F200
X=10
Z=10
…
;Machining the dressing contour
…
CYCLE435()
```

In the example, after the cycle, the “DRESSER” tool with cutting edge 1 is active.

Internally, an offset of 0.01 mm is taken into account in X and Z (for G18) and the contour itself is offset by 10 mm, so that workpiece drawing dimensions can be programmed. A form-truing allowance that can be taken into account is 0. After this, the dressing contour can be machined.

After machining, the cycle frame is cleared by calling CYCLE435() without transfer parameter.

9.3 Form-truing (CYCLE495)

The form-truing cycle is not programmed using an input screen form in SINUMERIK Operate.

Software option

For the function "Form-truing type: parallel to the axis" to be used, you need the "SINUMERIK Grinding Advanced" software option.

Syntax

```
CYCLE495(<_T>, <_DD>, <_SC>, <_F>, <_VARI>, <_D>, <_DX>, <_DZ>,
<S_PA>, <S_N>, <_DMODE>, <_AMODE>, <S_FW>, <S_HW>)
```

Parameter

No.	Parameter mask	Parameter internal	Data type	Meaning		
1		<_T>	STRING[20]	Tool name of the grinding wheel		
2		<_DD>	INT	Cutting edge number of the grinding wheel		
3		<_SC>	REAL	Lift-off distance for avoiding obstacles, incremental		
4		<_F>	REAL	Form-truing feedrate		
5		<_VARI>	INT	Machining type		
				UNITS:	Form-truing type	
					1 =	Parallel to the axis
					2 =	Parallel to the contour
				TENS:	Machining direction	
					0 =	Pulling Possible with cutting edge positions 1 to 4
					1 =	Pushing Possible with cutting edge positions 1 to 4
					2 =	Alternating Possible with cutting edge positions 1 to 8
					3 =	Start → end Possible with cutting edge positions 1 to 8
					4 =	End → start Possible with cutting edge positions 1 to 8
				HUNDREDS:	Infeed direction	
					1 =	Infeed X for G18 or Y- for G19
					2 =	Infeed X+ for G18 or Y+ for G19
					3 =	Infeed Z- for G18 and for G19
					4 =	Infeed Z+ for G18 and for G19

No.	Parameter mask	Parameter internal	Data type	Meaning		
6		<_D>	REAL	Dressing value for form-truing type parallel to the axis		
7		<_DX>	REAL	Dressing value X for G18 or Y for G19 for form-truing type parallel to the contour		
8		<_DZ>	REAL	Dressing value Z for G18 and G19 for form-truing type parallel to the contour		
9		<S_PA>	REAL	Form-truing allowance		
10		<S_N>	INT	Number of strokes in the form-truing program		
11		<_DMODE>	INT	Display mode		
				UNITS:	Machining plane G17/G18/G19	
					0 =	Compatibility, the plane effective before the cycle call remains active
					1 =	G17 (only active in the cycle)
					2 =	G18 (only active in the cycle)
					3 =	G19 (only active in the cycle)
12		<_AMODE>	INT	Alternative mode		
				UNITS:	Form-truing selection, new/continue	
					1 =	New
					2 =	Continue
				TENS:	Select form-truing allowance	
					0 =	From the rough contour to the lowest point of the contour
					1 =	From the rough contour to the highest point of the contour
13		<S_FW>	REAL	Clear angle of the dressing tool		
14		<S_HW>	REAL	Holder angle of the dressing tool		

Parameter S_N specifies how many strokes are generated in a form-truing program.

Note

Machining direction

For "alternating" machining direction, parameter S_N should be set to 1, so that the direction changes at each stroke. Otherwise, use values >1, to achieve shorter machining times.

In addition, the following setting data are used:

SD55880 $SCS_GRIND_CONT_RELEASE_ANGLE

SD55881 $SCS_GRIND_CONT_RELEASE_DIST

SD55884 $SCS_GRIND_CONT_BLANK_OFFSET

Additional information

Additional information can be found in the SINUMERIK Operate Commissioning Manual.

Function

With the cycle, you form-true grinding wheels using a dressing tool. A dressing tool must be selected as active tool.

The OEM or the end user is responsible for the management of this.

The cycle runs in the active coordinate system, i.e. the cycle does not change any tool offsets or work offsets.

You program the grinding wheel contour as G code ("free contour"). This can be located in a subroutine or in the main program between two labels. The contour is transferred to the form-truing cycle CYCLE495 using contour call cycle CYCLE62. Form-truing is carried out until the form-truing allowance has been machined.

You have the option of form-truing either parallel with the axis or parallel with the contour:

- For form-truing parallel with the contour, the contour is moved along with each stroke. Taking into account the unmachined part (blank), contour segments can be omitted while machining.
- For form-truing in parallel to the axis, the form is established through longitudinal cuts parallel to the machining axis.

Form-truing can be interrupted, and continued again at the point of interruption. However, between stopping and continuing, no additional form-truing may be started, no wheel changed and no power on carried out.

The actual form-truing allowance is saved to the channel-specific GUD variables S_GC_CONT_R[2] for diagnostic purposes/progress display and for continuing after an interruption.

Sequence

Main program:

- Selection of the dressing tool coordinate system
  - with CYCLE435 (Page 250)(,,,) or
  - with OEM or end user cycle
    As a consequence, a dressing tool must be selected as active tool.
- Prepositioning, dressing tool
  The position of the dressing tool should be selected so that the tool can approach the grinding wheel for form-truing without any risk of collision.
- Contour call with CYCLE62(,,,) (Page 248)
  With the contour call, the form of the grinding wheel is transferred to the form-truing cycle.
- Form-truing with CYCLE495(,,,)
  The grinding wheel form is trued using CYCLE495.
- Deselection of the dressing tool coordinate system
  - with CYCLE435 (Page 250)(,,,) or
  - with OEM or end user cycle

Example

```
;*Main program
T="WHEEL" D1
CYCLE435("WHEEL",1,"DRESSER_6",1,0,0,10,10,0,0,0)
G0 X10
Z-40
CYCLE62("CONTOUR",0,,)
CYCLE495("WHEEL",1,.5,100,131,0.01,,,0.345,100,0,11,90,85)
CYCLE435()
M30


;*
N10 G00 G90
N20 G01 X=2.5 Z=-37 F=100
N30 Z=-23.03906 F=100
N40 G03 X=0 Z=-23 CR=400 RND=0 F=50
N50 G03 X=0 Z=-3 CR=50 F=100
N60 G01 X=0 Z=-2.8
N70 Z=-0.2
N80 X=-1 F=50
N90 Z=2 F=100
N100 M17
```

9.4 Oscillating cycles (CYCLE4071 ... CYCLE4079)

9.4.1 Note regarding oscillating cycles

Oscillating cycles are not programmed using input screen forms in SINUMERIK Operate.

9.4.2 CYCLE4071 - longitudinal grinding with infeed at the reversal point

Syntax

```
CYCLE4071(<S_A>, <S_B>, <S_W>, <S_U>, <S_I>, <S_K>, <S_H>, <S_A1>,
<S_A2>)
```

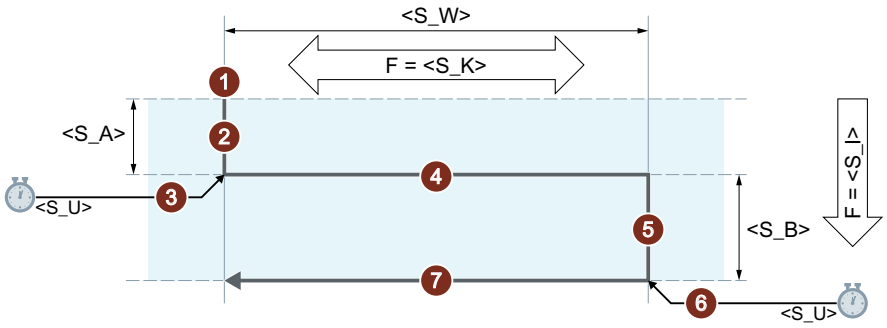
Parameters

No.	Parameter	Data type	Meaning
1	<S_A>	REAL	Infeed depth at the start
2	<S_B>	REAL	Infeed depth at the end
3	<S_W>	REAL	Grinding width
4	<S_U>	REAL	Sparking-out time
5	<S_I>	REAL	Feedrate for infeed
6	<S_K>	REAL	Feedrate for transverse infeed
7	<S_H>	INT	Number of repetitions
8	<S_A1>	AXIS	Infeed axis (optional) or 1st geometry axis
9	<S_A2>	AXIS	Oscillating axis (optional) or 2nd geometry axis

Function

The cycle is used for the execution of repeating infeeds. The infeed depth at the start and at the end can be different. There is a tangential motion between the infeeds.

Sequence



① Start of the cycle at the current position of the oscillating axis.

② Traversing of the infeed axis to the infeed depth at the start <S_A> with the feedrate for infeed <S_I>.

③ Sparking out with the sparking-out time <S_U>.

④ Traversing of the oscillating axis with the grinding width <S_W> as travel path and the feedrate for transverse infeed <S_K>.

⑤ Traversing of the infeed axis to the infeed depth at the end <S_B> with the feedrate for infeed <S_I>.

⑥ Sparking out with the sparking-out time <S_U>.

⑦ Traversing of the oscillating axis with the grinding width <S_W> as travel path to the starting point and the feedrate for transverse infeed <S_K>.

Indicates reiterating sequential steps.

The sequence is repeated until the programmed number of repetitions <S_H> has been reached.

Note

The sequence cannot be interrupted with a single block.

Example

Executing two oscillating motions with the following cycle parameters:

- Infeed depth at the start: 0.02 mm
- Infeed depth at the end: 0.01 mm
- Stroke: 100 mm
- Sparking-out time: 1 s
- Infeed feedrate: 1 mm/min
- Transverse feedrate: 1000 mm/min
- Repetitions: 2
- Oscillating and infeed axes: Standard geometry axes

Program code

```
N10 T1 D1
```

Program code

```
N20 CYCLE4071(0.02,0.01,100,1,1,1000,2)
N30 M30
```

9.4.3 CYCLE4072 - longitudinal grinding with infeed at the reversal point and cancel signal

Syntax

```
CYCLE4072(<S_GAUGE>, <S_A>, <S_B>, <S_W>, <S_U>, <S_I>, <S_K>,
<S_H>, <S_A1>, <S_A2>)
```

Parameters

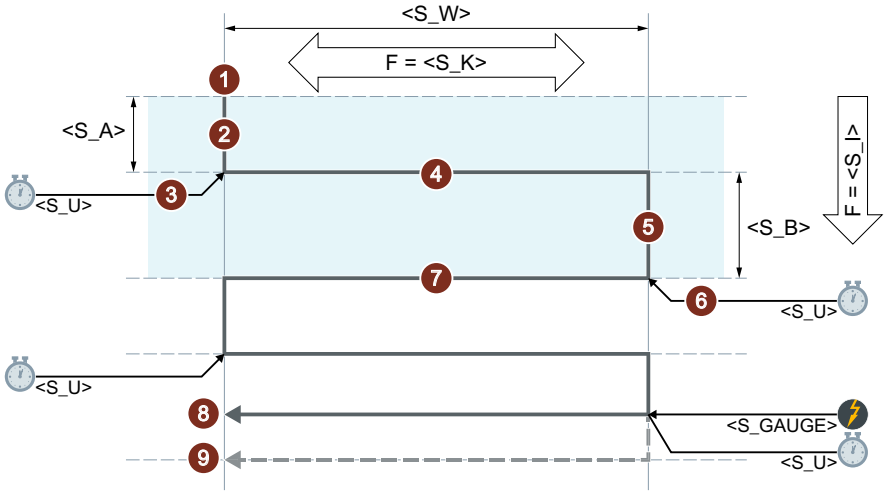
No.	Parameter	Data type	Meaning
1	<S_GAUGE>	STRING	Cancel conditions for infeed: 1. Number of a rapid input 2. Logical expression
2	<S_A>	REAL	Infeed depth at the start
3	<S_B>	REAL	Infeed depth at the end
4	<S_W>	REAL	Grinding width
5	<S_U>	REAL	Sparking-out time
6	<S_I>	REAL	Feedrate for infeed
7	<S_K>	REAL	Feedrate for transverse infeed
8	<S_H>	INT	Number of repetitions
9	<S_A1>	AXIS	Infeed axis (optional) or 1st geometry axis
10	<S_A2>	AXIS	Oscillating axis (optional) or 2nd geometry axis

Function

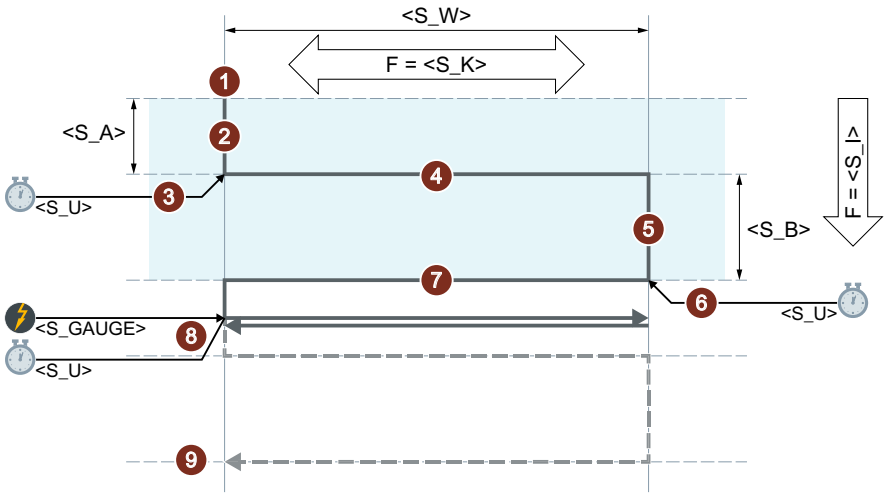
The cycle is used for the execution of repeating infeeds taking into account an external cancel signal. The infeed depth can be different at the start and at the end. There is a tangential motion between the infeeds. The depth infeed is cancelled when the cancel condition is satisfied. A complete stroke is always performed after the cancellation of the depth infeed.

Sequence

Cancellation of the infeed at the end



Cancellation of the infeed at the start



① Start of the cycle at the current position of the oscillating axis.

② Traversing of the infeed axis to the infeed depth at the start <S_A> with the feedrate for infeed <S_I>.

③ Sparking out with the sparking-out time <S_U>.

④ Traversing of the oscillating axis with the grinding width <S_W> as travel path and the feedrate for transverse infeed <S_K>.

⑤ Traversing of the infeed axis to the infeed depth at the end <S_B> with the feedrate for infeed <S_I>.

⑥ Sparking out with the sparking-out time <S_U>.

⑦ Traversing of the oscillating axis with the grinding width <S_W> as travel path to the starting point and the feedrate for transverse infeed <S_K>.

⑧ Cancel signal: The machining stops when the next start point is reached.

⑨ Without Cancel signal: The sequence is repeated until the programmed number of repetitions <S_H> has been reached.

Indicates reiterating sequential steps.

Note

The sequence cannot be interrupted with a single block.

Resources

As resources, the cycle uses a block-wide synchronized action and a synchronized action variable. The synchronized action is determined dynamically from the free area of the synchronized action range (CUS.DIR - 1 ..., CMA.DIR - 1000 ..., CST.DIR – 1199 ...). SYG_IS[1] is used as the synchronized action variable.

Examples

Example 1: Oscillation with two strokes:

Cycle parameters

- Infeed depth at the start: 0.02 mm
- Infeed depth at the end: 0.01 mm
- Stroke: 100 mm
- Sparking-out time: 1 s
- Infeed feedrate: 1 mm/min
- Transverse feedrate: 1000 mm/min
- Repetitions: 2
- Oscillating and infeed axes: Standard geometry axes

Cancel signal: Rapid input 1 ($A_IN[1])

Program code

```
N10 T1 D1
N20 CYCLE4072("1",0.02,0.01,100,1,1,1000,2)
N30 M30
```

Example 2: Oscillation with two strokes:

Cycle parameters

- Infeed depth at the start: 0.02 mm
- Infeed depth at the end: 0.01 mm
- Stroke: 100 mm
- Sparking-out time: 1 s
- Infeed feedrate: 1 mm/min
- Transverse feedrate: 1000 mm/min
- Repetitions: 2
- Oscillating and infeed axes: Standard geometry axes

Cancel signal: Variable $A_DBR[20] < 0.01

Program code

```
N10 T1 D1
N20 CYCLE4072("($A_DBR[20]<0.01)",0.02,0.01,100,1,1,1000,2)
N30 M30
```

9.4.4 CYCLE4073 - longitudinal grinding with continuous infeed

Syntax

```
CYCLE4073(<S_A>, <S_B>, <S_W>, <S_U>, <S_K>, <S_H>, <S_A1>, <S_A2>)
```

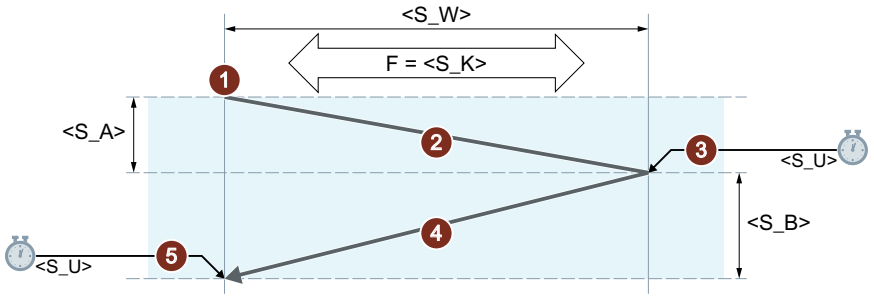
Parameters

No.	Parameter	Data type	Meaning
1	<S_A>	REAL	Infeed depth at the start
2	<S_B>	REAL	Infeed depth at the end
3	<S_W>	REAL	Grinding width
4	<S_U>	REAL	Sparking-out time
5	<S_K>	REAL	Feedrate for transverse infeed
6	<S_H>	INT	Number of repetitions
7	<S_A1>	AXIS	Infeed axis (optional) or 1st geometry axis
8	<S_A2>	AXIS	Oscillating axis (optional) or 2nd geometry axis

Function

The cycle is used for the execution of repeating infeeds. The infeed from the start to the end and from the end to the start can be different.

Sequence



① Start of the cycle at the current position of the oscillating axis with infeed depth 0.

② Traversing of the oscillating axis with the grinding width <S_W> as travel path and feedrate for transverse infeed <S_K> with continuous increase in the infeed depth up to the infeed depth at the start <S_A>.

③ Sparking out with the sparking-out time <S_U>.

④ Traversing of the oscillating axis with the grinding width <S_W> as travel path to the starting point and feedrate for transverse infeed <S_K> with continuous increase in the infeed depth up to the infeed depth at the end <S_B>.

⑤ Sparking out with the sparking-out time <S_U>.

Indicates reiterating sequential steps.

The sequence is repeated until the programmed number of repetitions <S_H> has been reached.

Note

The sequence cannot be interrupted with a single block.

Example

Oscillation with two strokes:

Cycle parameters

- Infeed depth at the start: 0.02 mm
- Infeed depth at the end: 0.01 mm
- Stroke: 100 mm
- Sparking-out time: 1 s
- Transverse feedrate: 1000 mm/min
- Repetitions: 2
- Oscillating and infeed axes: Standard geometry axes

Program code

```
N10 T1 D1
N20 CYCLE4073(0.02,0.01,100,1,1000,2)
N30 M30
```

9.4.5 CYCLE4074 - longitudinal grinding with continuous infeed and cancel signal

Syntax

```
CYCLE4074(<S_GAUGE>, <S_A>, <S_B>, <S_W>, <S_U>, <S_K>, <S_H>,
<S_A1>, <S_A2>)
```

Parameters

No.	Parameter	Data type	Meaning
1	<S_GAUGE>	STRING	Cancel conditions for infeed: 1. Number of a rapid input 2. Logical expression
2	<S_A>	REAL	Infeed depth at the start
3	<S_B>	REAL	Infeed depth at the end
4	<S_W>	REAL	Grinding width
5	<S_U>	REAL	Sparking-out time
6	<S_K>	REAL	Feedrate for transverse infeed

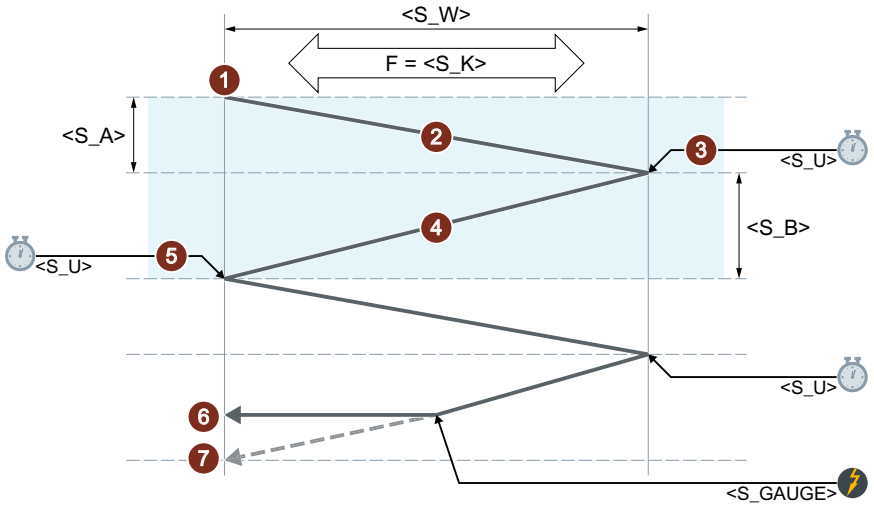
No.	Parameter	Data type	Meaning
7	<S_H>	INT	Number of repetitions
8	<S_A1>	AXIS	Infeed axis (optional) or 1st geometry axis
9	<S_A2>	AXIS	Oscillating axis (optional) or 2nd geometry axis

Function

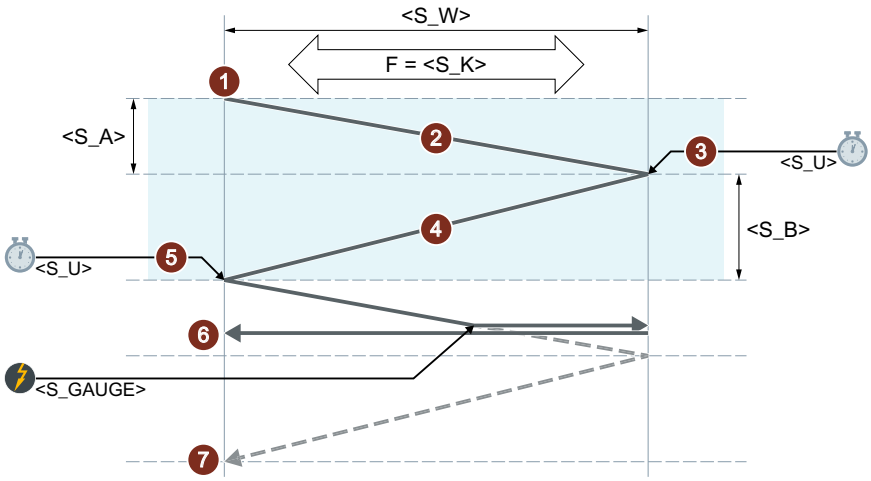
The cycle is used for the execution of repeating infeeds taking into account e.g. an external cancel signal. The infeed depth can be different at the start and at the end. The depth infeed is cancelled when the cancel condition is satisfied. A complete stroke is always performed after the cancellation of the depth infeed.

Sequence

Cancellation of the infeed from the end to the start



Cancellation of the infeed from the start to the end



① Start of the cycle at the current position of the oscillating axis with infeed depth 0.

② Traversing of the oscillating axis with the grinding width <S_W> as travel path and feedrate for transverse infeed <S_K> with continuous increase in the infeed depth up to the infeed depth at the start <S_A>.

③ Sparking out with the sparking-out time <S_U>.

④ Traversing of the oscillating axis with the grinding width <S_W> as travel path to the starting point and feedrate for transverse infeed <S_K> with continuous increase in the infeed depth up to the infeed depth at the end <S_B>.

⑤ Sparking out with the sparking-out time <S_U>.

⑥ Cancel signal: The depth infeed is canceled. The machining stops when the next start point is reached.

⑦ Without Cancel signal: The sequence is repeated until the programmed number of repetitions <S_H> has been reached.

Indicates reiterating sequential steps.

Note

The sequence cannot be interrupted with a single block.

Resources

As resources, the cycle uses a block-wide synchronized action and a synchronized action variable. The synchronized action is determined dynamically from the free area of the synchronized action range (CUS.DIR - 1 ..., CMA.DIR - 1000 ..., CST.DIR – 1199 ...). SYG_IS[1] is used as the synchronized action variable.

Examples

Example 1: Oscillation with two strokes:

Cycle parameters

- Infeed depth at the start: 0.02 mm
- Infeed depth at the end: 0.01 mm
- Stroke: 100 mm
- Sparking-out time: 1 s
- Transverse feedrate: 1000 mm/min
- Repetitions: 2
- Oscillating and infeed axes: Standard geometry axes

Cancel signal: Rapid input 1 ($A_IN[1])

Program code

```
N10 T1 D1
N20 CYCLE4074("1",0.02,0.01,100,1,1000,2)
N30 M30
```

Example 2: Oscillation with two strokes:

Cycle parameters

- Infeed depth at the start: 0.02 mm
- Infeed depth at the end: 0.01 mm
- Stroke: 100 mm
- Sparking-out time: 1 s
- Transverse feedrate: 1000 mm/min
- Repetitions: 2
- Oscillating and infeed axes: Standard geometry axes

Cancel signal: Variable $A_DBR[20] < 0.01

Program code

```
N10 T1 D1
N20 CYCLE4074("($A_DBR[20]<0.01)",0.02,0.01,100,1,1000,2)
N30 M30
```

9.4.6 CYCLE4075 - surface grinding with infeed at the reversal point

Syntax

```
CYCLE4075(<S_I>, <S_J>, <S_K>, <S_A>, <S_R>, <S_F>, <S_P>, <S_A1>,
<S_A2>)
```

Parameters

No.	Parameter	Data type	Meaning
1	<S_I>	REAL	Infeed depth at the start
2	<S_J>	REAL	Infeed depth at the end
3	<S_K>	REAL	Total infeed depth
4	<S_A>	REAL	Grinding width
5	<S_R>	REAL	Feedrate for infeed
6	<S_F>	REAL	Feedrate for transverse infeed
7	<S_P>	REAL	Sparking-out time
8	<S_A1>	AXIS	Infeed axis (optional)
9	<S_A2>	AXIS	Oscillating axis (optional)

Function

The cycle is used for machining with a total infeed depth in infeed steps. The infeed depths at the start and at the end can be different. There is a tangential motion between the infeeds.

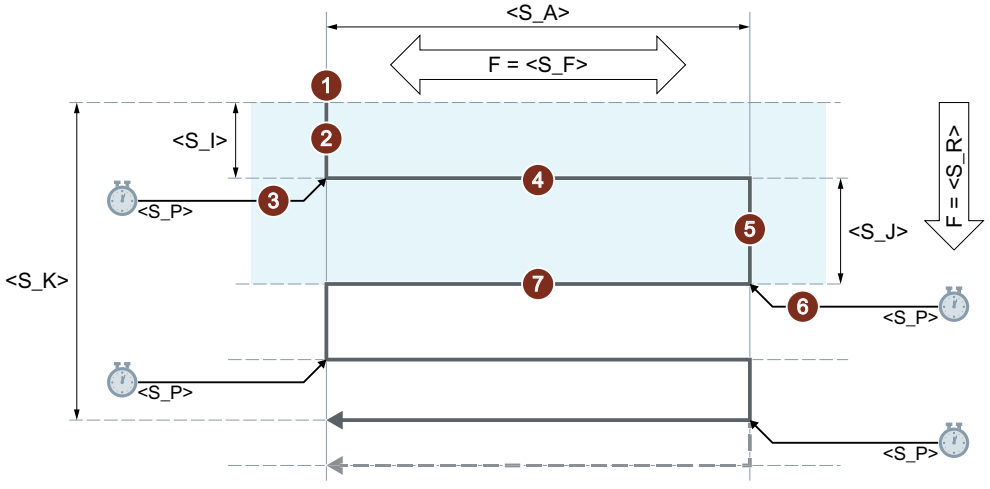
The positional data P1 to P4 can be negative or positive.

The specification of the infeed axis and/or oscillating axis is optional. If one or both parameters are not specified, the cycle uses the first two geometry axes of the channel.

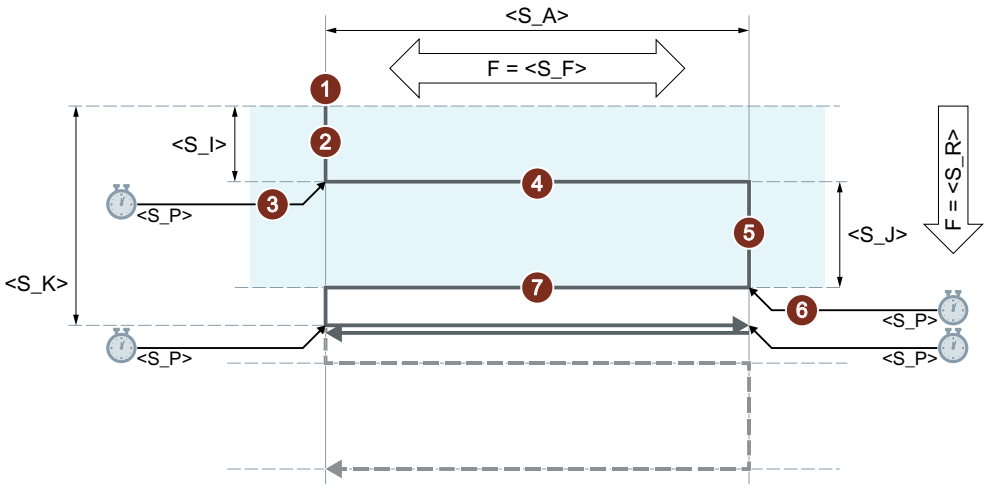
If the sum of the infeed depth at the start and end is 0 or the total infeed depth is 0, only one sparking-out stroke is performed.

Sequence

Total infeed depth reached with infeed at the second reversal point



Total infeed depth reached with infeed at the first reversal point



① Start of the cycle at the current position of the oscillating axis.

② Traversing of the infeed axis to the infeed depth at the start <S_I> with the feedrate for infeed <S_R>.

③ Sparking out with the sparking-out time <S_P>.

④ Traversing of the oscillating axis with the grinding width <S_A> as travel path and the feedrate for transverse infeed <S_F>.

⑤ Traversing of the infeed axis to the infeed depth at the end <S_J> with the feedrate for infeed <S_R>.

⑥ Sparking out with the sparking-out time <S_P>.

⑦ Traversing of the oscillating axis with the grinding width <S_A> as travel path to the starting point and the feedrate for transverse infeed <S_F>.

Indicates reiterating sequential steps.

The sequence is repeated until the total infeed depth <S_K> has been reached. The last stroke is then distributed unevenly.

Note

The sequence cannot be interrupted with a single block.

Example

Oscillation with:

- 0.02 mm infeed depth at the start
- 0.01 mm infeed depth at the end
- Total infeed depth 1 mm
- 100 mm stroke
- Infeed feedrate 1 mm/min
- Transverse feedrate 1000 mm/min

- 1 second sparking-out time
- Standard geometry axes

```
Program code
N10 T1 D1
N20 CYCLE4075(0.02,0.01,1,100,1,1000,1)
N30 M30
```

9.4.7 CYCLE4077 - surface grinding with infeed at the reversal point and cancel signal

Syntax

```
CYCLE4077(<S_GAUGE>, <S_I>, <S_J>, <S_K>, <S_A>, <S_R>, <S_F>,
<S_P>, <S_A1>, <S_A2>)
```

Parameters

No.	Parameter	Data type	Meaning
1	<S_GAUGE>	STRING	Cancel condition for infeed: • Number of a rapid input • Logical expression
2	<S_I>	REAL	Infeed depth at the start
3	<S_J>	REAL	Infeed depth at the end
4	<S_K>	REAL	Total infeed depth
5	<S_A>	REAL	Grinding width
6	<S_R>	REAL	Feedrate for infeed
7	<S_F>	REAL	Feedrate for transverse infeed
8	<S_P>	REAL	Sparking-out time
9	<S_A1>	AXIS	Infeed axis (optional)
10	<S_A2>	AXIS	Oscillating axis (optional)

Function

The cycle is used for machining with a total infeed depth in infeed steps. The infeed depths at the start and at the end can be different. There is a tangential motion between the infeeds. The depth infeed is cancelled when the cancel signal of the rapid input is 1 or the cancel condition is satisfied. A complete stroke is performed after the cancellation.

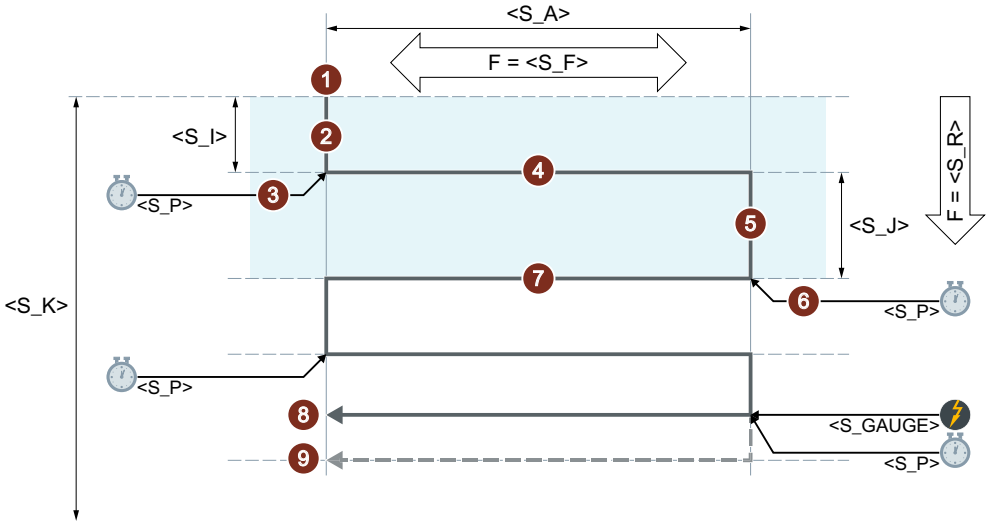
The positional data P2 to P5 can be negative or positive.

The specification of the infeed axis and/or oscillating axis is optional. If one or both parameters are not specified, the cycle uses the first two geometry axes of the channel.

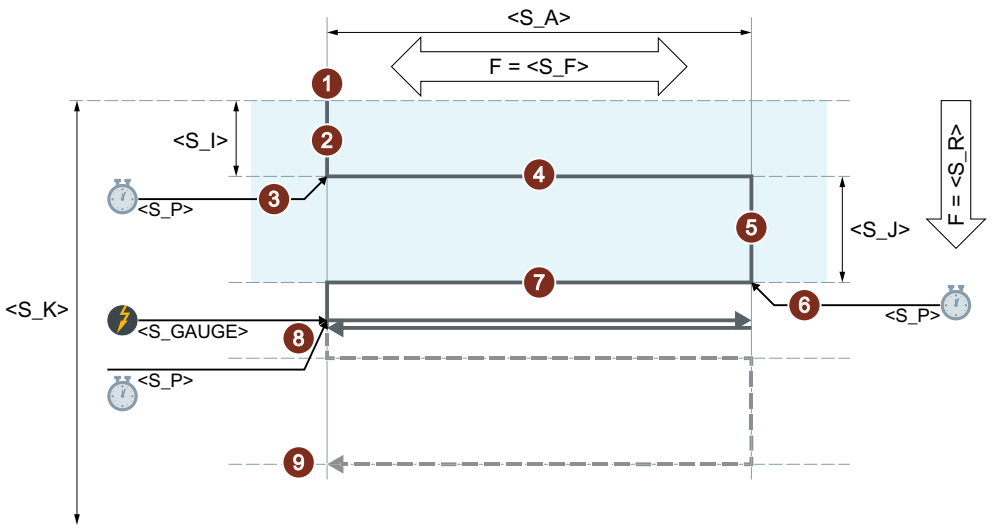
If the sum of the infeed depth at the start and end is 0 or the total infeed depth is 0, only one sparking-out stroke is performed.

Sequence

Cancellation of the infeed at the end



Cancellation of the infeed at the start



① Start of the cycle at the current position of the oscillating axis.

② Traversing of the infeed axis to the infeed depth at the start <S_I> with the feedrate for infeed <S_R>.

③ Sparking out with the sparking-out time <S_P>.

④ Traversing of the oscillating axis with the grinding width <S_A> as travel path and the feedrate for transverse infeed <S_F>.

⑤ Traversing of the infeed axis to the infeed depth at the end <S_J> with the feedrate for infeed <S_R>.

⑥ Sparking out with the sparking-out time <S_P>.

⑦ Traversing of the oscillating axis with the grinding width <S_A> as travel path to the starting point and the feedrate for transverse infeed <S_F>.

⑧ Cancel signal: The machining stops when the next start point is reached.

⑨ Without Cancel signal: The sequence is repeated until the total infeed depth <S_K> has been reached. The last stroke is then distributed unevenly.

Indicates reiterating sequential steps.

Note

The sequence cannot be interrupted with a single block.

Resources

As resources, the cycle uses a block-wide synchronized action and a synchronized action variable. The synchronized action is determined dynamically from the free area of the synchronized action range (CUS.DIR - 1 ..., CMA.DIR - 1000 ..., CST.DIR – 1199 ...). SYG_IS[1] is used as the synchronized action variable.

Examples

Example 1

Oscillation with:

- 0.02 mm infeed depth at the start
- 0.01 mm infeed depth at the end
- Total infeed depth 1 mm
- 100 mm stroke
- Infeed feedrate 1 mm/min
- Transverse feedrate 1000 mm/min
- 1 second sparking-out time
- Standard geometry axes

Cancel signal: Rapid input 1 ($A_IN[1])

Program code

```
N10 T1 D1
N20 CYCLE4077("1",0.02,0.01,1,100,1,1000,1)
N30 M30
```

Example 2

Oscillation with:

- 0.02 mm infeed depth at the start
- 0.01 mm infeed depth at the end
- Total infeed depth 1 mm
- 100 mm stroke
- Infeed feedrate 1 mm/min
- Transverse feedrate 1000 mm/min
- 1 second sparking-out time
- Standard geometry axes

Cancel signal: Dual-port RAM variable 20 less than 0.01 ($A_DBR[20] < 0.01)

Program code

```
N10 T1 D1
N20 CYCLE4077("($A_DBR[20]<0.01)",0.02,0.01,1,100,1,1000,1)
N30 M30
```

9.4.8 CYCLE4078 - surface grinding with continuous infeed

Syntax

```
CYCLE4078(<S_I>, <S_J>, <S_K>, <S_A>, <S_F>, <S_P>, <S_A1>, <S_A2>)
```

Parameters

No.	Parameter	Data type	Meaning
1	<S_I>	REAL	Infeed depth from the start to the end
2	<S_J>	REAL	Infeed depth from the end to the start
3	<S_K>	REAL	Total infeed depth
4	<S_A>	REAL	Grinding width
5	<S_F>	REAL	Feedrate
6	<S_P>	REAL	Sparking-out time
7	<S_A1>	AXIS	Infeed axis (optional)
8	<S_A2>	AXIS	Oscillating axis (optional)

Function

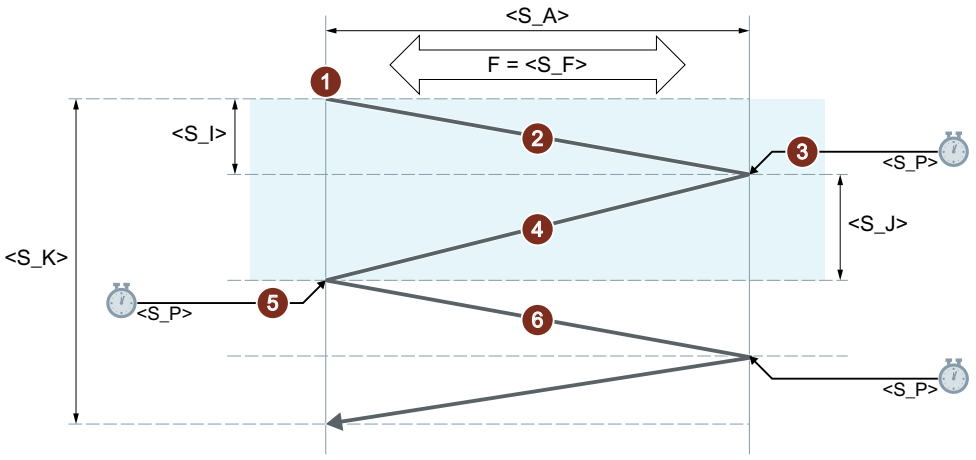
The cycle is used for machining with a total infeed depth by means of continuous infeed. The infeed depths from the start to the end and from the end to the start can be different.

The positional data P1 to P4 can be negative or positive.

The specification of the infeed axis and/or oscillating axis is optional. If one or both parameters are not specified, the cycle uses the first two geometry axes of the channel.

If the sum of the infeed depths P1 and P2 is 0 or the total infeed depth is 0, only one sparking-out stroke is performed.

Sequence



① Start of the cycle at the current position of the oscillating axis with infeed depth 0.

② Traversing of the oscillating axis with the grinding width <S_A> as travel path and feedrate <S_F> with continuous increase in the infeed depth up to the infeed depth at the start **<S_I>**.

③ Sparking out with the sparking-out time <S_P>.

④ Traversing of the oscillating axis with the grinding width <S_A> as travel path to the starting point and feedrate <S_F> with continuous increase in the infeed depth up to the infeed depth at the end <S_J>.

⑤ Sparking out with the sparking-out time <S_P>.

⑥ Traversing of the oscillating axis with the grinding width <S_A> as travel path to the starting point and feedrate <S_F>.

Indicates reiterating sequential steps.

The sequence is repeated until the total infeed depth <S_K> has been reached. The last stroke is then distributed unevenly.

Note

The sequence cannot be interrupted with a single block.

Example

Oscillation with:

- 20 mm infeed depth at the start
- 10 mm infeed depth at the end
- Total infeed depth 100 mm
- 100 mm stroke
- Feedrate 1000 mm/min
- 1 second sparking-out time
- Standard geometry axes

Program code

```
N10 T1 D1
N20 CYCLE4078(20,10,100,100,1000,1)
N30 M30
```

9.4.9 CYCLE4079 - surface grinding with intermittent infeed

Syntax

```
CYCLE4079(<S_I>, <S_J>, <S_K>, <S_A>, <S_R>, <S_F>, <S_P>, <S_A1>,
<S_A2>)
```

Parameters

No.	Parameter	Data type	Meaning
1	<S_I>	REAL	Infeed depth at the start
2	<S_J>	REAL	Infeed depth at the end
3	<S_K>	REAL	Total infeed depth
4	<S_A>	REAL	Grinding width
5	<S_R>	REAL	Feedrate for infeed
6	<S_F>	REAL	Feedrate for transverse infeed
7	<S_P>	REAL	Sparking-out time
8	<S_A1>	AXIS	Infeed axis (optional)
9	<S_A2>	AXIS	Oscillating axis (optional)

Function

The cycle is used for machining with a total infeed depth in infeed steps. The infeed depths at the start and at the end can be different. There is a tangential motion between the infeeds.

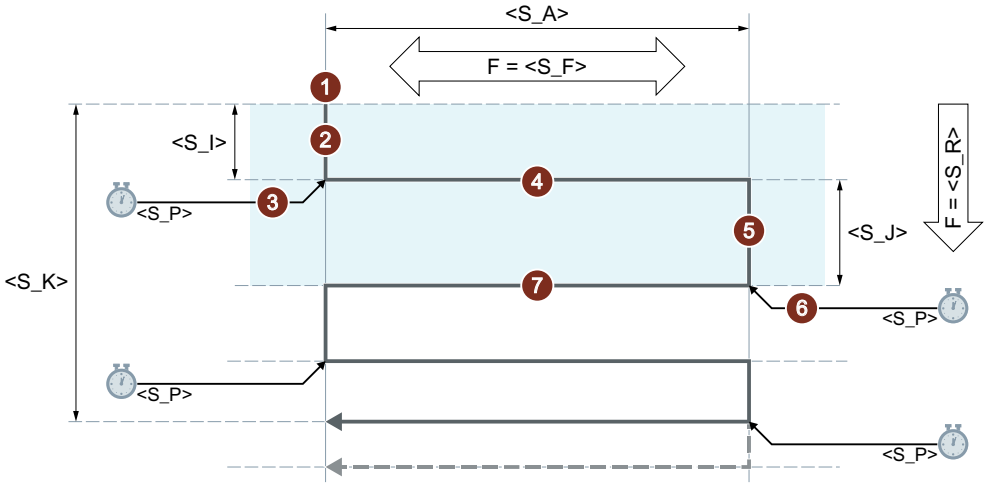
The positional data P1 to P4 can be negative or positive.

The specification of the infeed axis and/or oscillating axis is optional. If one or both parameters are not specified, the cycle uses the first two geometry axes of the channel.

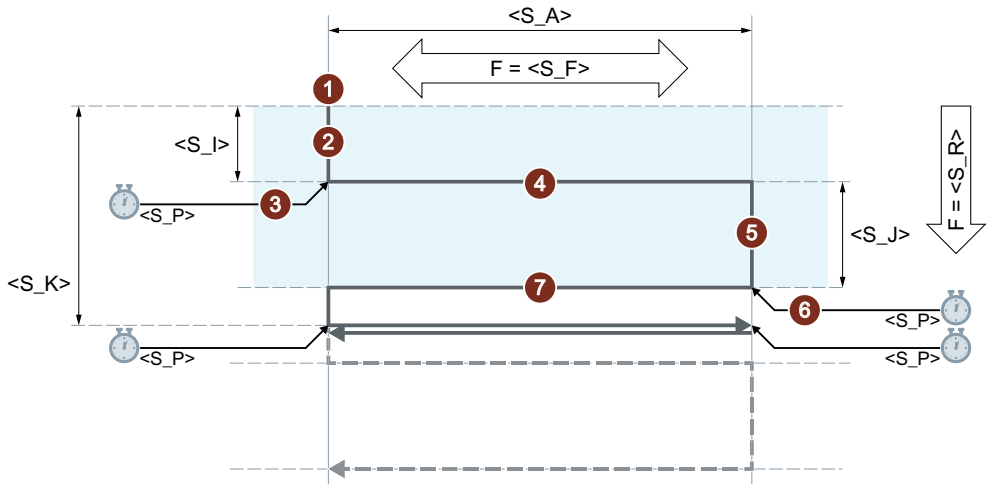
If the sum of the infeed depth at the start and end is 0 or the total infeed depth is 0, only one sparking-out stroke is performed.

Sequence

Total infeed depth reached with infeed at the second reversal point



Total infeed depth reached with infeed at the first reversal point



① Start of the cycle at the current position of the oscillating axis.

② Traversing of the infeed axis to the infeed depth at the start <S_I> with the feedrate for infeed <S_R>.

③ Sparking out with the sparking-out time <S_P>.

④ Traversing of the oscillating axis with the grinding width <S_A> as travel path and the feedrate for transverse infeed <S_F>.

⑤ Traversing of the infeed axis to the infeed depth at the end <S_J> with the feedrate for infeed <S_R>.

⑥ Sparking out with the sparking-out time <S_P>.

⑦ Traversing of the oscillating axis with the grinding width <S_A> as travel path to the starting point and the feedrate for transverse infeed <S_F>.

Indicates reiterating sequential steps.

The sequence is repeated until the total infeed depth <S_K> has been reached. The last stroke is then distributed unevenly.

Note

The sequence cannot be interrupted with a single block.

Example

Oscillation with:

- 0.02 mm infeed depth at the start
- 0.01 mm infeed depth at the end
- Total infeed depth 1 mm
- 100 mm stroke
- Infeed feedrate 1 mm/min
- Transverse feedrate 1000 mm/min

- 1 second sparking-out time
- Standard geometry axes

Program code

```
N10 T1 D1
N20 CYCLE4079(0.02,0.01,1,100,1,1000,1)
N30 M30
```

9.5 Aligning a grinding wheel (CYCLE400)

9.5.1 Function

Grinding machines with B axis that can be swiveled are supported using the "Aligning grinding wheel" functions.

The maximum angular range when aligning is limited by the traversing range of the participating rotary axes. Technological limits are also placed on the angular range depending on the tool used. After aligning, the cutting-edge position is automatically adapted.

Definition of angle ß

Angle ß – that is independent of the machine – is used to align grinding wheels.

In the initial state of the machine kinematics, a grinding wheel can be oriented according to Z or X.

Machining the opposite side

It can be selected as to whether the side of the grinding wheel that corresponds to the cutting-edge position is machined – or the opposite side.

Retraction

Retraction is possible before swiveling the grinding wheel.

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

9.5.2 Calling the cycle

Procedure

1. The part program to be executed has been created and you are in the editor.

2. Press the "Various" softkey.

3. Press the "Align grinding wheel" softkey.
   The "Align Grinding Wheel" window opens.

Parameter

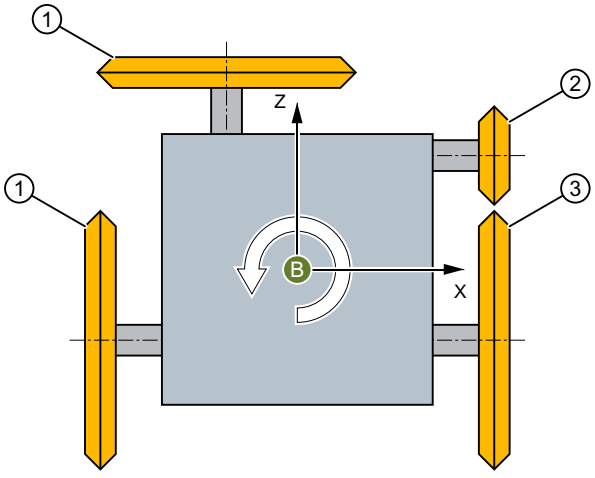
Parameter	Description	Unit
TC	Name of the swivel data set	
ß	Angle of the tool to the axis of rotation	Degrees
Machining the opposite side	• Yes: Machining opposite the cutting edge position • No: Machining on the cutting edge position side	
Retraction	• Yes: Retract prior to swiveling • No: Do no retract prior to swiveling	

Grinding with a B axis (only for cylindrical grinding machines)

10

10.1 Overview

Cylindrical grinding machines with a B axis are supported by means of tool carriers.



① External grinding wheel
② Internal grinding wheel
③ Face grinding wheel

Figure 10-1 Example: Turret with four grinding spindles

Tool carrier

A separate tool carrier is set up for each grinding spindle. Each tool carrier has head kinematics with the B axis as the first rotary axis. A semiautomatic rotary axis in the direction of the grinding spindle is set up as the second rotary axis (value: 0° or 180°).

The respective basic position is determined via the offset angle of the B axis (e.g. 0°, 90°, 180°, 270°, arbitrary). If a spindle is mechanically at a slightly inclined angle to the 90° direction (e.g. 3°), then the direction vector of the second rotary axis is also entered accordingly (however, the direction component Y must be 0).

Select the appropriate tool carrier in the "T, S, M" window.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

"Beta" alignment angle

You can define a skew angle in relation to the basic position with β.

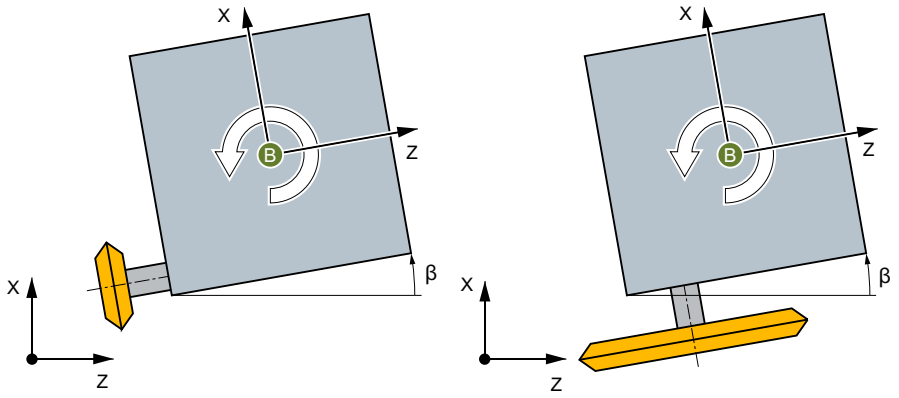
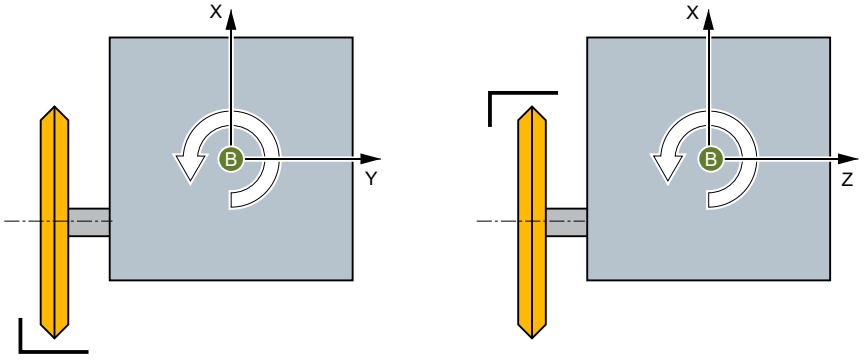


Figure 10-2 Beta rotation (B axis)

Changing the cutting edge position

The second rotary axis is controlled with the aid of the "Machining opposite" selection box, and the cutting edge position is changed via the CUTMOD function (e.g. internal grinding).



Machining opposite = no

Machining opposite = yes

10.2 T, S, M window for set-up B axis

Alignment of the B axis

Display	Meaning
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 (for replacement tool strategy)
TC 	Name of the swivel data set
ß	Input of the angle for the tool alignment
Machining opposite	• Yes: Machining opposite the cutting edge position • No: Machining on the cutting edge position side
Spindles 1 and 2 (e.g. S1)	Spindle selection for master spindle and 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.
G zero offset	Selection of the zero offset (basic reference, G54 - 57) You can select zero offsets from the tool list of settable zero offsets via the "Zero offset" softkey.
Measurement unit	Selection of the unit of measurement (inch, mm). The setting made here has an effect on the programming.
Machining plane	Selection of the machining plane (G17(XY), G18 (ZX), G19 (YZ))
Gear stage	Specification of the gear stage (auto, I - V)
Stop position	Input of the spindle position in 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 and continues to turn in the same direction.

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.

   - OR -

   Press the "Select tool" softkey to open the tool list, position the cursor on the desired tool and press the "In manual" softkey.

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

4. Enter the desired parameters.
5. Press the <CYCLE START> key.

   The tool is loaded into the spindle.

Note

Angular alignment and cutting edge position

The "Beta" and "Machining opposite" fields must always be entered together.

Note

Selection of the swivel data set

If only one swivel data set is available, the "TC" selection box is omitted.

Please also refer to the machine manufacturer's instructions.

10.3 Measuring in JOG

10.3.1 Grinding wheel alignment for grinding

The T, S, M window has input fields for aligning the grinding wheel if a tool carrier has been set up.

Aligning the tools for the B axis

- TC
  Name of the swivel data set
  Note: If only one swivel data set is available, the "TC" selection field is omitted.
- ß
  Input of the angle for the tool alignment
- Machining the opposite side
  Yes: Machining opposite the cutting edge position
  No: Machining on the cutting edge position side

10.3.2 Manually measuring the grinding tool (with B axis)

Reference point

A dresser serves as reference point for the measurement of length X or Z.

The reference point of the dresser can be represented by a zero offset or a dressing tool. This setting is permanently stored in the machine data and specified by the machine manufacturer.

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

Procedure

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

2. Press the "Meas. tool" softkey.

3. Press the "Measure wheel" softkey.

Select tool

4. Press the "Select tool" softkey.
   The "Tool Selection" window opens.

OK

5. In the "Tool Selection" window, select the grinding tool that you want to measure and press the "OK" softkey.
   The cutting edge location must be entered in the tool list.
   - OR -
   Tool list
   Press the "Tool list" softkey, select in the tool list the grinding tool that you want to measure and press the "In manual" softkey.

In manual

   The tool is transferred to the "Measure: "Grinding Wheel" window.

SELECT

6. Select the "Dresser" entry in the "Reference point" selection field.

Dresser as tool

Select dresser

7. Position the cursor in the "TR" field, press the "Select dresser" softkey, select the dresser for measuring the tool length and press the "OK" softkey.

OK

   - OR -

Dresser as zero offset

Select work offs.

7. Position the cursor to the "Zero offset" field and press the "Select ZO" softkey.

8. Select in the "Zero Offset - G54 ... G509" window the desired zero offset and press the "In manual" softkey.

In manual

X

9. Press the "X" or "Z" softkey, depending on which tool length you want to measure.

Z

10. Scratch the dresser using the tool.

Set length

11. Press the "Set length" softkey.
    The tool length is calculated automatically and entered in the tool list.
    The cutting edge location is considered automatically.

Note

Active grinding tool

Tool measurement is possible only with an active grinding tool.

10.3.3 Manually measuring the dresser (with B axis)

Reference point

A grinding wheel serves as reference point for measuring length X and length Z.

Procedure

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

2. Press the "Meas. tool" softkey.

3. Press the "Measure dresser" softkey.

Dresser as tool

4. Press the "Select dresser" softkey.
   The "Tool Selection" window opens.

5. In the "Tool Selection" window, select the dressing tool to be measured and press the "OK" softkey.
   The cutting edge location must be entered in the tool list.
   - OR -

   Press the "Tool list" softkey, select in the tool list the dressing tool to be measured and press the "In manual" softkey.
   The tool is transferred to the "Measure: Dresser" window.
   - OR -

Dresser as zero offset

4. Position the cursor in the "Zero offset" field and press the "Select ZO" softkey.

5. Select in the "Zero Offset - G54 ... G509" window the desired zero offset and press the "In manual" softkey.

6. Position the cursor in the "TR" field, press the "Select grinding wheel" softkey.

7. Select the grinding wheel to be used for measuring the dresser length and press the "OK" softkey.

8. Press the "X" or "Z" softkey, depending on which tool length you want to measure.

9. Scratch the dresser using the grinding tool.

10. Press the "Set length" softkey.

10.3.4 Calibrating the swivel axis

Requirement

- A tool carrier has been set up.
- The grinding wheel has been loaded.
- The swivel data set to be measured is active, swivel position β0 has been approached.

Note

Prior to calibration of the swivel axis, we recommend dressing the grinding wheel with β=0°.

Procedure

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

2. Press the "Meas. tool" softkey.

3. Press the "Swivel axis" softkey.

The "Calibration: Swivel Axis" window opens.

4. If multiple swivel data sets have been created, select the one desired in the "TC" selection field. Swivel position β0 is specified at 0°.

   In the "Machining opposite" selection field, select "Yes" or "No" to determine on which side of the cutting edge position the measurement is to be performed.

5. Select the measuring axis (X or Z).

   With measuring axis X, the diameter positions are used for the calculation, with measuring axis Z, the shoulder positions.

6. Press the <CYCLE START> key.

   The first β angle is automatically swung to the fixed default angle of 0°.

7. Then scratch the workpiece and press the "Save β0" softkey.

8. Retract the wheel manually.
9. Enter the required angles in the "ß1" or "ß2" field.

10. Press the <CYCLE START> key.
    β1 or β2 is automatically swung in.

11. Scratch the workpiece and press the "Save β1" or "Save β2" softkey.

12. Press the "Calculate" softkey.
    Offset vector I3 (X and Z) of the swivel data set is displayed as the measurement result. Internally, closing vector I1 is calculated and written to the tool carrier.

Note

Calibration is only possible with an active tool.

The tool length of the grinding wheel is used directly for the swivel arm calculation. This means that the lengths of the grinding wheel are exactly known.

If the lengths of the grinding wheel are not exactly known, then the grinding wheel and swivel arm form a single unit. With this unit it is not possible to tell whether the length comes from the wheel or the swivel arm. With this scenario, and if multiple grinding wheels are used, we recommend using one tool carrier per grinding wheel.

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

The collision monitoring is only valid for single-channel machines.

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, are not monitored and can therefore cause collisions.

Additional information

Further information about the kinematic chain and collision avoidance is provided in:

- Basic Functions Function Manual
- Monitoring and Compensating Function Manual

11.1 Activating collision avoidance

Precondition

- Collision avoidance is setup and an active machine model (kinematic chains) 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.

- OR -

4. Press the "Other views" and "Machine area" softkeys.

An active machine model (kinematic chains) is displayed for simultaneous recording.

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)

Multi-channel view

12

12.1 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

12.2 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.

12.3 Multi-channel view for large operator panels

On the OP015 and OP019 operator panels as well as on the PC, you have the option of displaying up to four channels next to each one. This simplifies the creation and run-in for multi-channel programs.

Constraints

- OP015 with a resolution of 1024x768 pixels: up to three channels visible
- OP019 with a resolution of 1280x1024 pixels: up to four channels visible
- The operation of a OP019 requires a PCU50.5

3- or 4-channel view in the "Machine" operating area

Use the multi-channel view settings to select the channels and specify the view.

Channel view	Display in the "Machine" operating area
3-channel view	The following windows are displayed one above the other for each channel: • Actual Value window • T,F,S window • Block Display window Selecting functions • The T,F,S window is overlaid by pressing one of the vertical softkeys.
4-channel view	The following windows are displayed one above the other for each channel: • Actual Value window • G functions (the "G functions" softkey is omitted). "All G functions" is accessed with the Menu forward key. • T,S,F window • Block Display window Selecting functions • The window showing the G codes is overlaid if you press one of the vertical softkeys.

Toggling between the channels

Press the <CHANNEL> key to toggle between the channels.

Press the <NEXT WINDOW> key to toggle within a channel column between the three or four windows arranged one above the other.

Note

2-channel display

Unlike the smaller operator panels, the T,F,S window is visible for a 2-channel view in the "Machine" operating area.

Program operating area

You can display as many as ten programs next to each other in the editor.

Displaying a program

You can define the width of the program in the Editor window using the settings in the editor. This means that you can distribute programs evenly - or you can widen the column with the active program .

Channel status

When required, channel messages are displayed in the status display.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

12.4 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.

Manage tools 13

13.1 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

13.2 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.

13.3 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.

Tool types

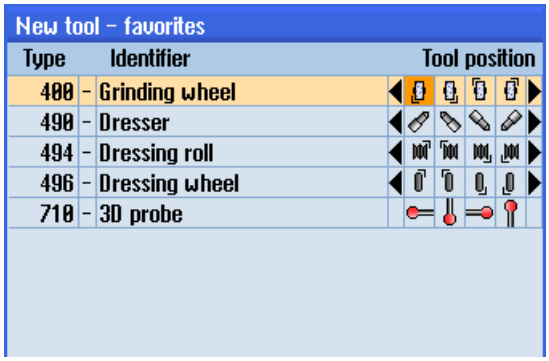


Figure 13-1 Example for the list of favorites (cylindrical grinding)

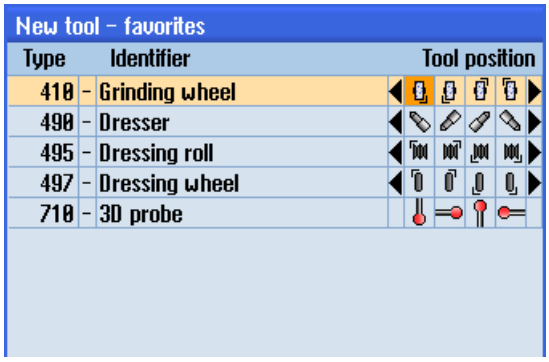


Figure 13-2 Example for the list of favorites (surface grinding)

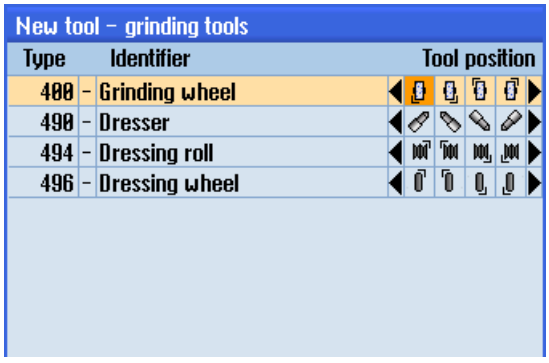


Figure 13-3 Tools listed in the "New Tool - Grinding Tools" window (cylindrical grinding)

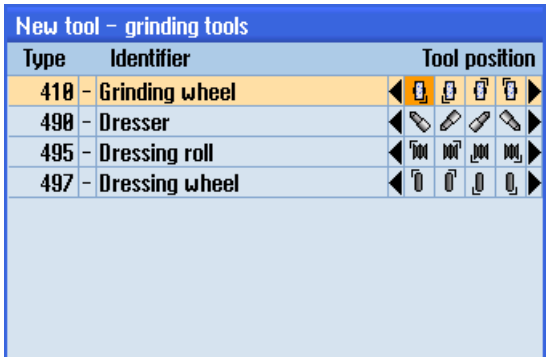


Figure 13-4 Tools listed in the "New Tool - Grinding Tools" window (surface grinding)

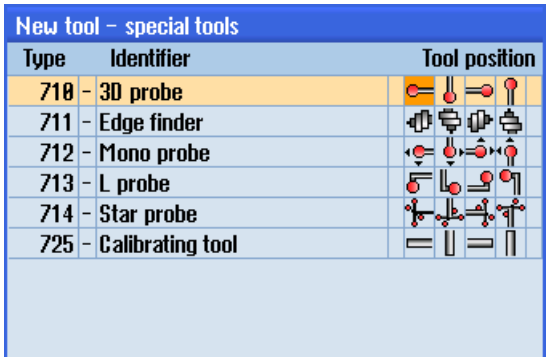


Figure 13-5 Available tools in the "New Tool - Special Tools" window

13.4 Tool dimensioning

This section provides an overview of the dimensioning of tools.

Tool types

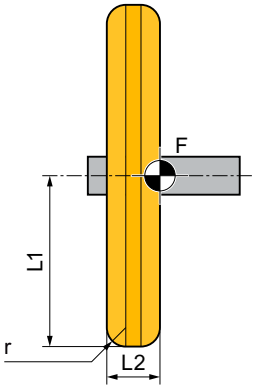


Figure 13-6 Grinding wheel

The following table shows the effect of the working planes:

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

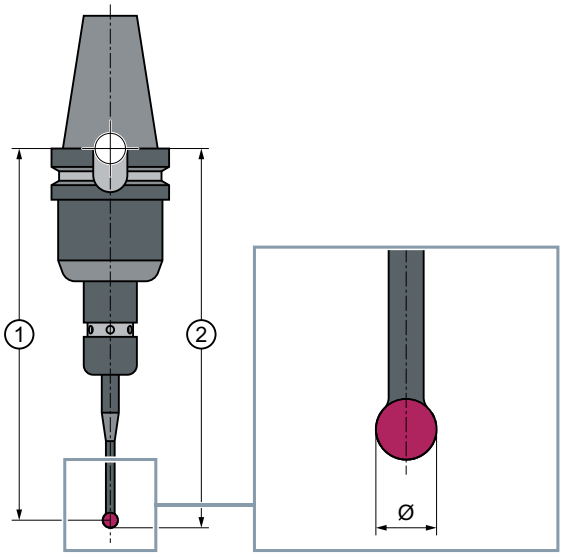


Figure 13-7 3D probe

Machine manufacturer

The tool length is measured to the center of the ball or to the ball circumference.

Please observe the information provided by the machine manufacturer.

Note

A 3D probe must be calibrated before use.

13.5 Tool list

13.5.1 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 sister tool number.

For the tool display, i.e. when displaying the cutting edge positions, the machine coordinate system is taken into account.

Tool parameters

Column heading	Meaning
Location W L B 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. • Transfer location • Loader • Loading station • Load position 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). The icon identifies the position of the tool; this was selected when the tool was created.
	You have the option of changing the tool position or the tool type using the <SELECT> key.
Tool name	The tool is identified by the name and the sister 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	Sister tool number (for replacement tool strategy).
D	Cutting edge number
Wheel Ø	Wheel diameter (for grinding wheel - type 400, type 410)

Column heading	Meaning
Length X or length Z - Cylindrical grinding	Tool length Geometry data length X or length Z
Length Y or length Z - Surface grinding	Tool length Geometry data length Y or length Z
Cutting edge radius	Tool radius (for grinding wheel - type 400, type 410; dresser - type 490, dressing roller - type 495, dressing wheel - type 497)
Ø	Tool diameter (for 3D-probe - type 710; edge-probe - type 711; mono-probe - type 712; L-probe - type 713; calibration tool - type 725)
Outer Ø	Outer diameter (for star-type probe - type 714)

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.

Further information

Additional information on configuring and setup of the tool list is provided in the Tool Management Function Manual.

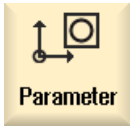
Icons in the tool list

Icon/ Marking		Meaning
Tool type		
Red cross		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 tool tip will provide a brief description.
Green frame		The tool is preselected.

Icon/ Marking		Meaning
Magazine/location number		
Green double arrow		The magazine location is positioned at the change position.
Gray double arrow		The magazine location is positioned at the load position.
Red cross		The magazine location is disabled.

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

Procedure



1. Select the "Parameter" operating area.

2. Press the "Tool list" softkey.
   The "Tool List" window opens.

13.5.2 Additional data

The following tool type requires geometry data that is not included in the tool list display.

Tool types with additional geometry data

Tool type	Additional parameters
710 3D-probe	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)
712 Mono-probe	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) Offset angle (angle)
713 L-probe	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) Offset angle (angle) Boom length (length)

Tool type	Additional parameters
714 Star-type probe	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) Offset angle (angle) Ball diameter (Ø)
725 Calibration tool	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)

You can use the configuration file to specify the data to be displayed in the "Additional Data" window.

Machine manufacturer

Please refer to 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.

13.5.3 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 requested grinding or special tool via the corresponding softkeys.

Note

Grinding tools

Additional grinding tools are available depending on the particular machine configuration.

Procedure

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 opens.

- OR -

If you want to create a tool that is not in the list of favorites, press "Grinders 400-499" or " Spec. tool 700-900".

The "New Tool - Grinding Tools" or "New Tool - Special Tools" window opens.

- or -

4. Select the tool by placing the cursor on the corresponding tool type and on the icon of the desired cutting edge position.

5. If more than four cutting edge positions are available, select the desired cutting edge position with the aid of the <cursor left> and <cursor right> keys.

6. 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 positions

If you have configured several load positions for a magazine, then the "Select Load Position" window opens when a tool is created directly in an empty magazine location or when the "Load" softkey is pressed.

Select the required load position 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:

- Name
- Tool location type
- Size of tool

Further information

Additional information on the configuration options is provided in the Tool Management Function Manual.

13.5.4 Measuring a tool - tool list

You can measure the tool offset data for the individual tools directly from the tool list.

Note

Tool measurement is only possible with an active tool.

Procedure

1. The tool list opens.

2. Select the tool that you want to measure in the tool list and press the "Measure tool" softkey.

You jump to the "JOG" operating area and the tool to be measured is entered in the "Measure: Length manual" screen in the "T" field.

3. Select the cutting edge number D and the sister tool number ST.

4. Press the "X", "Y" or "Z" softkey, depending on which tool length you want to measure.

...

5. Traverse the tool towards the workpiece in the direction that is to be measured and scratch it.
6. Enter the position of the workpiece edge in X0, Y0 or Z0.

   If no value is entered for X0, Y0 or Z0, the value is taken from the actual value display.

7. Press the "Set length" softkey.

   The tool length is calculated automatically and entered in the tool list.

13.5.5 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 control configuration.

Tool cutting edges that are not required can be deleted.

Procedure

1. The tool list opens.
2. Position the cursor on the tool for which you would like to store more cutting edges.

3. Press the "Cutting 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 1. The offset data is preassigned with the values of the cutting edge where the cursor is located.
5. Enter the offset data for the second 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.

13.5.6 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. Place the cursor on the tool that you would like to delete.

3. Press the "Delete tool" softkey.

   A safety prompt is displayed.

4. Press the "OK" softkey if you really want to delete the tool.

   Use this softkey to delete the tool.

   If the tool is in a magazine location, it is unloaded and then deleted.

Multiple load points - tool in magazine location

If you have 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.

13.5.7 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 tool list.

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 tool list in the NC memory outside the magazine.

Should you want to use the tool again later, simply load the tool with the tool data into the corresponding magazine location again. Then the same tool data does not have to be entered more than once.

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 pre-assigned 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.

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 "Load 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 "Load Point Selection" window.

3. Confirm your selection with "OK".

- OR -

Undo your selection with "Cancel".

13.5.8 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.

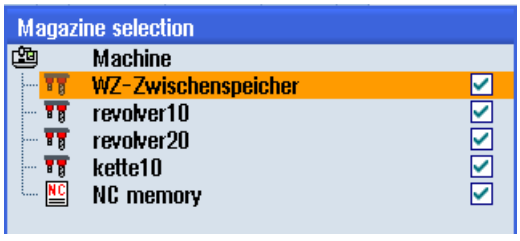
If there is only one magazine, you will move from one area to the next (i.e. from the buffer memory to the magazine, from the magazine to the NC memory, and from the NC memory back to the buffer memory) each time you press the softkey. The cursor is positioned at the beginning of the magazine each time.

- 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 configuration options is provided in the Tool Management Function Manual.

13.5.9 Code carrier connection

13.5.9.1 Overview

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.

13.5.9.2 Managing tools on a code carrier

With a code carrier connection, in the list of favorites, there is also a tool available.

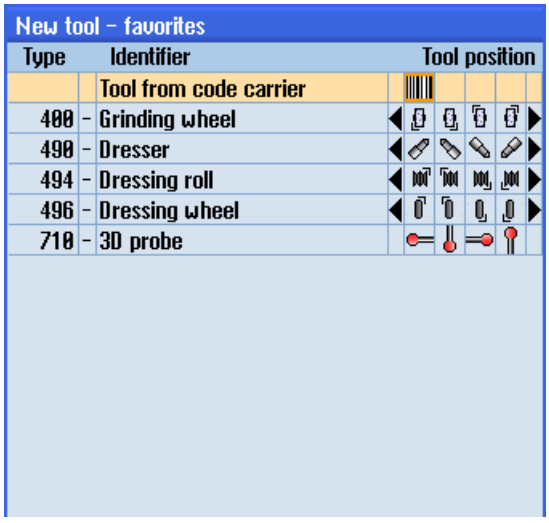


Figure 13-8 New tool from code carrier in the list of favorites

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 opens.

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.

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 then written to the code carrier.

According to the appropriate setting, after being read out on the code carrier, the unloaded tool on the code carrier is deleted from the NC memory.

Delete tool on code carrier

1. The tool list is opened.
2. Position the cursor on the tool on 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.

13.5.10 Managing a tool in a file

If, in the tool list settings, the "Permit tool in/out file" option is activated, then an additional entry is available in the list of favorites.

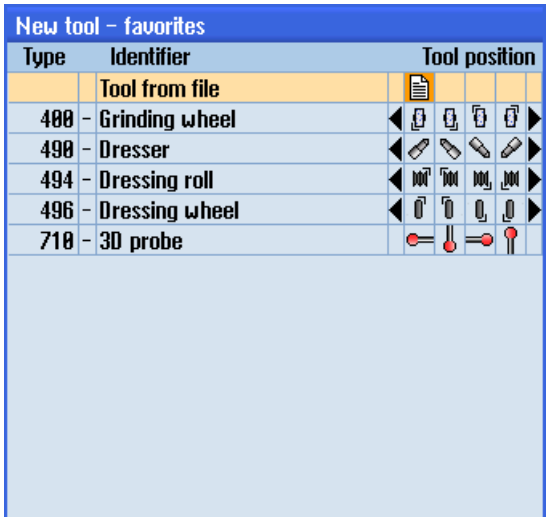


Figure 13-9 New tool from file in the list of favorites

Creating a new tool from file

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 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 opened.

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 the tool names

   The tool is unloaded and the data of the tool are written to the file.

According to the corresponding setting, after having been read out, the unloaded tool is deleted from the NC memory.

Deleting a tool in a file

1. The tool list is opened.
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 the tool names

   The tool is unloaded and the data of the tool are written to the file. The tool is then deleted from the NC memory.

13.6 Tool wear

13.6.1 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 control 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.

In addition, you can disable tools when you no longer wish to use them.

Note

Combination of monitoring types

You have the option to activate the monitoring of a tool by type or any combination of monitoring types.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Tool parameters

Column heading	Meaning
Location W L B 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. • Transfer location • Loader • Loading station • 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 (this 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 sister 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	Sister tool number (for replacement tool strategy).
D	Cutting edge number
Δ length X, Δ length Z - Cylindrical grinding	Wear for length X or wear for length Z
Δ length Y, Δ length Z - Surface grinding	Wear for length Y or wear for length Z
Δ cutting edge radius	Tool wear of the cutting edge radius (for grinding wheel - type 400, type 410; dresser - type 490, dressing roller - type 495, dressing wheel - type 497)
Δ Ø	Tool wear of the diameter (for 3D-probe - type 710; edge-probe - type 711; mono-probe - type 712; L-probe - type 713; calibration tool - type 725)
Δ outer Ø	Tool wear of the outer diameter (for star-type probe - type 714)
T C	Selection of tool monitoring - by tool life (T) - by count (C) - by wear (W) - wear, summed offset (S) The wear monitoring is configured via a machine data item. Please refer to the machine manufacturer's instructions.

Column heading	Meaning
Tool life Workpiece count Wear Wear, summed offset * *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 activated.

Further parameters

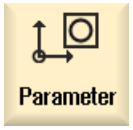
If you have set up unique cutting edge numbers, these are displayed in the first column.

Column heading	Meaning
D no.	Unique cutting edge number
SN	Cutting edge number
SC 	Summed offsets Display of the existing setup offsets

Icons in the wear list

Icon/ Marking		Meaning
Tool type		
Red cross		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 frame		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 load position.
Red cross		The magazine location is disabled.

Procedure



1. Select the "Parameter" operating area.

2. Press the "Tool wear" softkey.

13.6.2 Reactivating a tool

You can replace disabled tools or make them ready for use again.

Preconditions

In order to be able 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 cancelled.

Reactivating and positioning

When the "Reactivate with positioning" function is configured, the selected tool's magazine location will also be positioned at a 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 positions

If you have configured several loading points for a magazine, then the "Load Point Selection" window appears after pressing the "Load" softkey.

Select the required load position and confirm with the "OK" softkey.

Further information

Additional information on the configuration options is provided in the Tool Management Function Manual.

13.7 Tool data OEM

You have the option of configuring the list according to your requirements.

Depending on the machine configuration, grinding-specific parameters are displayed in the list with OEM tool data.

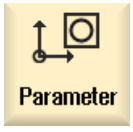
Grinding tool-specific parameters

Column heading	Meaning
Min. radius	Minimum radius Limit value for the radius of the grinding wheel to monitor the geometry.
Actual radius	Actual radius Displays the sum of the geometry value, wear value and, if set, the base dimension.
Min. width	Minimum wheel width Limit value for the width of the grinding wheel to monitor the geometry.
Actual width	Actual wheel width The width of the grinding wheel measured, for example, after the dressing.
Maximum speed	Specifies the maximum speed
Max. peripheral velocity	Specifies the maximum peripheral velocity
Wheel angle	Specifies the angle of inclined grinding wheel
Spindle number	Specifies the programmed spindle number (e.g. grinding wheel peripheral velocity) and spindle to be monitored (e.g. wheel radius and width).
Param. rad.calc.	Selects the parameter to calculate the radius • Length X • Length Y • Length Z • Radius
Chaining rule	This parameter defines which tool parameters of tool cutting edge 2 (D2) and tool cutting edge 1 (D1) have to be chained to one another. When the value of a chained parameter is modified, then the other cutting edge is automatically used when chaining the parameter.

Further information

- Additional information on grinding tools is provided in the Tools Function Manual
- You can find further information on configuring OEM tool data in the SINUMERIK Operate Commissioning Manual.

Procedure



1. Select the "Parameter" operating area.

2. Press the "OEM tool" softkey.
3. Position the cursor on a grinding tool.

13.8 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 W L B LP * 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. • Transfer location • Loader • Loading station • 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 (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. The icon identifies the position of the tool; this was selected when the tool was created.
	You have the option of changing the tool position or the tool type by pressing the <SELECT> key.
Tool name	The tool is identified by the name and number of the sister tool (ST). 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	Sister tool number.
D	Cutting edge number
G	Disabling the magazine location
Ü	Marking 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

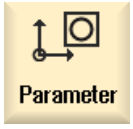
If you have set up unique cutting edge numbers, these are displayed in the first column.

Column heading	Meaning
D no.	Unique cutting edge number
SN	Cutting edge number

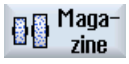
Magazine list icons

Icon/ marking		Meaning
Tool type		
Red cross		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 frame		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 load position.
Red cross		The magazine location is disabled.

Procedure



1. Select the "Parameter" operating area.



2. Press the "Magazine" softkey.

13.8.1 Positioning a 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.

13.8.2 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.

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 number in the "...magazine" field and the required magazine location number in "Location" field.

   Press the "OK" softkey.

   The tool is relocated to the specified magazine location.

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.

13.8.3 Deleting/unloading/loading/relocating all tools

You have the option of deleting all tools from the magazine list, unloading all tools from the magazine list, loading them to the magazine list, and relocating them in the magazine list at the same time.

Preconditions

The following requirements must be met so that the "Delete all," "Unload all," "Load all," or "Relocate all" softkey is displayed and is available:

- Magazine management is set up
- There is no tool in the buffer / in the spindle

Procedure

1. The magazine list opens.

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 asking whether you really want to delete, unload, load, or relocate all tools.

3. Press the "OK" softkey to go ahead and delete, unload, load, or relocate the tools.

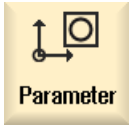
   The tools are deleted, unloaded, loaded, or relocated in the displayed order, i.e. as specified by the sorting and the set filter.

4. Press the "Cancel" softkey if you wish to cancel the unloading operation.

13.9 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.

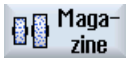
Procedure



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.

4. Press the "Acc. to type" softkey to display the tools arranged by tool type. Identical types are sorted according to their radius.

Press the "Acc. to name" softkey to display the tool names in alphabetical order.

The sister tool numbers are used to sort tools with the same names.

- OR -

Press the "Acc. to T number" softkey to display the tools sorted numerically.

The list is sorted according to the specified criteria.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

13.10 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 for equipping.

Filter criteria

- Only display the first cutting edge
- Only tools that are ready to use
- Only tools with active code
- Only tools that have reached the prewarning limit
- Only locked tools
- 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

Note

Multiple selection

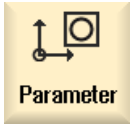
You have the option of selecting several criteria. You will receive an appropriate message if conflicting filter options are selected.

You can configure OR logic operations for the various filter criteria.

Machine manufacturer

Please observe the information provided by the machine manufacturer.

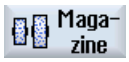
Procedure



1. Select the "Parameter" operating area.

2. Press the "Tool list", "Tool wear" or "Magazine" softkey.

...



3. Press the ">>" 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.

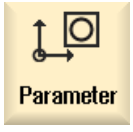
13.11 Specific search in the tool management lists

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 sister tool number.
    You have the option of only entering a part of the name as search term.
  - You enter the D number and activate if necessary, the check box "Active D number".
- **Magazine locations or magazines**
  If only one magazine is configured, then the search is made according to 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**
  The empty location search is realized using the tool size. The tool size is defined by the number of half locations required to the right, left, top and bottom. All four directions are of significance for a box magazine. For a chain magazine, a disk-type or a turret, only the half locations to the right and left are of significance. The maximum number of half locations which a tool can occupy is limited to 7.
  If the lists with the location type are used, then the empty location search is made using the location type and location size.
  You can enter the location type as numerical value or as text depending on the particular configuration.

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

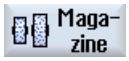
Procedure



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.

5. Press the "OK" softkey.

   The search is started.

6. Press the "Search" softkey again if the tool that was found is not the tool that was being searched for.

   The search term is kept and with "OK" you start the search for the next tool that corresponds to the entry.

7. Press the "Cancel" softkey to cancel the search.

13.12 Tool details

13.12.1 Displaying tool details

The following parameters of the selected tool can be displayed using softkeys in the "Tool Details" window.

- Tool data (Page 342)
- Grinding data (Page 343)
- Cutting edge data (Page 344)
- Monitoring data (Page 345)

Procedure

1. The tool list, the wear list, the OEM tool list or the magazine is open.

...

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.

The tool data is displayed in the list.

4. Press the "Grinding data" softkey if you want to display the grinding data.

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.

13.12.2 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 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 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
	I	Tool has been in use
Tool size 	Standard	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	

13.12.3 Grinding 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 sister tool number. You can enter the name as text or number.
ST	Sister tool number (for replacement tool strategy)
D quantity	Number of created cutting edges
D	Cutting edge number
Minimum wheel radius	Specifies the minimum wheel radius

Parameter	Meaning
Actual wheel radius	Specifies the actual wheel radius.
Minimum wheel width	Specifies the minimum wheel width
Actual wheel width	Specifies the actual wheel width
Maximum speed	Specifies the maximum speed
Maximum peripheral velocity	Specifies the maximum peripheral velocity.
Angle of inclined wheel	Specifies the angle of inclined wheel
Spindle number	Specifies the spindle number
Parameter for calculating the radius	Selected parameter for the radius calculation
Chaining rule	This parameter defines which tool parameters of tool cutting edge 2 (D2) and tool cutting edge 1 (D1) have to be chained to one another.

13.12.4 Cutting edge data

The "Tool Details" window provides the following data on the selected tool when the "Cutting edge data" softkey is active.

Parameter	Description		
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 replacement tool strategy)		
D quantity	Number of created cutting edges		
D	Cutting edge number		
Tool type	Tool icon with type number and current cutting edge position		
Cylindrical grinding			
	Length X or diameter	Length Z or diameter	
Geometry	Geometry data, length X	Geometry data, length Z	
Wear	Tool wear for length X	Tool wear for length Z	
Surface grinding			
	Length X	Length Z or diameter	Length Y or diameter
Geometry	Geometry data, length X	Geometry data, length Z	Geometry data, length Y
Wear	Tool wear for length X	Tool wear for length Z	Tool wear for length Y
	Radius		
Geometry	Cutting edge radius		
Wear	Wear of the cutting edge radius		
	Ø		
Geometry	Tool diameter		

Parameter	Description
Wear	Tool wear, diameter
Cutting edge OEM parameters 1 - 2	

13.12.5 Monitoring data

The "Tool Details" window provides the following data on the selected tool when the "Monitoring data" softkey is active.

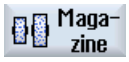
Parameter	Meaning
Magazine location	The magazine number is specified first, followed by the location number in the magazine. When there is only one magazine, only the location number is displayed.
Tool name	The tool is identified by the name and the sister tool number. You can enter the name as text or number.
ST	Sister tool number (for replacement 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
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	

13.13 Changing a tool type

Procedure

1. The tool list, the wear list, the OEM tool list or the magazine opens.

...



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 by pressing the "Grinders 400-499" or " Spec. tool 700-900" softkey.

5. Press the "OK" softkey.
   The new tool type is accepted and the corresponding icon is displayed in the "Type" column.

13.14 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.

13.14.1 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 lands on 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

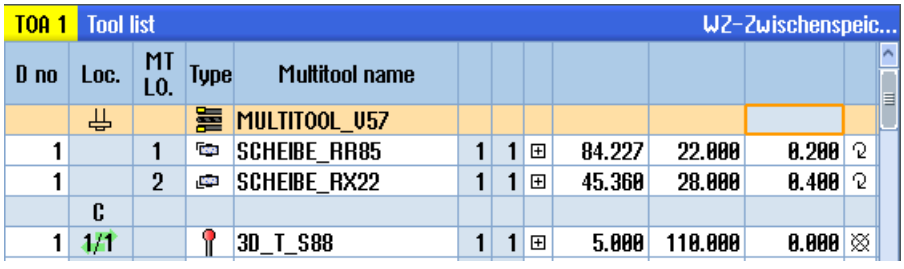


Figure 13-10 Tool list with multitool in the spindle

Procedure

1. Select the "Parameter" operating area.

2. Press the "Tool list" softkey.
   The "Tool list" window is opened.

13.14.2 Create multitool

The multitool can be selected in the list of special tool types.

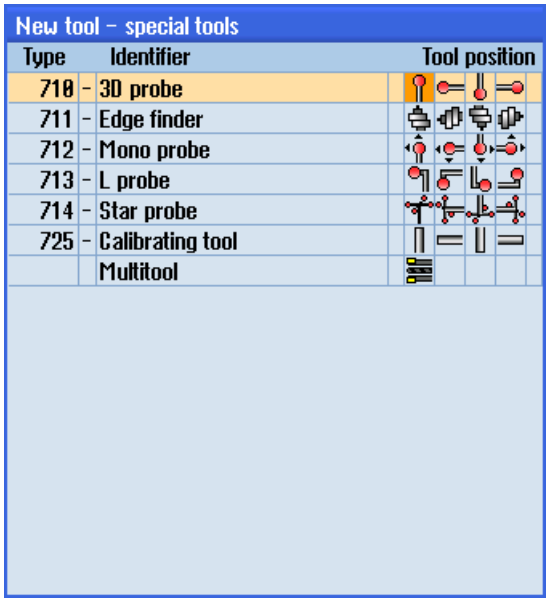


Figure 13-11 Selection list for special tools with multitool

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.

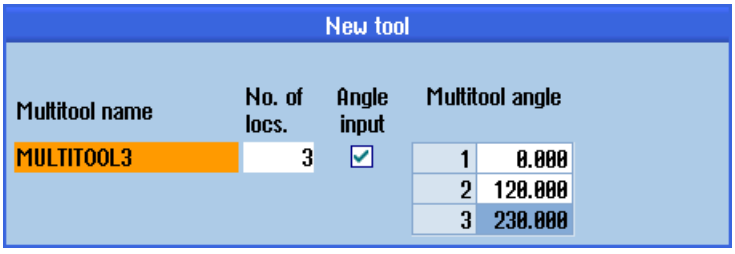
Spec.tool 700-900

4. Press the "Special tool 700-900" softkey.

OK

5. Select the multitool and press the "OK" softkey.
   The "New Tool" window opens.

6. Enter the multitool name and define the number of multitool locations.
   If you wish to define the distance of the tools based on the angle, select the "Angle input" check box, and for each multitool location, enter the distance to the reference location as an angle.



The multitool is created in the tool list.

Note

The tool creation sequence can be defined differently.

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

13.14.3 Equipping multitool with tools

Precondition

A multitool has been created in the tool list.

Procedure

Tool list

1. The tool list opens.

Equipping the multitool with tools

SELECT

2. Select the required multitool, position the cursor on an empty multitool location.

New tool

3. Press the "New tool" softkey.

4. Select the required tool from the relevant selection list, e.g. favorites.

Load multitool

2. Select the required multitool, position the cursor on an empty multitool location.

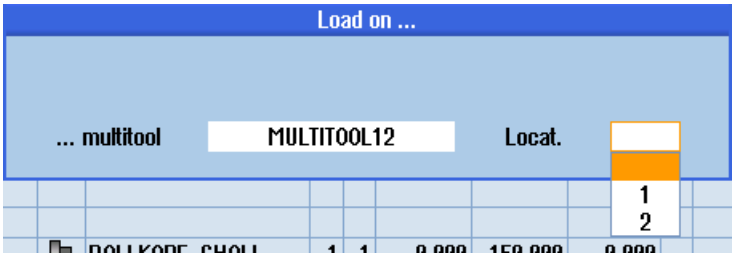
3. Press the "Load" softkey.
   The "Load with..." window opens.

4. Select the required tool.

Load 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 into which you wish to load the tool.

13.14.4 Removing a tool from multitool

If the multitool has been mechanically re-equipped with new tools, the old tools in the tool list must be removed from the multitool.

This is done by positioning the cursor on the line on which the tool is located that is to be removed. When it is unloaded, 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 would like to remove from the multitool and delete and press the "Delete tool" softkey.

13.14.5 Deleting multitool

Procedure

1. The tool list opens.
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.

13.14.6 Loading and unloading multitool

Procedure

1. The tool list is opened.

Loading a multitool into the magazine

2. Position the cursor on the multitool that you wish to load into the magazine.

3. Press the "Load" softkey.
   The "Load on" window opens.
   The "Locat." field contains the number of the first empty magazine location by default.

4. Press the "OK" softkey to load the multitool into the suggested 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 desired empty magazine location.

3. Press the "Load" softkey.
   The "Load with" window opens.

4. Select the desired 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.

13.14.7 Positioning 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 to bring the affected multitool location 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.

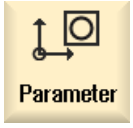
3. Press the "Position multitool" softkey.

13.14.8 Relocating a multitool

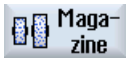
Multitools can be directly relocated to another magazine location within magazines, which means that you do not have to unload multitools with the tools in question from the magazine in order to relocate them to a different location.

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 relocate the multitool to the suggested magazine location.

   - OR -

   Enter the required magazine number in the "...magazine" field and the required magazine location number in "Location" field.

   Note:

   Please observe the information provided by the machine manufacturer.

Press the "OK" softkey.

The multitool with the tools is relocated to the specified magazine location.

13.14.9 Reactivating a multitool

Multitool and tools located on the multitool can be disabled independently of one another.

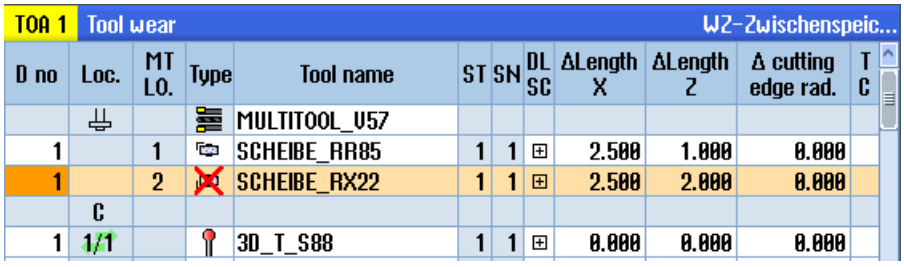
If a multitool is disabled, the tools of the multitool can no longer be loaded for use by means of a tool change.

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 tool life or the workpiece count has expired for one tool, only this tool is disabled.



D no	Loc.	MT LO.	Type	Tool name	ST	SN	DL SC	ΔLength X	ΔLength Z	Δ cutting edge rad.	T C
				MULTITOOL_V57							
1		1		SCHEIBE_RR85	1	1	⊞	2.500	1.000	0.000	
1		2		SCHEIBE_RX22	1	1	⊞	2.500	2.000	0.000	
	C										
1	1/1			3D_T_S88	1	1	⊞	0.000	0.000	0.000	

Reactivating

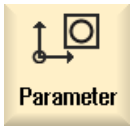
If a tool with an expired tool life or workpiece count that is mounted on a multitool is reactivated, the tool life or workpiece count is set to the target value for this tool and the tool and the multitool are enabled again.

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.

Preconditions

In order to be able 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 would like to activate 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 enabled again.

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 change 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.

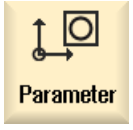
13.15 Settings for tool lists

In the "Settings" window you have the following options to set the view in the tool lists:

- Display only one magazine in "Magazine sort"
  - You can limit the display to one magazine. The magazine is displayed with the assigned buffer magazine locations and the tools not loaded.
  - You may set via a configuration if you want to jump to the next magazine by clicking softkey "Magazine selection" or if the dialog "Magazine selection" is switched over to any magazine.
- Display only spindle in buffer.
  In order to display only the spindle location during operation, the remaining locations of the buffer are hidden.
- Permits 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.
- Switch on adapter-transformed view
  - Geometry lengths and the application offsets are displayed in a transformed manner in the tool list.
  - In the tool wear list, the wear length and the sum offsets are displayed in a transformed manner.

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 corresponding check box for the desired setting.

Managing programs 14

14.1 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-FlashDrive Direct execution from a USB-FlashDrive is not recommended. There is no protection against contact problems, falling out, breakage through knocking or unintentional removal of the USB-FlashDrive 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 256 MB HMI user memory on CF card of the NCU" (not for SINUMERIK Operate on PCU50 or PC/PG).

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 passive 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 14-1 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), digits, and underscores.
- Type
  Directory: DIR or WPD
  Dressing program: DRS directory
  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 symbol.

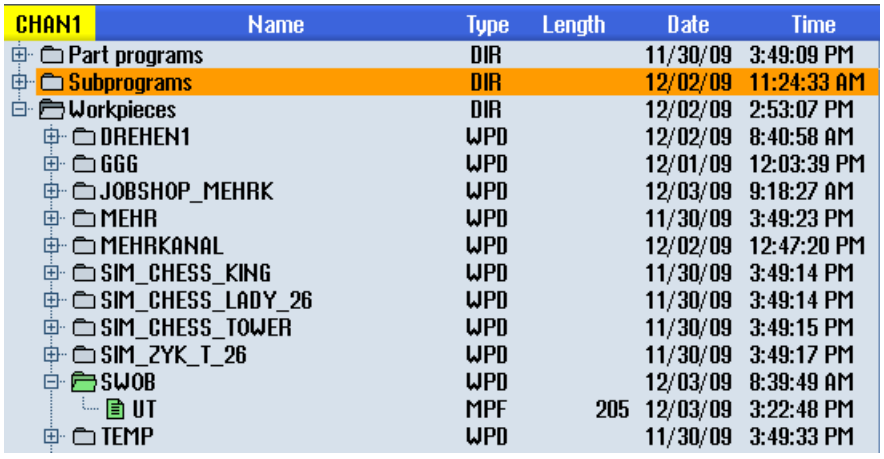


Figure 14-2 Active program shown in green

14.1.1 NC memory

The complete NC working memory is displayed along with all workpieces, main programs and subprograms as well as dressing programs.

You can create further subdirectories here.

Procedure

1. Select the "Program manager" operating area.

2. Press the "NC" softkey.

14.1.2 Local drive

Workpieces, main and subprograms as well as dressing programs that are saved in the user memory of the CF 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" softkey, you require option "Additional HMI user memory on CF card of the NCU" (not for SINUMERIK Operate on PCU50 or PC/PG).

Procedure

1. Select the "Program Manager" operating area.

2. Press the "Local drive" softkey.

14.1.3 Creating an NC directory on the local drive

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.

14.1.4 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 from the USB FlashDrive is not recommended, because machining can be undesirably interrupted, therefore resulting in workpiece damage.

Partitioned USB-FlashDrive (TCU)

If the USB-FlashDrive 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-FlashDrive is inserted in the front interface of the operator panel.

14.1.5 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.

14.2 Opening and closing the 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.

14.3 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.

14.4 Creating a directory / program / job list

14.4.1 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.

14.4.2 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.

14.4.3 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.

14.4.4 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.

14.4.5 Creating a new dressing program

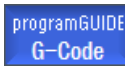
You can create dressing programs and then render G code blocks for them in a directory/ workpiece.

Procedure

1. Select the "Program manager" operating area.

2. Select a storage location and place the cursor on the "Dressing programs" folder.

3. Press the "New" softkey.



The "New Dressing Program" window opens.

The "DRS" file type is set as default.

4. Enter the desired program name and press the "OK" softkey.

14.4.6 Storing any new file

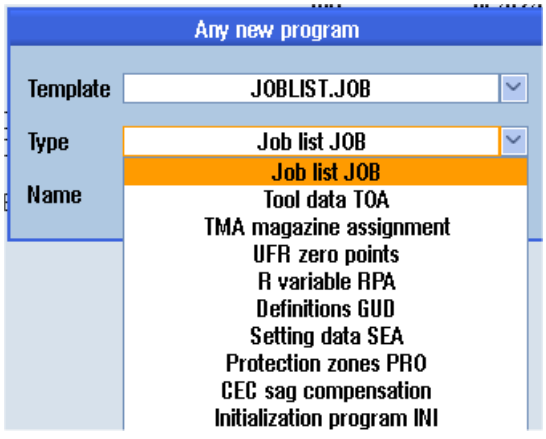
In each directory or subdirectory you can create a file in any format that you specify.

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.

14.4.7 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 (CF 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.

14.4.8 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.

6. To remove a program from the list, place the cursor on the appropriate line and press the "Delete" softkey.
   - OR -

   To delete all programs from the program list, press the "Delete all" softkey.

14.5 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.

14.6 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.

14.7 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.

14.8 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.

14.9 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.

Paste

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.

OK

6. Press the "OK" or "Overwrite all" softkey if you want to overwrite existing directories/programs.

Overwrite all

- OR -

Overwrite none

Press the "No overwriting" softkey if you do not want to overwrite already existing directories/programs.

- OR -

Skip

Press the "Skip" softkey if the copy operation is to be continued with the next file.

- OR -

OK

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.

14.10 Deleting a program/directory

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.

14.11 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.

14.12 Creating drives

14.12.1 Overview

Up to 21 connections to so-called logical drives (data carriers) can be configured. These drives can be accessed in the "Program manager" and "Startup" operating areas.

The following logical drives can be set up:

- USB interface
- Network drives
- CompactFlash card
- CompactFlash 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 CompactFlash card as a data carrier, you require the option "Additional HMI user memory on CF card of NCU" (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.

14.12.2 Setting up drives

The "Set Up Drives" window is available in the "Start-up" 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

Please 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 CompactFlash 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 CompactFlash card
	Manufacturer cycles	Access to the manufacturer cycle directory of the CompactFlash card
	Drive Windows	Access to a local PCU/PC directory.

Specifications for USB

Entry		Description
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		

Entry		Description
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		Description
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.
	CF_CARD	
	SYS_DRIVE	

Specifications for network drives

Entry		Description
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.
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		Description
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
Please refer to the machine manufacturer's specifications.

Entry		Description
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		

Entry		Description
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.

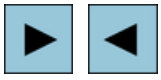
Specifications for the configured softkey

Entry		Description
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.
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_network_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 "Start-up" 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 the "OK" softkey.
   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.

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 "Start-up" 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 the "OK" softkey.

   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.

14.13 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.

14.14 EXTCALL

The EXTCALL command can be used to access files on a local drive, USB data carriers 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 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.

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-FlashDrive), if SD $SC 42700 is empty: e.g.
  EXTCALL "//ACTTCU/FRONT,1/TEST.SPF"
  - OR -
  Call of USB front connection (USB-FlashDrive), if SD42700 "//ACTTCU/FRONT,1" contains: EXTCALL "TEST.SPF"

- Call of network drive, if SD42700 is empty: e.g. EXTCALL "//computer name/enabled drive/TEST.SPF"
  - OR -
  Call of the network drive, if SD $SC42700 "//Computer name/enabled 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 CompactFlash 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, CompactFlash card and USB front connection

As abbreviation for the local drive, the CompactFlash card and the USB front connection you can use the abbreviation LOCAL_DRIVE:, CF_CARD: and USB: (e.g. EXTCALL "LOCAL_DRIVE:/spf.dir/TEST.SPF").

Alternatively, you can also use the abbreviations CF_Card and LOCAL_DRIVE.

Software options

To display the "Local drive" softkey, you require option "Additional HMI user memory on CF card of the NCU" (not for SINUMERIK Operate on PCU50 / PC).

NOTICE

Possible interruption when executing from USB FlashDrive

Direct execution from a USB-FlashDrive is not recommended.

There is no protection against contact problems, falling out, breakage through knocking or unintentional removal of the USB-FlashDrive 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.

Please observe the information provided by the machine manufacturer.

14.15 Execution from external storage (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 CompactFlash 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.

14.16 Backing up data

14.16.1 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 have the option of saving your archive in the binary and punched tape 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.

14.16.2 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 have the option of saving your archive in the binary and punched tape 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.

Precondition

The access rights depend on the relevant areas and range from protection level 7 (key switch position 0) to protection level 2 (password: Service).

Storage locations

- CompactFlash 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 CompactFlash Card in the user area you require the "Additional HMI user memory on CF card of NCU" option.

NOTICE
Possible data loss when using USB flash drives
USB-FlashDrives are not suitable as persistent memory media.

Procedure

1. Select the "Startup" 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.

14.16.3 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 option "Additional HMI user memory on CF card of the NCU" 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 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.

5. Press the "Cancel" softkey to cancel the read-in process.

14.16.4 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.

14.17 Setup data

14.17.1 Backing up 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.

14.17.2 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.

14.18 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.

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

14.19 V24

14.19.1 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.

Externally processing the punched tape format

If you wish to externally process an archive, then generate this in the punched tape format.

Procedure

1. Select the "Program manager" operating area, and press the "NC" or "Local. drive" softkey.

...

- OR -

Select the "Startup" operating area and press the "System data" softkey.

Reading-out archives

2. Select the directories or the files that you wish to send to V24.

3. Press the ">>" and "Archive" softkeys.

4. Press the "V24 send" softkey.

- OR -

Reading in an archive

Press the "V24 receive" softkey if you wish to read-in files via V24.

14.19.2 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	• Punched tape format (default setting) • 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
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

V24 setting	Meaning
Wait for XON for start receive V24	Only with protocol: Xon/Xoff
End of data transfer (hex)	Only for punched tape format Stop with end of data transfer character The default setting for the end of data transfer character is (HEX) 1A
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.

Alarm, fault, and system messages 15

15.1 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

Procedure

1. Select the "Diagnostics" operating area.

2. Press the "Messages" softkey.
   The "Messages" window appears.

15.2 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.

Alarm overview

You can display all upcoming alarms and acknowledge them.

The alarm overview contains the following information:

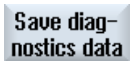
- Date and time
- Cancel criterion
  The delete criterion specifies the key or softkey that can be used to acknowledge the alarm.
- Alarm number
- Alarm text

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

Symbol	Meaning
	Dialog alarms of the HMI
	PLC alarm
	PLC alarm of the SQ type (alarm number from 800000)
	Safety alarms

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

15.3 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`.

15.4 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.

15.5 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".

15.6 PLC and NC variables

15.6.1 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 variable.
  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.

PLC variables	
Inputs	• Input bit (Ex), input byte (EBx), input word (EWx), input double word (EDx) • Input bit (Ix), input byte (IBx), input word (IWx), input double word (IDx)
Outputs	• Output bit (Ax), output byte (ABx), output word (AWx), output double word (ADx) • Output bit (Qx), output byte (QBx), output word (QWx), output double word (QDx)
Bit memory	Memory bit (Mx), memory byte (MBx), memory word (MWx), memory double word (MDx)
Times	Time (Tx)
Counters	• Counter (Cx) • Counter (Cx)
Data	• Data block (DBx): Data bit (DBXx), data byte (DBBx), data word (DBWx), data double word (DBDx) • Data block (VBx): Data bit (VBXx), data byte (VBBx), data word (VBWx), data double word (VBDx)

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: EB2, A1.2, DB2.DBW2, VB32000002
- 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 into an NC/PLC variable, the string is only displayed correctly if the variable is parameterized on the NC side as a field variable of type "A" (ASCII).

Example of a field variable

Variable	Format
DBx.DBBy[<number>]	A

Inserting variables

The start value for "Filtering/searching" 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 variables displayed channel-specifically, e.g. $R1:CHAN1 and $R1:CHAN2. The values of channel 1 and channel 2 are displayed, no matter in which channel you are.
- 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 system 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 the 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]

15.6.2 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.

15.7 Version

15.7.1 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

Information is provided in the "Nominal version" column as to whether the versions of the components deviate from the version supplied on the CompactFlash Card.

The version displayed in the "Actual version" column matches the version of the CF card.

The version displayed in the "Actual version" column does not match the version of the CF card.

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.

15.7.2 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 pre-assigned with <Machine name/no.>+<CF-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.

15.8 Logbook

15.8.1 Overview

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.

15.8.2 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.

15.8.3 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.

15.9 Remote diagnostics

15.9.1 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.

15.9.2 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.

15.9.3 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.

15.9.4 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".

Teaching in a program 16

16.1 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.

16.2 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.

16.3 Processing a program

16.3.1 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.

16.3.2 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.

16.3.3 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.

16.3.4 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.

16.4 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.

16.4.1 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
G641	Programmable corner rounding
G642	Axis-specific corner rounding

Parameter	Description
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

16.5 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.

Ctrl-Energy 17

17.1 Functions

The "Ctrl-Energy" function provides you with the following options to improve the energy utilization of your machine.

Ctrl-E Analysis: Measuring and evaluating the 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 multi-function device.

Depending on the configuration and connection of the SENTRON PAC, you have the possibility of either measuring the power of the whole machine or only a specific load.

Independent of this, the power is determined directly from the drives and displayed.

Ctrl-E Profiles: Control of energy saving states of the machine

To optimize the energy consumption, you have the option of defining energy saving profiles and saving them. For instance, your machine has a basic and a more sophisticated energy-saving mode – or under certain conditions, automatically switches itself off.

These defined energy states are saved as profiles. At the user interface, you have the possibility of activating these energy-saving profiles (e.g. the so-called tea break key).

Note

Ctrl-E Deactivating profiles

Disable Ctrl-E profiles before a series commissioning in order to prevent the NCU unintentionally shutting down.

Machine manufacturer

Please observe 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.

Further information

Additional information on configuring is provided in the Ctrl-Energy System Manual.

17.2 Ctrl-E analysis

17.2.1 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
- Measurement of the current energy consumption
- Comparison measurement for the energy consumption

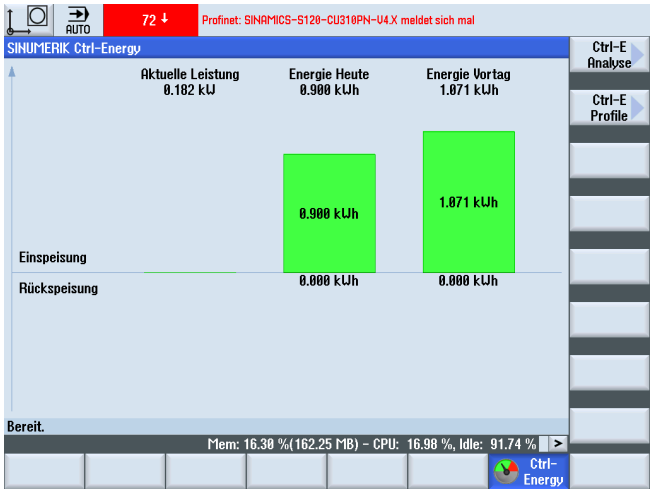


Figure 17-1 Ctrl-Energy entry screen with display of the current energy consumption

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 operating productively.
	A dark-green bar in the positive direction indicates that the machine is operating productively and consuming energy.
	A light-green bar in the negative direction indicates that the machine is feeding energy back into the power supply system.

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.

17.2.2 Displaying the energy analyses

You can obtain a detailed overview of the energy usage in the "Ctrl-E analyse" window.

You obtain the usage display for the following components:

- Sum of the axes
- Sum of the units, if auxiliary units 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. You obtain the summed usage values for all of the components.

3. Press the "Details", softkey to display the energy usage of individual drives and auxiliary units.

17.2.3 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.

Measurement of the energy consumption by part programs

The energy consumption of part programs can be measured. Single 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.

Save measurements

Save the measured consumption values so that you can subsequently compare the data.

Note

Up to 3 data sets are saved. The oldest data set is automatically overwritten if there are more than three 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

Please observe the information provided by the machine manufacturer.

Requirement

You have pressed the "Ctrl-E analyse" softkey and the "Ctrl-E analyse" window has been opened.

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" checkbox, enter the number of repetitions, select the required channel, and press the "OK" softkey.

   The trace is started.

3. Press the "Stop measurement" softkey.
   The measurement is terminated.

4. Press the "Save measurement" softkey to save the consumption values of the actual measurement.

The selection of the axis to be measured depends on the configuration.

17.2.4 Tracking measurements

You have the option of graphically displaying the actual and saved measurement curves.

Precondition

You have pressed the "Ctrl-E analyse" softkey and the "Ctrl-E analyse" window has been opened.

Procedure

1. Press the "Graphic" softkey.
   The actual measurement is displayed as a blue measurement curve in the "Ctrl-E analyse" window.

2. Press the "Saved measurements" softkey to display the measurements last saved.
   In addition, measurement curves with 3 different colors paths are displayed together with the measurement time.

3. Press the "Saved measurements" softkey again if you only want to see the actual measurement.

17.2.5 Tracking usage values

You have the option of displaying the actual and saved usage values in a detailed table.

Display	Meaning
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. • Manual (fixed value, z.B. base load, defined in the PLC) • Sentron PAC • Sum of the units (if defined in the PLC) • Sum of the axes • Total, machine

Display	Meaning
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"

Precondition

1. You have pressed the "Ctrl-E analyse" softkey and the "Ctrl-E analyse" window has been opened.
2. You have already saved measurements.

Procedure

Press the "Graphic" and "details" softkeys.

In the "Ctrl-E analysis" window: Details", measurement data and usage values of the last three saved measurements – and possibly an actual measurement – are displayed in a table.

17.2.6 Comparing usage values

You have the option of comparing usage values (power drawn and power fed back) of actual and saved measurements.

Precondition

1. You have pressed the "Ctrl-E analyse" softkey and the "Ctrl-E analyse" window has been opened.
2. You have already saved measurements.

Procedure

1. Press the "Graphic" 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 diagram.

3. Press the "Saved measurements" softkey to compare the last 3 measurements that were saved.

4. Press the "Saved measurements" softkey again if you only want to see the actual comparison.

17.2.7 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

Precondition

SENTRON PAC is connected.

Procedure

1. The "Ctrl-E Analysis" window is open.

2. 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.

3. Press the "Back" softkey to terminate the long-term measurement.

17.3 Ctrl-E profiles

17.3.1 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

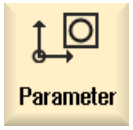
Energy-saving profile	Meaning
Simple energy-saving mode (machine standby)	Machine units that are not required are either throttled or shut down. When required, the machine is immediately ready to operate again.
Full energy-saving mode (NC standby)	Machine units that are not required are either throttled 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

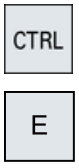


1. Select the "Parameter" operating area.

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.

Easy XML 18

18.1 Easy XML

The "Create user dialogs" function allows you to design customer- and application-specific HMI user interfaces with an XML-based script language.

This script language makes it possible to display machine-specific menus and dialog forms in the <CUSTOM> operating area on the HMI.

These scripts can also be executed from an NC program with the MMC(...) instruction.

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

Usage

The defined script instructions offer the following properties:

1. Display dialogs containing the following elements:
   - Softkeys
   - Variables
   - Texts and Help texts
   - Graphics and Help displays
2. Call dialogs by:
   - Pressing the (start) softkeys
3. Restructure dialogs dynamically:
   - Edit and delete softkeys
   - Define and design variable fields
   - Insert, exchange, and delete display texts (language-dependent or language-neutral)
   - Insert, exchange, and delete graphics
4. Initiate operations in response to the following actions:
   - Displaying dialogs
   - Inputting values (variables)
   - Selecting a softkey
   - Exiting dialogs
5. Data exchange between dialogs

6. Variables
   - Read (NC, PLC, drive and user variables)
   - Write (NC, PLC, drive and user variables)
   - Combine with mathematical, comparison or logic operators
7. Execute functions:
   - Subprograms
   - File functions
   - PI services
8. Apply protection levels according to user classes
9. Controlling dialog content with part program instructions

Calling user dialogs

If the "xmldial.xml" configuration file is stored in the /oem/sinumerik/hmi/appl directory, start the user dialogs by pressing the <CUSTOM> key.

The configured softkeys are displayed when the <CUSTOM> operating area is called. You open and operate the configured dialogs via the softkeys.

Note

After the initial copying of the file to the directory, a RESET of the control is required.

Additional information

Additional information on the configuration of user-specific dialogs is provided in the Easy XML Programming Manual.

18.2 SINUMERIK Integrate Run MyScreens

"Run MyScreens" allows you to design your own user interface for machine manufacturer- or user-specific functional expansions and implement a user-specific layout.

You also have the possibility to modify or replace configured Siemens or machine-manufacturer user interfaces.

With newly created user interfaces, you can process part programs, for example. The dialogs are created directly at the control.

Software option

To expand the number of dialogs, you require one of the following software options:

- SINUMERIK Integrate Run MyScreens
- SINUMERIK Integrate Run MyScreens + Run MyHMI
- SINUMERIK Integrate Run MyHMI / 3GL
- SINUMERIK Integrate Run MyHMI / WinCC

Machine manufacturer

Please observe the information provided by the machine manufacturer.

Usage

Defined script instructions permit the following functions:

1. Display dialogs containing the following elements:
   - Softkeys
   - Variables
   - Texts and Help texts
   - Graphics and Help displays
2. Call dialogs by:
   - Pressing the (start) softkeys
3. Restructure dialogs dynamically:
   - Edit and delete softkeys
   - Define and design variable fields
   - Insert, exchange, and delete display texts (language-dependent or language-neutral)
   - Insert, exchange, and delete graphics
4. Initiate operations in response to the following actions:
   - Displaying dialogs
   - Inputting values (variables)
   - Selecting a softkey
   - Exiting dialogs

5. Data exchange between dialogs
6. Variables
   - Read (NC, PLC and user variables)
   - Write (NC, PLC and user variables)
   - Combine with mathematical, comparison or logic operators
7. Execute functions:
   - Subprograms
   - File functions
   - PI services
8. Apply protection levels according to user classes

Additional information

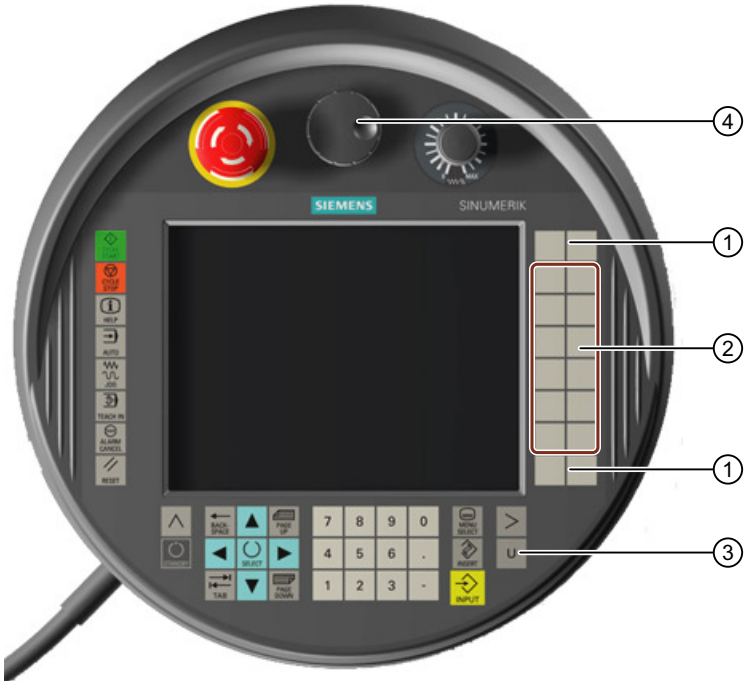
Additional information on the configuration of user-specific dialogs is provided in the SINUMERIK Integrate Run MyScreens Programming Manual.

Handheld terminals for multi-touch operation 19

19.1 HT 8

19.1.1 Overview

The mobile SINUMERIK HT 8 handheld terminal 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.



① Customer keys (user-defined)
② Traversing keys
③ User menu key
④ Handwheel (optional)

Operation

The 7.5" TFT color display provides touch operation.

Membrane keys are provided for traversing the axes, numeric input, control of the cursor, and for machine control panel functions (e.g. start and stop).

The HT 8 is equipped with an emergency stop button and two 3-stage acknowledgment buttons. You can also connect an external keyboard.

Customer keys

The four customer keys can be assigned arbitrarily and can be configured customer-specifically.

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

Integrated machine control panel

An MCP is integrated in the HT 8. It is comprised of keys (e.g. start and stop) as well as softkeys emulating keys.

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 8 has two acknowledgment buttons. You can then activate the acknowledgement function for operator actions which require approval (e.g. showing the traversing keys) with either hand.

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 with the HT 8 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.

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 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 user 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 8 is available with a hand wheel.

Further information

Additional information about connection and commissioning is provided in the Handheld Terminal HT 8 Equipment Manual.

19.1.2 Traversing keys

The traversing keys are not labeled. However, you can display a label for the keys in place of the vertical softkey bar.

Labeling of the traversing keys is displayed for up to six axes on the touch panel by default.

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

Showing and hiding

You can link the showing and hiding of the label to activation of the enabling button, for example. In this case, the traversing keys are displayed when you press the enabling button.

If you release the enabling button, the traversing keys are hidden again.

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

All existing vertical and horizontal softkeys are covered or hidden, i.e. other softkeys cannot be used.

19.1.3 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.

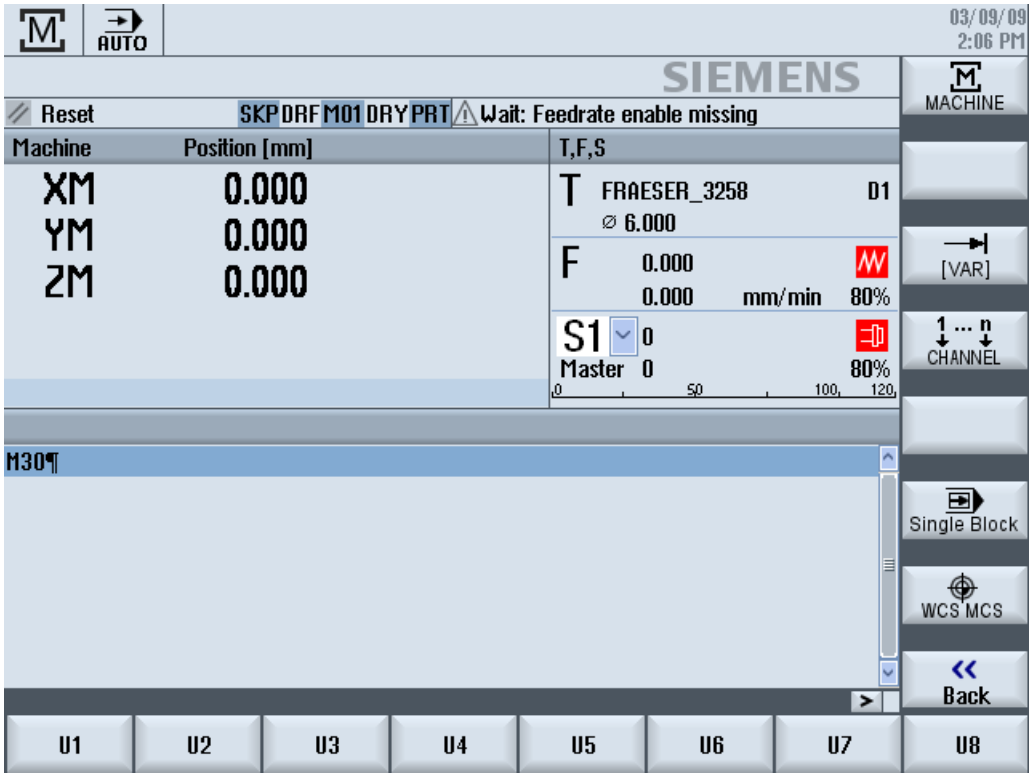
See Section "Controls on the machine control panel" for a description of the individual keys.

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. There are 8 additional softkeys available.

You use the "Back" softkey to hide the menu bar again

Softkeys on the machine control panel menu

Available softkeys:

"Machine" softkey	Select the "Machine" operating area
"[VAR]" softkey	Select the axis feedrate in the variable increment
"1… n CHANNEL" softkey	Changing the channel
"Single Block" softkey	Switch single block execution on/off
"WCS MCS" softkey	Switch between WCS and MCS
"Back" softkey	Close window

Note

The window will automatically disappear when changing regions areas with the <MENU SELECT> key.

19.1.4 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.

Procedure

1. Position the cursor on the desired entry field.
2. Click the input field.
   The virtual keyboard is displayed.
3. Enter your values via the virtual keyboard.
4. Press the <INPUT> key.

- OR -

Position the cursor on an another operator element.

The value is accepted and the virtual keyboard is closed.

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.

Special keys on the virtual keyboard

① "Tilde" key

- Changes the sign in a numerical entry field.
- A tilde character is inserted in a text box (e.g. program editor).

② "Eng" key

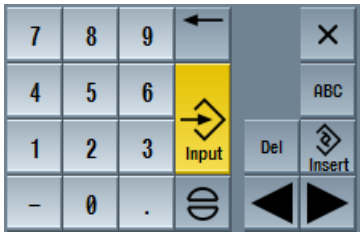
Toggles the keyboard assignment between the English keyboard assignment and the keyboard assignment for the current language setting.

③ Area for positioning the virtual keyboard.

④ "Num" key

Reduces the virtual keyboard to the number block.

Number block of the virtual keyboard



Press the "ABC" key to return to the full keyboard.

19.2 HT 10

19.2.1 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 local 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 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 user 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.

Further information

Additional information about connecting, commissioning and configuring user-specific keys is provided in the Handheld Terminal HT 10 Equipment Manual.

19.2.2 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.

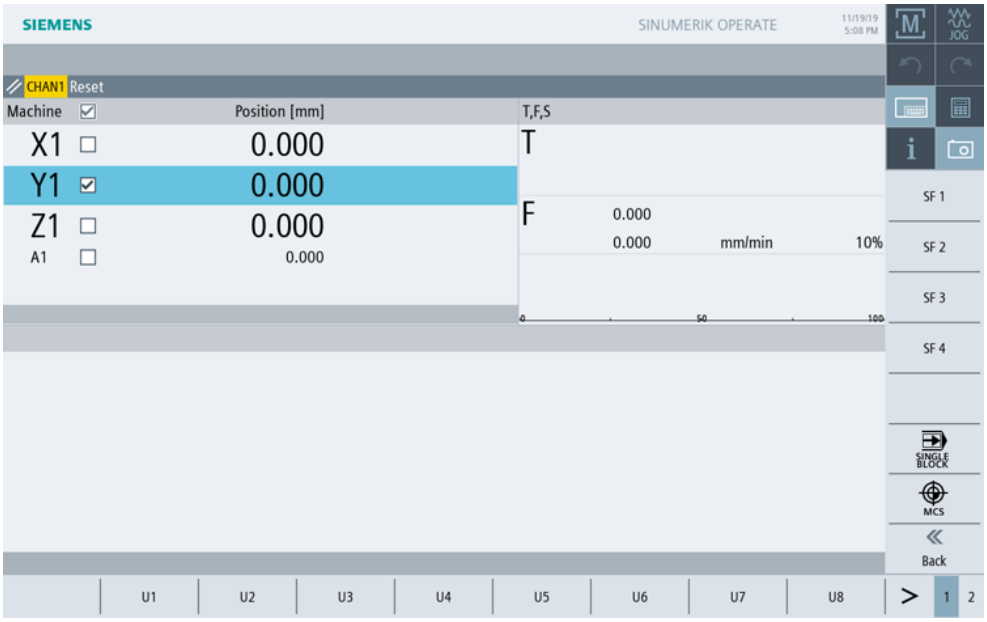
See Section "Controls on the machine control panel" for a description of the individual keys.

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.

Then traverse the axes through a fixed increment using the operator control elements on the touch screen.

19.2.3 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.

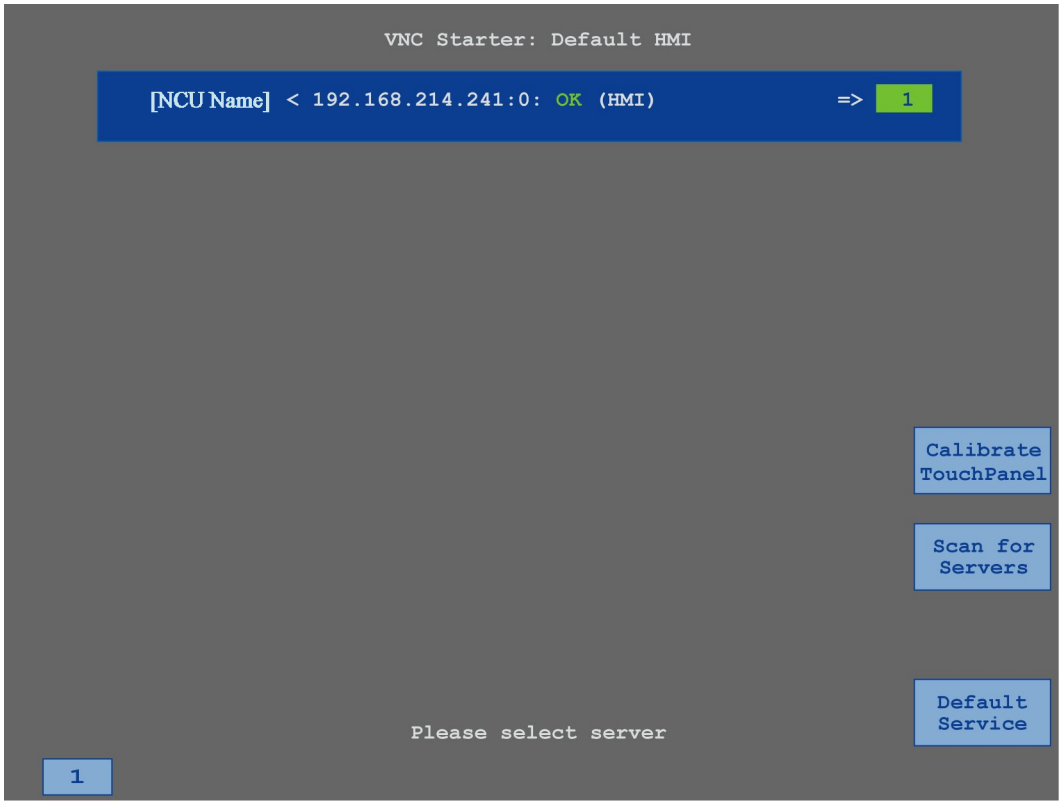
19.3 Calibrating the touch panel

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.

Appendix

20

20.1 Documentation overview SINUMERIK 840D sl

You will find extensive documentation on the functions of SINUMERIK 840D sl from version 4.8 SP4 at 840D sl documentation overview (https://support.industry.siemens.com/cs/ww/en/view/109766213).

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

The documentation is divided into the following categories:

- User: Operating
- User: Programming
- Manufacturer/Service: Configure
- Manufacturer/Service: Commissioning
- Manufacturer/Service: Functions
- Safety Integrated
- SINUMERIK Integrate / MindApp
- Information and training

Index

"

"SINUMERIK Operate Gen. 2"
    Multitouch panel, 63
"SINUMERIK Operate Generation 2" user interface, 63
    Function key block, 69
    Touch operator controls, 69
    Virtual keyboard, 70

A

ABC keyboard, 79
Acknowledgment button, 460, 467
Actual-value display, 37
Adapter-transformed view, 355
Advanced Surface, 200
Alarm log
    display, 417
    Sorting, 418
Alarms
    Backing up log data, 414
    cancel, 415
    Displaying, 414
    Sorting, 418
Any file, 370
Archive
    Generate in the program manager, 396
    generating in the system data, 397
    Punched tape format, 396
    read in archive from system data, 400
    reading-in in the Program Manager, 399
Auxiliary functions
    H functions, 201
    M functions, 201
Axes
    Defined increment, 140
    Direct positioning, 142
    Referencing, 88
    Repositioning, 155
    Traversing, 140
    Variable increment, 141

B

B axis
    Calibrating the swivel axis, 288
Backing up
    Data - in the Program Manager, 396
    Data - via the system data, 397
    Parameter, 405
    Setup data, 401
Base offset, 100
Basic blocks, 151
Binary format, 396
Block
    Search, 156
    Searching - interruption point, 159
Block search
    Mode, 160
    Program interruption, 159
    Search target parameters, 159, 160
    Search target specification, 158
    Using, 156

C

Calibrating the swivel axis
    B axis, 288
Camera
    Streaming, 58, 59
Channel switchover, 93
Chuck dimensions, 122
Cleaning mode, 57
Coarse and fine offsets, 100
Code carrier connection, 321
Collision avoidance, 291
    Displaying the machine model (kinematic chains), 293
    Machine operating area, 294
    Settings, 294
Configured stop, 163
Context-sensitive online help, 60
Contour call - CYCLE62
    Function, 248
    Parameter, 249
Coordinate system
    Switching over, 95
Counter-spindle, 123
Create
    User dialog, 455
Creating
    Dressing program, 369
    Multitool, 348
    Program block, 172

Ctrl-Energy
Comparing usage values, 450
Displaying measured curves, 449
Displaying usage values, 449
Energy analysis, 446, 447
Energy-saving profiles, 452
Functions, 445
Measuring the energy consumption, 448
Saved measuring curves, 449, 450
Cutting edge numbers, 313
CYCLE4071
External programming, 256
CYCLE4072
External programming, 258
CYCLE4073
External programming, 262
CYCLE4074
External programming, 263
CYCLE4075
External programming, 266
CYCLE4077
External programming, 269
CYCLE4078
External programming, 273
CYCLE4079
External programming, 275
CYCLE435 - Set dresser coordinate system
External programming, 250
CYCLE495 - form-truing
External programming, 252
CYCLE62- contour call
Function, 248
Parameter, 249
Cylinder error compensation, 125
Cylindrical grinding
Measuring a tool, 108

D

Data block (SB2), 148
Delete
Multitool, 351
Deleting
Directory, 382
Program, 382
Dictionary
Importing, 52
Directory
Copying, 380
Creating, 367
Deleting, 382
Highlight, 378
Inserting, 380
Naming convention, 367
Properties, 383
Selecting, 378
Display Manager
Operator controls, 82
Dressing program
Creating, 369
Program directory, 359
DRF (handwheel offset), 162
Drives
Logical drive, 384
Setting up, 384
DRY (dry run feed), 162
Duplo number, (See sister tool number)
DXF file
Changing the section, 181
Cleaning, 179
close, 179
Delete Element, 185
Delete range, 184
Increasing / decreasing the section, 180
Machining plane, 183
Open, 179
Rotating the drawing, 181
Saving, 185
Select contour and accept, 186
Select the machining range, 184
Snap radius, 183
Specifying a reference point, 186
Zero point, 186
DXF-Reader, 178

E

Editor
Calling, 167
Settings, 174
EES (execution from external storage), 387
Energy analysis
Details, 447
Displaying, 447
Long time measurement, 451
Energy consumption
Displaying, 446
Measuring, 448
Energy-saving profiles, 452
Equipping tools
Multitool, 349
EXTCALL call, 393

F

Feed data
  Actual value window, 39
Finger gestures, 65
Form-truing - CYCLE495
  External programming, 252
FTP drive, 362
Function
  REF POINT, 91
  REPOS, 91
  Single block, 92
  TEACH IN, 92
Function key block
  "SINUMERIK Operate Generation 2" user interface, 69

G

G Code program
  Creating, 369
G functions
  Display all G groups, 200
  Displaying selected G groups, 198
  Mold making, 200
GCC (G code converter), 162
Global R parameters, 190
Global user variables, 192
Gloves, 64

H

Handheld terminal 10, 466
Handheld terminal 8, 459
Handwheel
  Assigning, 128
Highlight
  Directory, 378
  Program, 378
HT 10
  Acknowledgment button, 467
  Overview, 466
  User menu, 468
  Virtual keyboard, 470
HT 8
  Acknowledgment button, 460
  Overview, 459
  Touch Panel, 472
  Traversing keys, 461
  User menu, 462
  Virtual keyboard, 464

I

IME
  Chinese characters, 49
  Korean characters, 53
Interruption point
  Approach, 159

J

Job list, 371

K

Key combinations - Simulation
  Changing the section, 227
  Enlarging/reducing a graphic, 225
  Feedrate, 219
  Moving the graphic, 226
  Override, 219
  Rotating the view, 226
  Single block mode, 220

L

Layer selection, 179
Loading
  Multitool, 351
Local drive
  Creating an NC directory, 360
Logbook
  Delete entries, 428
  Displaying, 427
  Edit the address data, 427
  Entry search, 429
  Making an entry, 428
  Output, 426
  Overview, 427
Long time measurement
  Energy analysis, 451

M

Machine control panel
  in the sidescreen, 78
  Operator controls, 30
Machine model, 291

Machine-specific information, 426
Machining times
Delete, 177
Display in block display, 40, 150
Representation, 230
Magazine
Deleting tools, 336
Loading tools, 336
Positioning, 334
Relocating tools, 336
Unloading tools, 336
Magazine list, 333
Magazine management, 307
Main spindle, 123
Manual mode, 135
Settings, 143
Traversing axes, 140
MDA
Deleting a program, 133
MDI
Executing a program, 132
Loading a program, 130
Saving a program, 131
Measurement
Tool, 317
Measuring
Workpiece zero, 119
Measuring a tool
Cylindrical grinding, 108
Surface grinding, 112
Messages
Displaying, 413
Sorting, 418
Mode groups, 93
MRD (Measuring Result Display), 163
Multi-channel view, 297
"Machine" operating area, 298
OP015, OP019, 301
Settings, 303
Multitool, 347
Creating, 348
Delete, 351
Equipping tools, 349
Loading, 351
Parameters in the tool list, 347
Positioning, 352
Reactivating, 353
Relocating, 352
Removing tool, 350
Unloading, 352
Multitouch panel
SINUMERIK Operate Gen. 2, 63
Widescreen format, 72

N

Navigation bar
Sidescreen, 72
NC/PLC variables
Changing, 422
displaying, 420
New contour
Function - Milling, 239
Parameter - Milling, 240

O

Online help
Context-sensitive, 60
Operating areas, 19
Changing, 42
Selecting, 42
Operating mode
AUTO, 92
Changing, 42
JOG, 91, 135
MDI, 92
REPOS, 91
Operator controls
Display Manager, 82
Machine control panel, 30
Operator panel fronts, 20
Keys, 22

P

Pages, 72
Panel
cleaning, 57
Parameter
Backing up, 405
Calculating, 44
Changing, 44
Entering, 43
Positioning
Multitool, 352
Preview
Program, 377
Program
Closing, 363
Copying, 380

Correcting, 154
Creating with cycle support, 235
Deleting, 382
Editing, 167
Executing, 365
Highlight, 378
Inserting, 380
Naming convention, 367
Open, 363
Preview, 377
Properties, 383
Renumbering blocks, 172
Replacing text, 169
Running-in, 148
Searching for a program position, 167
Selecting, 147, 378
Teach-in, 435
Program block
Copying and inserting, 170
Current, 40, 150
Delete, 170
insert, 170
Numbering, 171
Search, 167
Selecting, 170
Program blocks, 172
Program control
activate, 163
Modes of operation, 162
Program correction, 154
Program level, 152
Program list, 373
Program Manager, 357
Displaying HTML documents, 391
Displaying PDF documents, 391
Searching for directories and files, 375
Program runtime, 206
Programmed stop 1, 162
Programmed stop 2, 162
Programs
Managing, 357
Properties
Directory, 383
Program, 383
Protection levels
Softkeys, 56
PRT (no axis motion), 162

Q

Quantity, 327

R

R parameters, 191
Backing up, 405
Reactivating
Multitool, 353
Tool, 329
Reference, 88
Relocating
Multitool, 352
Tool, 335
Remote access
Allowing, 431
Setting, 430
Remote diagnostics, 430
Exit, 433
Requesting, 432
Removing tools
Multitool, 350
Repositioning, 155
RG0 (reduced rapid traverse), 162
Run MyScreens, 457

S

Save
Parameter, 405
Setup data, 401
SB (single blocks), 162
SB1, 148
SB2, 148
SB3, 148
Screen layout, 68
Screenshots
Copying, 419
Creating, 419
Open, 419
Search
in the Program Manager, 375
Search mode, 160
Searching
Logbook entry, 429
Selecting
Directory, 378
Program, 378
Set dresser coordinate system - CYCLE435
External programming, 250
Setting actual values, (See setting zero offsets)
Settings
Collision avoidance, 294

Editor, 174
For automatic operation, 208
For manual operation, 143
Multi-channel view, 303
Teach-in, 444
Tool lists, 355
Setup data
Backing up, 401
reading in, 402
Shortcuts
Operator panel fronts, 22
Sidescreen
ABC keyboard, 78
MCP, 78
Navigation bar, 72
Operator controls, 72
Overview, 72
Pages, 78
Requirements, 72
Show, 74
Standard widgets, 74
Widgets, 72
Simulation
3D view, 222
Before machining, 215
Canceling, 215
Changing the feedrate, 219
Changing the graphic section, 227
Dressing view, 221
Face view (cylindrical grinding), 222
Further side views (surface grinding), 222
Machine space, 223
Moving the graphic, 226
Non-modal, 220
Overview, 211
Program control, 219
Rotating the graphic, 226
Side view, 221
Starting, 215
stopping, 215
Tool paths, 224
Views, 221
Simultaneous recording
3D view, 222
Before machining, 217
Changing the graphic section, 227
Dressing view, 221
During machining, 218
Face view (cylindrical grinding), 222
Further side views (surface grinding), 222
Machine space, 223
Moving the graphic, 226
Overview, 211
Rotating the graphic, 226
Side view, 221
Starting - before machining, 217
Tool paths, 224
Views, 221
Single block
Coarse (SB1), 148
Fine (SB3), 148
SINUMERIK Operate Gen. 2
Screen layout, 68
Sister tool number, 312
Skip blocks, 163
SKP (skip blocks), 162
Special characters, 21
Spindle chuck data
Parameter, 124
Store chuck dimensions, 122
Spindle data
Actual value window, 40
Spindle speed limitation, 122
Standard widget
Actual values, 74
Alarms, 75
Axle load, 76
Camera, 78
Service life, 77
Tool, 76
Variables, 76
Zero point, 75
Startup, 87
Status display, 35
Supplement
User interface, 457
Surface grinding
Measuring a tool, 112
Switching over
Coordinate system, 95
Synchronized actions
Displaying status, 204
System data
Displaying HTML documents, 391
Displaying PDF documents, 391

T

Tailstock, 124
Teach-in, 435
Changing blocks, 438
Circle intermediate position CIP, 441
Continuous-path mode, 443
Deleting blocks, 439

General sequence, 435
Inserting a position, 441
Inserting blocks, 438
Motion type, 442
Parameter, 442
Rapid traverse G0, 441
Selecting, 437
Selecting a block, 439
Settings, 444
Traversing block G1, 441
Templates
Creating, 374
Storage locations, 374
Tool
Change type, 346
Delete, 318
Details, 342
Dimensioning, 310
Loading, 319
Measurement, 317
Reactivating, 329
Relocating, 335
Unloading, 319
Tool data
Actual value window, 39
Backing up, 401
reading in, 402
Tool life, 327
Tool list, 312
Tool lists
Settings, 355
Tool management, 305
List filtering, 338
Sorting lists, 337
Tool parameters, 310
Tool types, 308
Tool wear, 327
Tool wear list
Open, 326
Touch operator controls
Cancel alarms, 70
Changing the channel, 70
Touch Panel
calibration, 472
Transformed view, 355

U

Unique cutting edge numbers
Unique, 313
Unit of measurement
switching, 95
Unloading
Multitool, 352
USB drive, 361
User agreement, 89
User dialog
Creation, 455
User interface
Supplementing, 457
User variables, 189
Activating, 197
Backing up, 405
Channel GUD, 194
Defining, 197
Global GUD, 192, 197
Global R parameters, 190
Local LUD, 195
Program PUD, 196
R parameters, 191
Searching, 196

V

Variable screen forms, 424
Virtual keyboard
"SINUMERIK Operate Generation 2" user interface, 70
HT 10, 470
HT 8, 464
Virtual keys
ABC keyboard, 72
MCP keys, 72

W

Wear, 327
Wear, summed offset, 327
Widgets, 72
Work offsets
Active WO, 100
delete, 106
Delete, 106
Grinding frames, 103
Overview, 101
Seat offsets, 103
Working area limitation, 121
Workpiece
creating, 368
Starting machining, 145
Stopping machining, 145, 146
Workpiece counter, 206

Workpiece zero
Automatic measurement, 119
Manual measurement, 119

Z

Zero offsets
Displaying details, 104
Overview, 99
Settable ZO, 103
Setting, 97
Zero point
DXF file, 186
Zero point settings
Backing up, 401
reading in, 402