SIEMENS

SINUMERIK

SINUMERIK 840D sl Milling

Operating Manual

Valid for:
SINUMERIK 840D sl / 840DE sl
Software version
CNC system software for 840D sl/ 840DE sl V4.94
SINUMERIK Operate for PCU/PC V4.94

10/2020
6FC5398-7CP40-6BA3

Preface
Fundamental safety instructions 1
Introduction 2
Multitouch operation with SINUMERIK Operate 3
Setting up the machine 4
Execution in manual mode 5
Machining the workpiece 6
Simulating machining 7
Generating a G code program 8
Creating a ShopMill program 9
Programming technological functions (cycles) 10
Multi-channel view 11
Collision avoidance 12
Tool management 13
Managing programs 14
Alarm, error, and system messages 15
Working with Manual Machine 16
Teaching in a program 17
Handheld terminals for multi-touch operation 18
Ctrl-Energy 19
Appendix 20

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: 6FC5398-7CP40-6BA3
Ⓟ 10/2020 Subject to change

Copyright © Siemens AG 2008 - 2020.
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

This documentation is intended for users of milling 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.

Terms

The meanings of some basic terms used in this documentation are given below.

Program

A program is a sequence of instructions to the CNC which combine to produce a specific workpiece on the machine.

Contour

The term contour refers generally to the outline of a workpiece. More specifically, it refers to the section of the program that defines the outline of a workpiece comprising individual elements.

Cycle

A cycle, e.g. milling a rectangular pocket, is a subprogram defined in SINUMERIK Operate for executing a frequently repeated machining operation.

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/cs/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 **21**

1.1 General safety instructions 21

1.2 Warranty and liability for application examples 22

1.3 Security information 23

2 Introduction **25**

2.1 Product overview 25

2.2 Operator panel fronts 26
2.2.1 Overview 26
2.2.2 Keys of the operator panel 28

2.3 Machine control panels 36
2.3.1 Overview 36
2.3.2 Controls on the machine control panel 36

2.4 User interface 40
2.4.1 Screen layout 40
2.4.2 Status display 41
2.4.3 Actual value window 43
2.4.4 T,F,S window 45
2.4.5 Current block display 47
2.4.6 Operation via softkeys and buttons 48
2.4.7 Entering or selecting parameters 50
2.4.8 Pocket calculator 51
2.4.9 Pocket calculator functions 52
2.4.10 Context menu 54
2.4.11 Changing the user interface language 55
2.4.12 Entering Chinese characters 56
2.4.12.1 Function - input editor 56
2.4.12.2 Entering Asian characters 57
2.4.12.3 Editing the dictionary 59
2.4.13 Entering Korean characters 60
2.4.14 Protection levels 63
2.4.15 Work station safety 65
2.4.16 Cleaning mode 65
2.4.17 Display live image from a camera 65
2.4.18 Online help in SINUMERIK Operate 67

3 Multitouch operation with SINUMERIK Operate **71**

3.1 Multitouch panels 71

3.2 Touch-sensitive user interface 72

3.3 Finger gestures 73

3.4 Multitouch user interface ... 76
3.4.1 Screen layout ... 76
3.4.2 Function key block ... 77
3.4.3 Further operator touch controls ... 77
3.4.4 Virtual keyboard ... 78
3.4.5 Special "tilde" character ... 79

3.5 Expansion with side screen ... 80
3.5.1 Overview ... 80
3.5.2 Sidescreen with standard windows ... 80
3.5.3 Standard widgets ... 82
3.5.4 "Actual value" widget ... 82
3.5.5 "Zero point" widget ... 83
3.5.6 "Alarms" widget ... 83
3.5.7 "NC/PLC variables" widget ... 83
3.5.8 "Axle load" widget ... 84
3.5.9 "Tool" widget ... 84
3.5.10 "Service life" widget ... 85
3.5.11 "Program runtime" widget ... 85
3.5.12 Widget "Camera 1" and "Camera 2" ... 85
3.5.13 Sidescreen with pages for the ABC keyboard and/or machine control panel ... 86
3.5.14 Example 1: ABC keyboard in the sidescreen ... 87
3.5.15 Example 2: Machine control panel in the sidescreen ... 88

3.6 SINUMERIK Operate Display Manager ... 89
3.6.1 Overview ... 89
3.6.2 Screen layout ... 90
3.6.3 Operator controls ... 90

4 Setting up the machine ... 95

4.1 Switching on and switching off ... 95

4.2 Approaching a reference point ... 96
4.2.1 Referencing axes ... 96
4.2.2 User agreement ... 97

4.3 Operating modes ... 99
4.3.1 General ... 99
4.3.2 Modes groups and channels ... 101
4.3.3 Channel switchover ... 101

4.4 Settings for the machine ... 103
4.4.1 Switching over the coordinate system (MCS/WCS) ... 103
4.4.2 Switching the unit of measurement ... 103
4.4.3 Setting the zero offset ... 105

4.5 Measure tool ... 107
4.5.1 Overview ... 107
4.5.2 Manually measuring drilling and milling tools ... 107
4.5.3 Measuring drilling and milling tools with the workpiece reference point ... 108
4.5.4 Measuring drilling and milling tools with fixed reference point ... 109
4.5.5 Measuring radius or diameter ... 110
4.5.6 Fixed point calibration ... 111
4.5.7 Measuring the drilling and milling tool length with electrical tool probe ... 111
4.5.8 Calibrating the electrical tool probe ... 114

4.5.9 Manually measuring a turning tool (for milling/turning machine) ... 115
4.5.10 Manually measuring a turning tool using a tool probe (for milling/turning machine) ... 116
4.5.11 Logging tool measurement results ... 118

4.6 Measuring the workpiece zero ... 120
4.6.1 Overview ... 120
4.6.2 Sequence of operations ... 124
4.6.3 Examples with manual swiveling (swiveling in JOG mode) ... 125
4.6.4 Setting the edge ... 126
4.6.5 Edge measurement ... 127
4.6.6 Measuring a corner ... 132
4.6.7 Measuring a pocket and hole ... 135
4.6.8 Measuring a spigot ... 138
4.6.9 Aligning the plane ... 143
4.6.10 Defining the measurement function selection ... 145
4.6.11 Logging measurement results for the workpiece zero ... 146
4.6.12 Calibrating the electronic workpiece probe ... 147
4.6.12.1 Calibration of length and radius or diameter ... 147
4.6.12.2 Calibrate on sphere ... 149

4.7 Settings for the measurement result log ... 151

4.8 Zero offsets ... 153
4.8.1 Display active zero offset ... 154
4.8.2 Displaying the zero offset "overview" ... 155
4.8.3 Displaying and editing base zero offset ... 156
4.8.4 Displaying and editing settable zero offset ... 157
4.8.5 Displaying and editing details of the zero offsets ... 158
4.8.6 Deleting a zero offset ... 159
4.8.7 Measuring the workpiece zero ... 160

4.9 Monitoring axis and spindle data ... 162
4.9.1 Specify working area limitations ... 162
4.9.2 Editing spindle data ... 162

4.10 Displaying setting data lists ... 164

4.11 Handwheel assignment ... 165

4.12 MDA ... 167
4.12.1 Loading an MDA program from the Program Manager ... 167
4.12.2 Saving an MDA program ... 168
4.12.3 Editing/executing a MDI program ... 169
4.12.4 Deleting an MDA program ... 170

5 Execution in manual mode ... 171

5.1 General ... 171

5.2 Selecting a tool and spindle ... 172
5.2.1 T, S, M windows ... 172
5.2.2 Selecting a tool ... 174
5.2.3 Starting and stopping a spindle manually ... 174
5.2.4 Position spindle ... 175

5.3 Traversing axes ... 177
5.3.1 Traverse axes by a defined increment ... 177
5.3.2 Traversing axes by a variable increment ... 178

5.4 Positioning axes ... 179
5.5 Swiveling ... 180
5.6 Manual retraction ... 185
5.7 Simple face milling of the workpiece ... 187
5.8 Simple workpiece machining operations with milling/turning machines ... 190
5.8.1 Simple workpiece face milling (milling/turning machine) ... 190
5.8.2 Simple stock removal of workpiece (for milling/turning machine) ... 192
5.9 Default settings for manual mode ... 196
6 Machining the workpiece ... 197
6.1 Starting and stopping machining ... 197
6.2 Selecting a program ... 199
6.3 Testing a program ... 200
6.4 Displaying the current program block ... 201
6.4.1 Current block display ... 201
6.4.2 Displaying a basic block ... 202
6.4.3 Display program level ... 203
6.5 Correcting a program ... 205
6.6 Repositioning axes ... 206
6.7 Starting machining at a specific point ... 207
6.7.1 Use block search ... 207
6.7.2 Continuing program from search target ... 209
6.7.3 Simple search target definition ... 210
6.7.4 Defining an interruption point as search target ... 210
6.7.5 Entering the search target via search pointer ... 211
6.7.6 Parameters for block search in the search pointer ... 212
6.7.7 Block search mode ... 212
6.7.8 Block search for position pattern ... 214
6.8 Controlling the program run ... 216
6.8.1 Program control ... 216
6.8.2 Skip blocks ... 217
6.9 Overstore ... 219
6.10 Editing a program ... 221
6.10.1 Searching in programs ... 221
6.10.2 Replacing program text ... 223
6.10.3 Copying/pasting/deleting a program block ... 224
6.10.4 Renumbering a program ... 226
6.10.5 Creating a program block ... 226
6.10.6 Opening additional programs ... 228
6.10.7 Editor settings ... 229
6.11 Working with DXF files ... 233
6.11.1 Overview ... 233
6.11.2 Displaying CAD drawings ... 234
6.11.2.1 Open a DXF file ... 234

6.11.2.2 Cleaning a DXF file ... 234
6.11.2.3 Enlarging or reducing the CAD drawing ... 235
6.11.2.4 Changing the section ... 236
6.11.2.5 Rotating the view ... 236
6.11.2.6 Displaying/editing information for the geometric data ... 237
6.11.3 Importing and editing a DXF file in the editor ... 238
6.11.3.1 General procedure ... 238
6.11.3.2 Specifying a reference point ... 238
6.11.3.3 Assigning the machining plane ... 239
6.11.3.4 Setting the tolerance ... 239
6.11.3.5 Selecting the machining range / deleting the range and element ... 239
6.11.3.6 Saving the DXF file ... 241
6.11.3.7 Transferring the drilling positions ... 241
6.11.3.8 Accepting contours ... 244

6.12 Importing shapes from CAD programs ... 247
6.12.1 Reading in CAD data into an editor and processing ... 249
6.12.1.1 General procedure ... 249
6.12.1.2 Import from CAD ... 249
6.12.1.3 Defining reference points ... 250
6.12.1.4 Accepting the machining steps ... 252

6.13 Display and edit user variables ... 253
6.13.1 Overview ... 253
6.13.2 Global R parameters ... 254
6.13.3 R parameters ... 255
6.13.4 Displaying global user data (GUD) ... 257
6.13.5 Displaying channel GUDs ... 258
6.13.6 Displaying local user data (LUD) ... 259
6.13.7 Displaying program user data (PUD) ... 260
6.13.8 Searching for user variables ... 260

6.14 Displaying G Functions and Auxiliary Functions ... 263
6.14.1 Selected G functions ... 263
6.14.2 All G functions ... 265
6.14.3 G functions for mold making ... 265
6.14.4 Auxiliary functions ... 266

6.15 Displaying superimpositions ... 268

6.16 Mold making view ... 271
6.16.1 Overview ... 271
6.16.2 Starting the mold making view ... 273
6.16.3 Adapting the mold making view ... 273
6.16.4 Specifically jump to the program block ... 274
6.16.5 Searching for program blocks ... 275
6.16.6 Changing the view ... 276
6.16.6.1 Enlarging or reducing the graphical representation ... 276
6.16.6.2 Moving and rotating the graphic ... 277
6.16.6.3 Modifying the viewport ... 277

6.17 Displaying the program runtime and counting workpieces ... 279

6.18 Setting for automatic mode ... 281

7 Simulating machining 285

7.1 Overview 285

7.2 Simulation before machining of the workpiece 293

7.3 Simultaneous recording before machining of the workpiece 294

7.4 Simultaneous recording during machining of the workpiece 295

7.5 Different views of the workpiece 296
7.5.1 Plan view 296
7.5.2 3D view 297
7.5.3 Side view 297
7.5.4 Turning view 298
7.5.5 Half section 298

7.6 Editing the simulation display 300
7.6.1 Blank display 300
7.6.2 Showing and hiding the tool path 300

7.7 Program control during the simulation 301
7.7.1 Changing the feedrate 301
7.7.2 Simulating the program block by block 302

7.8 Changing and adapting a simulation graphic 303
7.8.1 Enlarging or reducing the graphical representation 303
7.8.2 Panning a graphical representation 304
7.8.3 Rotating the graphical representation 304
7.8.4 Modifying the viewport 305
7.8.5 Defining cutting planes 305

7.9 Displaying simulation alarms 307

8 Generating a G code program 309

8.1 Graphical programming 309

8.2 Program views 310

8.3 Program structure 314

8.4 Fundamentals 315
8.4.1 Machining planes 315
8.4.2 Current planes in cycles and input screens 315
8.4.3 Programming a tool (T) 316

8.5 Generating a G code program 317

8.6 Blank input 318

8.7 Machining plane, milling direction, retraction plane, safe clearance and feedrate (PL, RP, SC, F) 320

8.8 Selection of the cycles via softkey 321

8.9 Calling technology functions 325
8.9.1 Hiding cycle parameters 325
8.9.2 Setting data for cycles 325
8.9.3 Checking cycle parameters 325
8.9.4 Programming variables 326

8.9.5 Changing a cycle call ... 326
8.9.6 Compatibility for cycle support ... 327
8.9.7 Additional functions in the input screens ... 327
8.10 Measuring cycle support ... 328
9 Creating a ShopMill program ... 329
9.1 Program views ... 330
9.2 Program structure ... 335
9.3 Fundamentals ... 336
9.3.1 Machining planes ... 336
9.3.2 Polar coordinates ... 336
9.3.3 Absolute and incremental dimensions ... 337
9.4 Creating a ShopMill program ... 340
9.5 Program header ... 341
9.6 Program header (for milling/turning machine) ... 343
9.7 Generating program blocks ... 346
9.8 Tool, offset value, feed and spindle speed (T, D, F, S, V) ... 347
9.9 Defining machine functions ... 349
9.10 Call work offsets ... 351
9.11 Repeating program blocks ... 352
9.12 Specifying the number of workpieces ... 354
9.13 Changing program blocks ... 355
9.14 Changing program settings ... 356
9.15 Selection of the cycles via softkey ... 358
9.16 Calling technology functions ... 363
9.16.1 Additional functions in the input screens ... 363
9.16.2 Checking input parameters ... 363
9.16.3 Setting data for technological functions ... 363
9.16.4 Changing a cycle call ... 364
9.16.5 Programming variables ... 364
9.16.6 Compatibility for cycle support ... 365
9.17 Measuring cycle support ... 366
9.18 Example, standard machining ... 367
9.18.1 Workpiece drawing ... 368
9.18.2 Programming ... 368
9.18.3 Results/simulation test ... 379
9.18.4 G code machining program ... 381
10 Programming technological functions (cycles) ... 385
10.1 Know-how protection ... 385
10.2 Drilling ... 386
10.2.1 General ... 386
10.2.2 Centering (CYCLE81) ... 387

10.2.3 Drilling (CYCLE82) ... 388
10.2.4 Reaming (CYCLE85) ... 392
10.2.5 Deep-hole drilling 1 (CYCLE83) ... 393
10.2.6 Deep-hole drilling 2 (CYCLE830) ... 398
10.2.7 Boring (CYCLE86) ... 408
10.2.8 Tapping (CYCLE84, 840) ... 410
10.2.9 Drill and thread milling (CYCLE78) ... 417
10.2.10 Positioning and position patterns ... 421
10.2.11 Arbitrary positions (CYCLE802) ... 423
10.2.12 Row position pattern (HOLES1) ... 426
10.2.13 Grid or frame position pattern (CYCLE801) ... 427
10.2.14 Circle or pitch circle position pattern (HOLES2) ... 429
10.2.15 Displaying and hiding positions ... 431
10.2.16 Repeating positions ... 433

10.3 Milling ... 434
10.3.1 Face milling (CYCLE61) ... 434
10.3.2 Rectangular pocket (POCKET3) ... 436
10.3.3 Circular pocket (POCKET4) ... 443
10.3.4 Rectangular spigot (CYCLE76) ... 450
10.3.5 Circular spigot (CYCLE77) ... 455
10.3.6 Multi-edge (CYCLE79) ... 459
10.3.7 Longitudinal groove (SLOT1) ... 463
10.3.8 Circumferential groove (SLOT2) ... 469
10.3.9 Open groove (CYCLE899) ... 475
10.3.10 Long hole (LONGHOLE) - only for G code programs ... 484
10.3.11 Thread milling (CYCLE70) ... 486
10.3.12 Engraving (CYCLE60) ... 490

10.4 Contour milling ... 497
10.4.1 General ... 497
10.4.2 Representation of the contour ... 497
10.4.3 Creating a new contour ... 499
10.4.4 Creating contour elements ... 500
10.4.5 Changing the contour ... 505
10.4.6 Contour call (CYCLE62) - only for G code program ... 506
10.4.7 Path milling (CYCLE72) ... 507
10.4.8 Contour pocket/contour spigot (CYCLE63/64) ... 512
10.4.9 Predrilling contour pocket (CYCLE64) ... 514
10.4.10 Milling contour pocket (CYCLE63) ... 517
10.4.11 Residual material contour pocket (CYCLE63) ... 522
10.4.12 Milling contour spigot (CYCLE63) ... 523
10.4.13 Residual material contour spigot (CYCLE63) ... 527

10.5 Turning - milling/turning machine ... 530
10.5.1 General ... 530
10.5.2 Stock removal (CYCLE951) ... 530
10.5.3 Groove (CYCLE930) ... 534
10.5.4 Undercut form E and F (CYCLE940) ... 538
10.5.5 Thread undercut (CYCLE940) ... 544
10.5.6 Thread turning (CYCLE99), only for G code ... 550
10.5.6.1 Special aspects of the selection alternatives for infeed depths ... 577
10.5.7 Thread chain (CYCLE98) ... 578
10.5.7.1 Special aspects of the selection alternatives for infeed depths ... 587

10.5.8 Cut-off (CYCLE92) ... 588

10.6 Contour turning - Milling/turning machine ... 592
10.6.1 General information ... 592
10.6.2 Representation of the contour ... 593
10.6.3 Creating a new contour ... 594
10.6.4 Creating contour elements ... 596
10.6.5 Changing the contour ... 602
10.6.6 Contour call (CYCLE62) ... 603
10.6.7 Stock removal (CYCLE952) ... 604
10.6.8 Stock removal residual (CYCLE952) ... 619
10.6.9 Grooving (CYCLE952) ... 622
10.6.10 Grooving residual material (CYCLE952) ... 633
10.6.11 Plunge turning (CYCLE952) ... 637
10.6.12 Plunge turning residual material (CYCLE952) ... 647

10.7 Further cycles and functions ... 652
10.7.1 Swivel plane/tool (CYCLE800) ... 652
10.7.2 Swiveling tool (CYCLE800) ... 662
10.7.2.1 Swiveling tool/preloading milling tools - only for G code program (CYCLE800) ... 662
10.7.3 Aligning turning tools (CYCLE800) - millling/turning machine ... 664
10.7.4 High-speed settings (CYCLE832) ... 668
10.7.5 Subroutines ... 672
10.7.6 Adapt to load (CYCLE782) ... 674
10.7.7 Interpolation turning (CYCLE806) ... 676
10.7.7.1 Function ... 676
10.7.7.2 Selecting/deselecting interpolation turning - CYCLE806 ... 677
10.7.7.3 Calling the cycle ... 678
10.7.7.4 Parameter ... 678

10.8 Additional cycles and functions in ShopMill ... 679
10.8.1 Transformations ... 679
10.8.2 Translation ... 680
10.8.3 Rotation ... 680
10.8.4 Scaling ... 681
10.8.5 Mirroring ... 682
10.8.6 Cylinder surface transformation ... 682
10.8.7 Straight or circular machining ... 685
10.8.8 Programming a straight line ... 687
10.8.9 Programming a circle with known center point ... 688
10.8.10 Programming a circle with known radius ... 689
10.8.11 Helix ... 690
10.8.12 Polar coordinates ... 691
10.8.13 Straight polar ... 692
10.8.14 Circle polar ... 693
10.8.15 Obstacle ... 694

11 Multi-channel view ... 697

11.1 Multi-channel view ... 697

11.2 Multi-channel view in the "Machine" operating area ... 698

11.3 Multi-channel view for large operator panels ... 701

11.4 Setting the multi-channel view ... 703

12 Collision avoidance 705
12.1 Activate collision avoidance 707
12.2 Set collision avoidance 708
13 Tool management 711
13.1 Lists for the tool management 711
13.2 Magazine management 713
13.3 Tool types 714
13.4 Tool dimensioning 717
13.5 Tool list 724
13.5.1 Additional data 727
13.5.2 Creating a new tool 728
13.5.3 Measuring the tool 730
13.5.4 Managing several cutting edges 730
13.5.5 Delete tool 731
13.5.6 Loading and unloading tools 731
13.5.7 Selecting a magazine 733
13.5.8 Code carrier connection 734
13.5.9 Managing a tool in a file 736
13.6 Tool wear 739
13.6.1 Reactivating a tool 741
13.7 Tool data OEM 743
13.8 Magazine 744
13.8.1 Positioning a magazine 746
13.8.2 Relocating a tool 746
13.8.3 Deleting / unloading / loading / relocating all tools 747
13.9 Tool details 749
13.9.1 Displaying tool details 749
13.9.2 Tool data 749
13.9.3 Cutting edge data 750
13.9.4 Monitoring data 752
13.10 Changing a tool type 753
13.11 Graphic display 754
13.12 Sorting tool management lists 756
13.13 Filtering the tool management lists 757
13.14 Specific search in the tool management lists 759
13.15 Settings for tool lists 761
13.16 Working with Multitool 762
13.16.1 Tool list for multitool 762
13.16.2 Create multitool 763
13.16.3 Equipping multitool with tools 765
13.16.4 Removing a tool from multitool 766
13.16.5 Deleting multitool 767

13.16.6 Loading and unloading multitool ... 767
13.16.7 Reactivating the multitool ... 768
13.16.8 Relocating a multitool ... 769
13.16.9 Positioning a multitool ... 770

14 Managing programs ... 771

14.1 Overview ... 771
14.1.1 NC memory ... 774
14.1.2 Local drive ... 774
14.1.3 USB drives ... 776
14.1.4 FTP drive ... 776
14.2 Opening and closing the program ... 778
14.3 Executing a program ... 780
14.4 Creating a directory / program / job list / program list ... 782
14.4.1 File and directory names ... 782
14.4.2 Creating a new directory ... 782
14.4.3 Creating a new workpiece ... 783
14.4.4 Creating a new G code program ... 784
14.4.5 Creating a new ShopMill program ... 784
14.4.6 Storing any new file ... 785
14.4.7 Creating a job list ... 786
14.4.8 Creating a program list ... 788
14.5 Creating templates ... 789
14.6 Searching directories and files ... 790
14.7 Displaying the program in the Preview. ... 792
14.8 Selecting several directories/programs ... 793
14.9 Copying and pasting a directory/program ... 795
14.10 Deleting a program/directory ... 797
14.10.1 Deleting a program/directory ... 797
14.11 Changing file and directory properties ... 798
14.12 Set up drives ... 799
14.12.1 Overview ... 799
14.12.2 Setting up drives ... 799
14.13 Viewing PDF documents ... 805
14.14 EXTCALL ... 807
14.15 Execution from external memory (EES) ... 809
14.16 Backing up data ... 810
14.16.1 Generating an archive in the Program Manager ... 810
14.16.2 Generating an archive via the system data ... 811
14.16.3 Reading in an archive in the Program Manager ... 813
14.16.4 Read in archive from system data ... 814
14.17 Setup data ... 815
14.17.1 Backing up setup data ... 815
14.17.2 Reading-in set-up data ... 817

14.18 Recording tools and determining the demand ... 819
14.18.1 Overview ... 819
14.18.2 Opening tool data ... 820
14.18.3 Checking the loading ... 820
14.19 Backing up parameters ... 822
14.20 RS-232-C ... 825
14.20.1 Reading-in and reading-out archives via a serial interface ... 825
14.20.2 Setting V24 in the program manager ... 827
14.21 Multiple clamping ... 829
14.21.1 Multiple clamping ... 829
14.21.2 Program header setting, "Clamping" ... 830
14.21.3 Creating a multiple clamping program ... 831
15 Alarm, error, and system messages ... 833
15.1 Displaying alarms ... 833
15.2 Displaying an alarm log ... 836
15.3 Displaying messages ... 837
15.4 Sorting, alarms, faults and messages ... 838
15.5 Creating screenshots ... 839
15.6 PLC and NC variables ... 840
15.6.1 Displaying and editing PLC and NC variables ... 840
15.6.2 Saving and loading screen forms ... 844
15.7 Version ... 845
15.7.1 Displaying version data ... 845
15.7.2 Save information ... 846
15.8 Logbook ... 847
15.8.1 Displaying and editing the logbook ... 847
15.8.2 Making a logbook entry ... 848
15.9 Remote diagnostics ... 850
15.9.1 Setting remote access ... 850
15.9.2 Permit modem ... 851
15.9.3 Request remote diagnostics ... 852
15.9.4 Exit remote diagnostics ... 853
16 Working with Manual Machine ... 855
16.1 Manual Machine ... 855
16.2 Measuring the tool ... 857
16.3 Measuring the workpiece zero ... 858
16.4 Setting the zero offset ... 859
16.5 Set limit stop ... 860
16.6 Simple workpiece machining ... 861
16.6.1 Traversing axes ... 861
16.6.2 Angular milling ... 862
16.6.3 Straight and circular machining ... 863

16.6.3.1 Straight milling ... 863
16.6.3.2 Circular milling ... 864

16.7 More complex machining ... 866
16.7.1 Drilling with Manual Machine ... 867
16.7.2 Milling with Manual Machine ... 868
16.7.3 Contour milling with manual machine ... 869
16.7.4 Turning with manual machine - milling/turning machine ... 869

16.8 Simulation and simultaneous recording ... 871

17 Teaching in a program ... 873

17.1 Overview ... 873

17.2 Select teach in mode ... 875

17.3 Processing a program ... 876
17.3.1 Inserting a block ... 876
17.3.2 Editing a block ... 876
17.3.3 Selecting a block ... 877
17.3.4 Deleting a block ... 877

17.4 Teach sets ... 879
17.4.1 Input parameters for teach-in blocks ... 880

17.5 Settings for teach-in ... 882

18 Handheld terminals for multi-touch operation ... 883

18.1 HT 8 ... 883
18.1.1 HT 8 overview ... 883
18.1.2 Traversing keys ... 885
18.1.3 Machine control panel menu ... 886
18.1.4 Virtual keyboard ... 888

18.2 HT 10 ... 890
18.2.1 HT 10: Overview ... 890
18.2.2 Machine control panel menu ... 892
18.2.3 Virtual keyboard ... 894

18.3 Calibrating the touch panel ... 895

19 Ctrl-Energy ... 897

19.1 Functions ... 897

19.2 Ctrl-E analysis ... 898
19.2.1 Displaying energy consumption ... 898
19.2.2 Displaying the energy analyses ... 899
19.2.3 Measuring and saving the energy consumption ... 900
19.2.4 Tracking measurements ... 901
19.2.5 Tracking usage values ... 901
19.2.6 Comparing usage values ... 902
19.2.7 Long-term measurement of the energy consumption ... 903

19.3 Ctrl-E profiles ... 904
19.3.1 Using the energy-saving profile ... 904

20 Appendix 907
20.1 Documentation overview SINUMERIK 840D sl 907
Index 909

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 Security information

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. Siemens' products and solutions constitute one element of such a concept.

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

For additional information on industrial security measures that may be implemented, please visit
https://www.siemens.com/industrialsecurity (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 are available and that the latest product versions are used. Use of product versions that are no longer supported, and failure to apply the latest updates may increase customer's exposure to cyber threats.

To stay informed about product updates, subscribe to the Siemens Industrial Security RSS Feed under
https://www.siemens.com/industrialsecurity (https://new.siemens.com/global/en/products/services/cert.html#Subscriptions).

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.

Introduction

2

2.1 Product overview

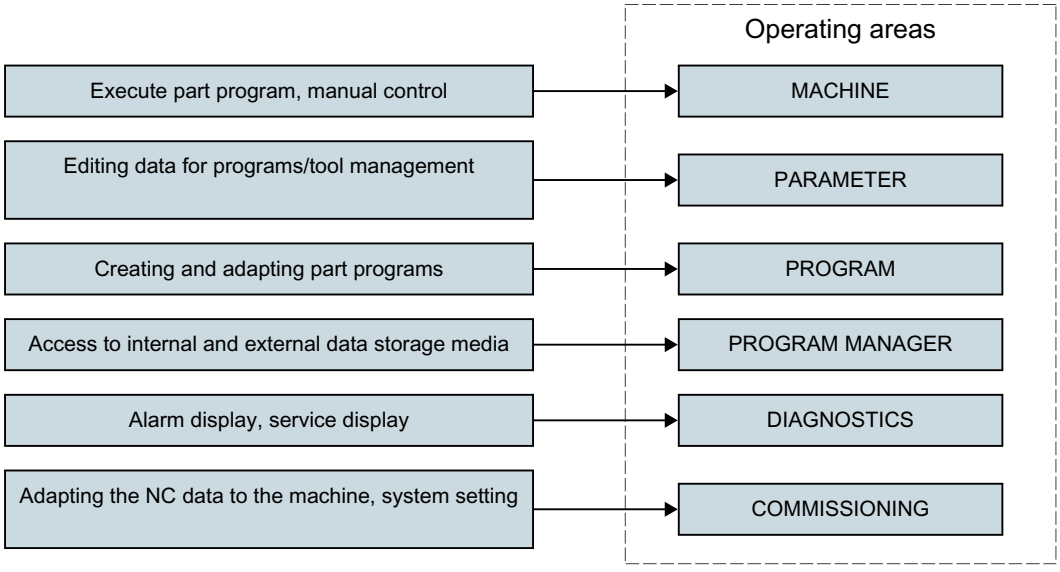
The SINUMERIK control system is a CNC (Computerized Numerical Control) for machine tools.

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:



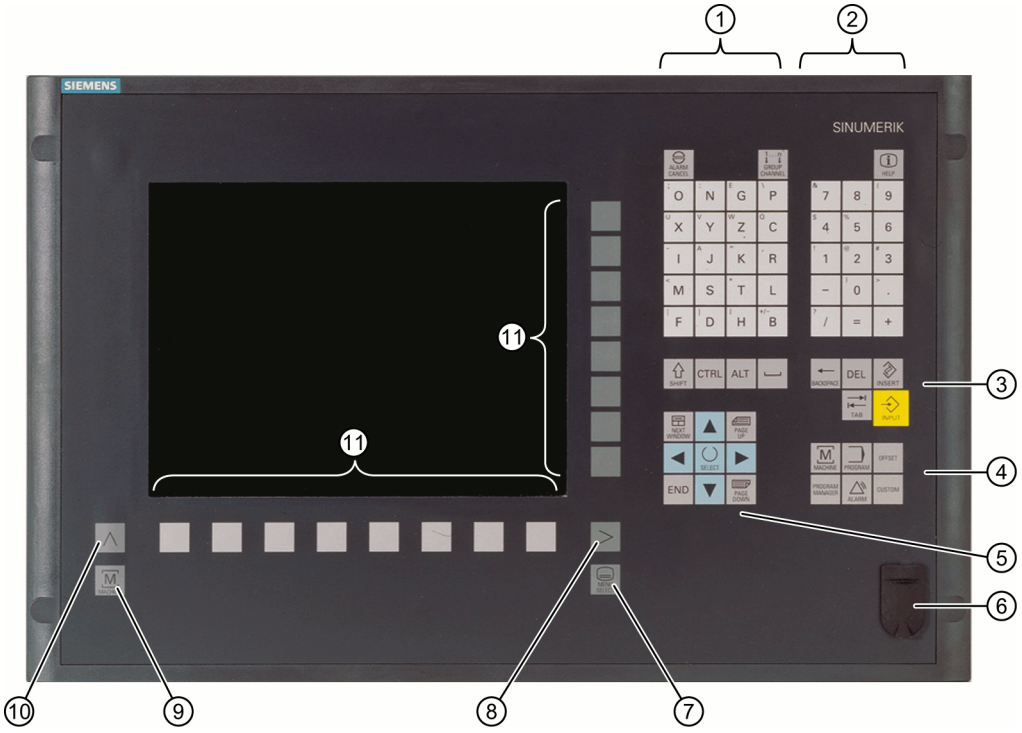
2.2 Operator panel fronts

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:

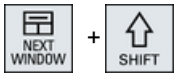
- Operating Components Equipment Manual - Handheld Units (https://support.industry.siemens.com/cs/ww/en/view/109736210)
- Equipment Manual OP 010 (https://support.industry.siemens.com/cs/ww/en/view/109759204)
- Equipment Manual OP 012 (https://support.industry.siemens.com/cs/ww/en/view/109741627)
- Equipment 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.

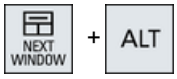
Keys and key combinations

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



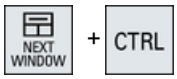
<NEXT WINDOW> + <SHIFT>

- 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>

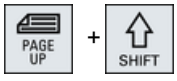
- Moves the cursor to the beginning of a program.
- Moves the cursor to the first row of the current column.

<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.

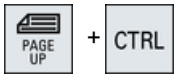
<PAGE UP>

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>

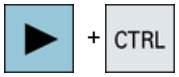
In the program manager and in the program editor, from the cursor position, selects directories or program blocks up to the end of the window.

<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.

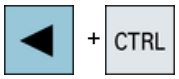


<Cursor right> + <CTRL>

- 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.

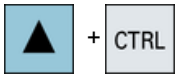


<Cursor left> + <CTRL>

- 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.

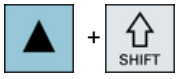
<Cursor up>

- 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.

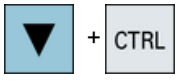


<Cursor up> + <SHIFT>

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

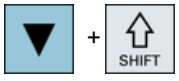
<Cursor down>

- 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>

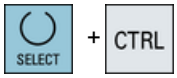
In the program manager and in the program editor, selects a contiguous selection of directories and program blocks.

<SELECT>

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

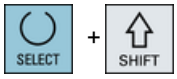
Activates checkboxes.

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.



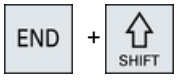
<SELECT> + <SHIFT>

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.

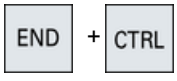
Selects the last entry in selection lists and in selection boxes.



<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>

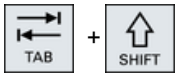
- Editing box
  Deletes a character selected to the left of the cursor.
- Navigation
  Deletes all of the selected characters to the left of the cursor.

<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.

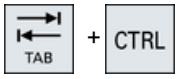
<TAB>

- 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

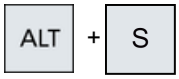
<SHIFT> + <ALT> + <T>

Starts "HMI Trace".

SHIFT + ALT + T

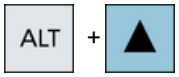
<SHIFT> + <ALT> + <T>

Exits "HMI Trace".



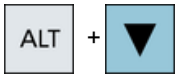
<ALT> + <S>

Opens the editor to enter Asian characters.



<ALT> + <Cursor up>

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

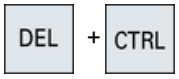


<ALT> + <Cursor down>

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>

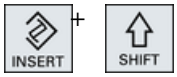
Opens a directory with all of the subdirectories.

<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

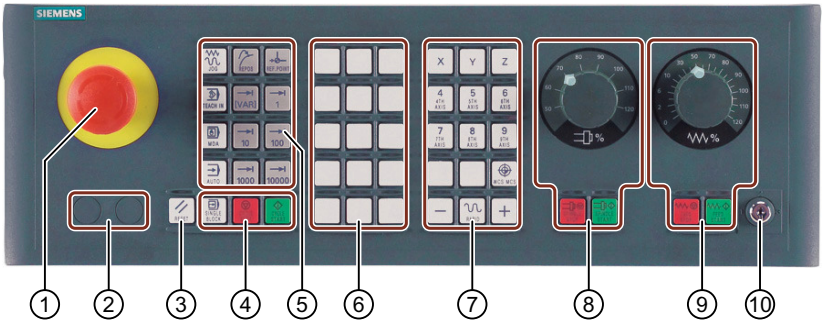
The machine tool can be equipped with a machine control panel by Siemens or with a specific machine control panel from the machine manufacturer.

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.

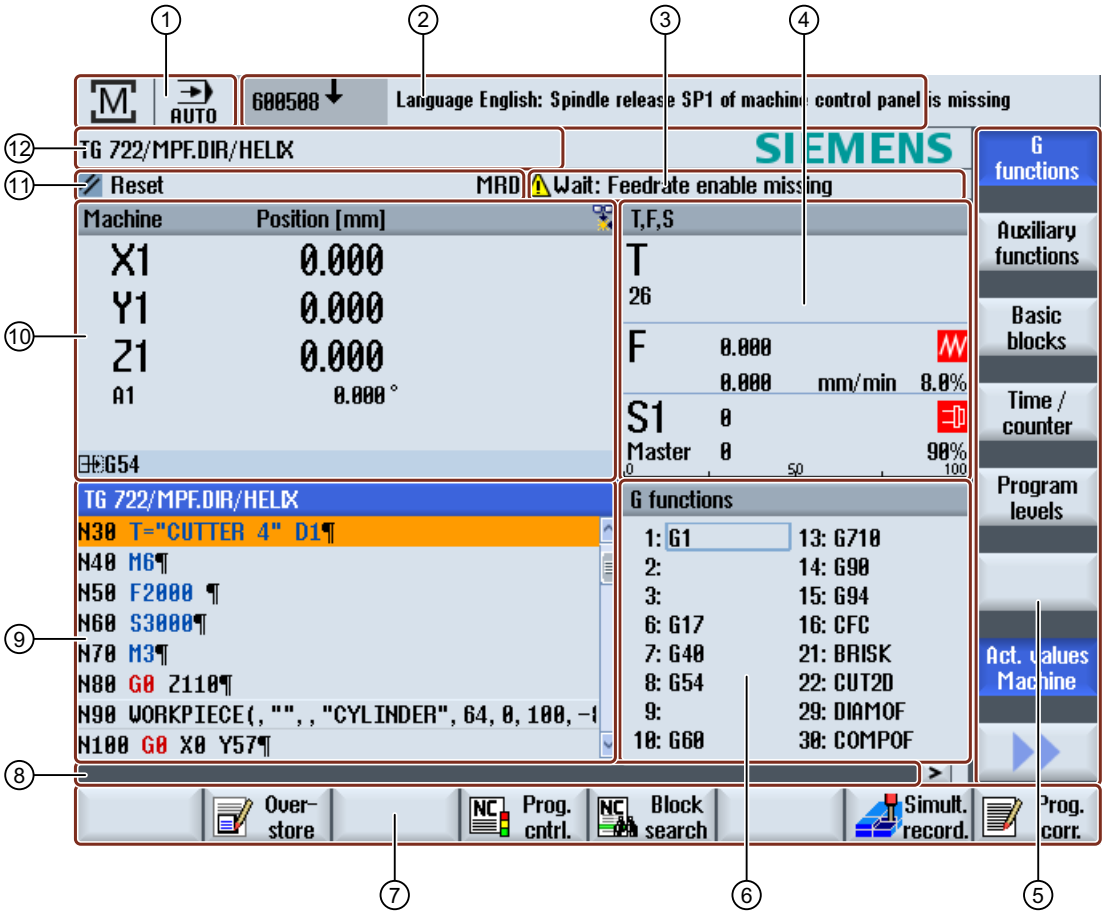
<FEED START>

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 status (S)
   - Spindle utilization rate in percent
   - Name of active tool holder with display of a rotation in space and plane
   - Name of the active kinematic transformation
5. Vertical softkey bar

⑥ Display of active G functions, all G functions, auxiliary functions and input window for different functions (for example, skip blocks, program control).
⑦ Horizontal softkey bar
⑧ Dialog line to provide additional user notes.
⑨ Operating window with program block display
⑩ Position display of the axes in the actual values 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

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

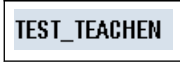
Active mode or function

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

Alarms and messages

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

Second line

Display	Meaning
	Program path and program name

The displays in the second line can be configured.

Machine manufacturer

Please observe the information provided by the machine manufacturer.

Third line

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

The machine manufacturer settings determine which program controls are displayed.

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

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

Overview (Page 120)

Zero offsets (Page 153)

Set collision avoidance (Page 708)

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.

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

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

Tool data

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

Feed data

Display	Meaning
F	
	Feed disable

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

Spindle data

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

Note

Display of logical spindles

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

Machine manufacturer

Please observe the information provided by the machine manufacturer.

2.4.5 Current block display

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

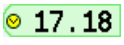
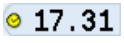
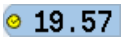
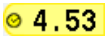
Display of current program

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

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

Setting for automatic mode (Page 281)

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
	Buffer value (Memory Save)
	Retrieve from buffer memory (Memory Recall)
	Delete buffer memory contents (Memory Clear)
	Delete individual character (Backspace)

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

   - OR -

   Press the "DEG" key to calculate the trigonometric functions in radian.
   The designation of the key changes to "RAD".
3. Press the key for the desired trigonometric function, e.g. "SIN".
4. Enter the numerical value.

RAD

DEG

SIN

...

ATAN

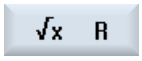
Further mathematical functions

Press the keys in the specified order:

Square number

Number

Square root



Number

Exponential function

Base number

Exponent

Residue class calculation

Number

Divider

Absolute value

Number

Integer component

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

2.4.12.1 Function - input editor

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

Note

Call the input editor with <Alt + S>

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

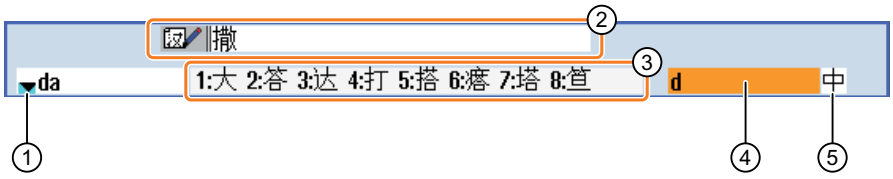
The editor is available for the following Asian languages:

- Simplified Chinese
- Traditional Chinese

Input types

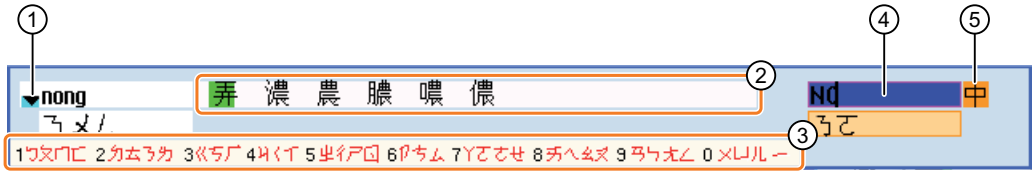
Input type	Description
Pinyin input	Latin letters are combined phonetically to denote the sound of the character. The editor lists all of the characters from the dictionary that can be selected.
Zhuyin input (only traditional Chinese)	Non-Latin letters are combined phonetically to denote the sound of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

Structure of the editor



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

Figure 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.

2.4.12.2 Entering Asian characters

Precondition

The control has been switched over to Chinese.

Procedure

Editing characters using the Pinyin method

+

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

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

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

Editing characters using the Zhuyin method (only traditional Chinese)

+

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

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

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

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

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

2.4.12.3 Editing the dictionary

Learning function of the input editor

Requirement:

The control has been switched over to Chinese.

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

1. The editor provides a further line in which the combined characters and phonetic notations are displayed.
   The first part of the phonetic notation is displayed in the field for selecting the phonetic notation from the dictionary. Various characters are listed for this particular phonetic notation.
2. Press the number key to insert the associated character into the additional line.
   The next part of the phonetic notation is displayed in the field for selecting the phonetic notation from the dictionary.
3. Repeat step 2 until the complete phonetic notation has been compiled.

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

Compiled characters are deleted using the <BACKSPACE> key.

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

Importing a dictionary

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

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

Line structure:

Pinyin phonetic spelling <TAB> Chinese characters <LF>

OR

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

<TAB> - tab key

<LF> - line break

Store the created dictionary in one of the following paths:

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

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

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

Example:

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

2.4.13 Entering Korean characters

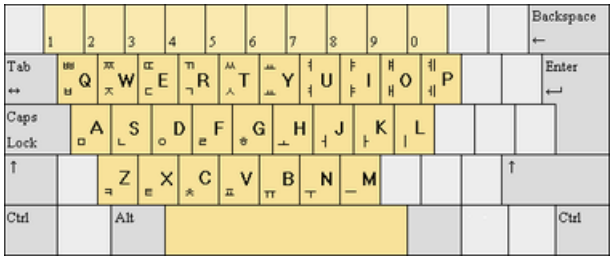
You can enter Korean characters in the input fields on classic panels (without touch operation) using the input editor IME (Input Method Editor).

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 Work station safety

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

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

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

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

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

2.4.16 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.

2.4.17 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.

See also

Additional functions in the input screens (Page 327)

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

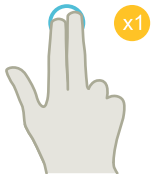
- Thermoplus KCL type 955
- KCL Men at Work type 301
- Camapur Comfort type 619
- Comasec PU (4342)

3.3 Finger gestures

Finger gestures

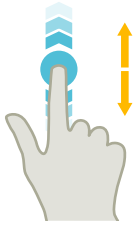
Tap

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



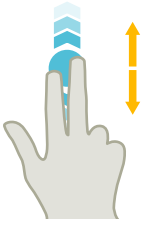
Tap with 2 fingers

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



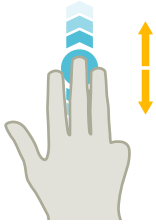
Flick vertically with one finger

- 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

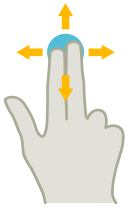
- Zoom in on graphic contents (e.g. simulation, mold making view)

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

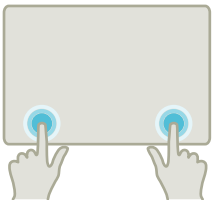
- Rotate graphic contents (e.g. simulation, mold making view)

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

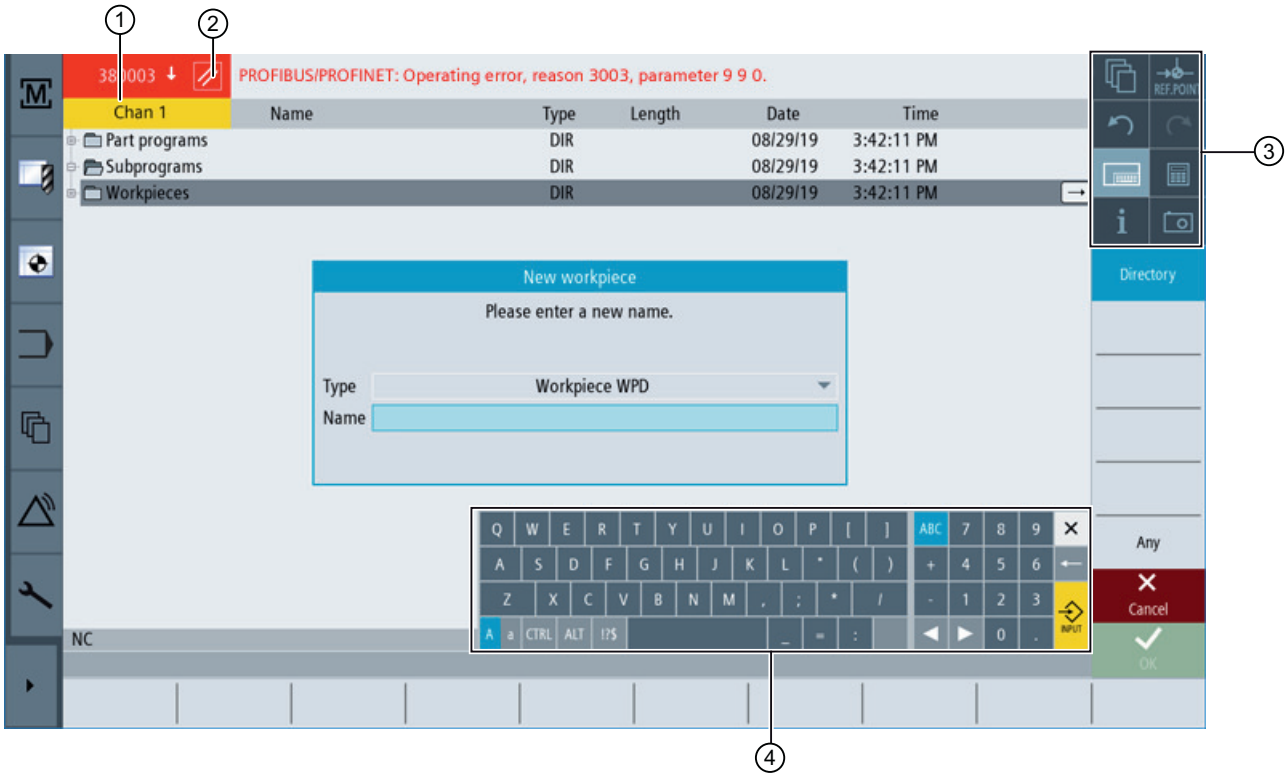
Flicking gestures with several fingers

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.



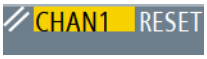
① Changing the channel
② Cancel alarms
③ Function key block
④ Virtual keyboard

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.

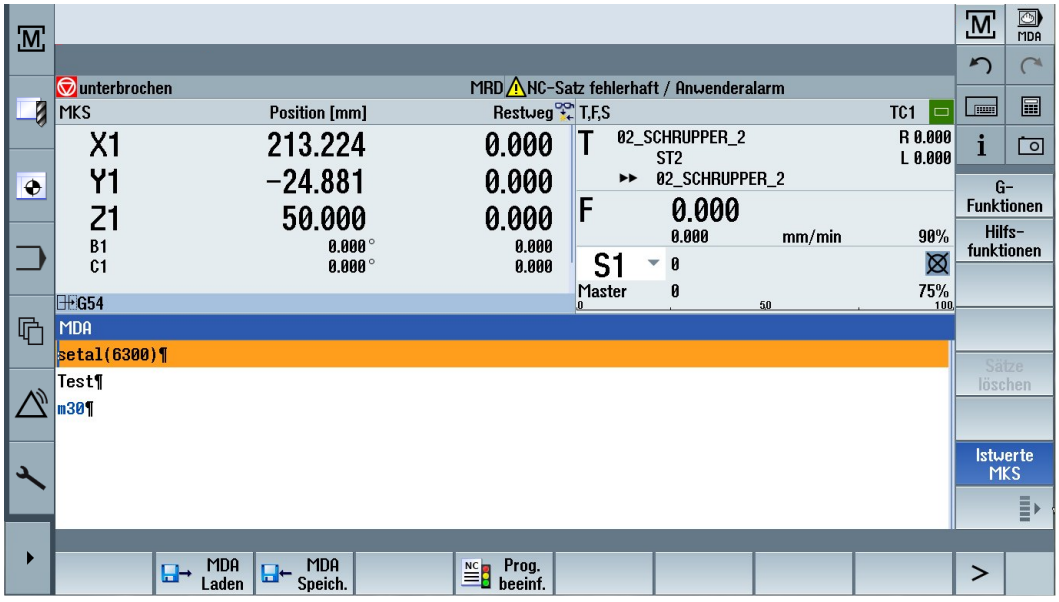
Further information

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.



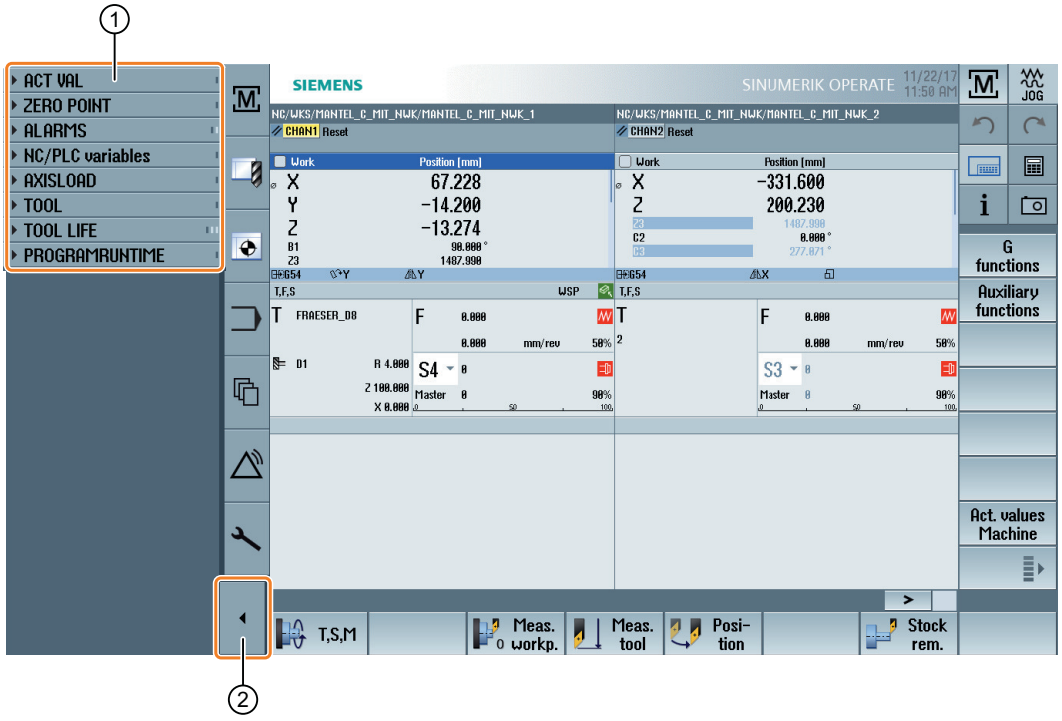
Navigation bar

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.

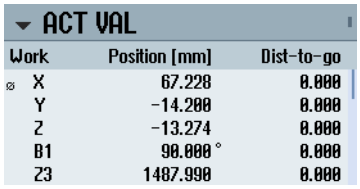
Open widgets

- 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.

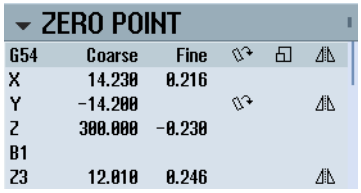
The distance-to-go for the current NC block is displayed while a program is running.



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.



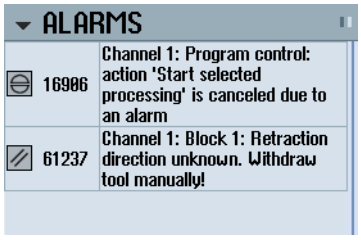
3.5.6 "Alarms" widget

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.

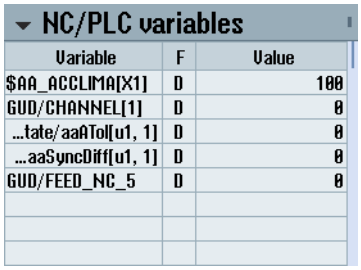
Wipe horizontally to switch between alarms and messages.



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.



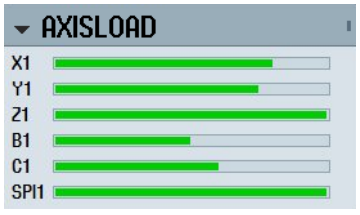
Only those variables that are currently displayed in the "NC/PLC variables" screen in the "Diagnostics" operating area are shown. To update the list in the "NC/PLC variables" widget following a change in the "NC/PLC variables" screen in the "Diagnostics" operating area, collapse and expand the widget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

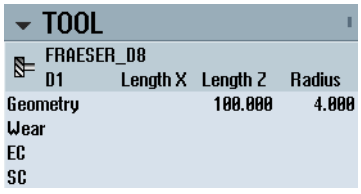


3.5.9 "Tool" widget

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:

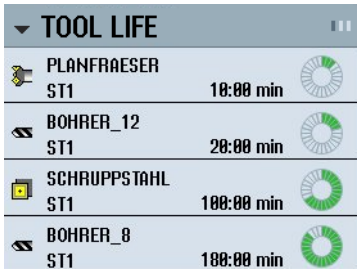
- Operating time of tool (standard time monitoring)
- Finished workpieces (quantity monitoring)
- Tool wear (wear monitoring)

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

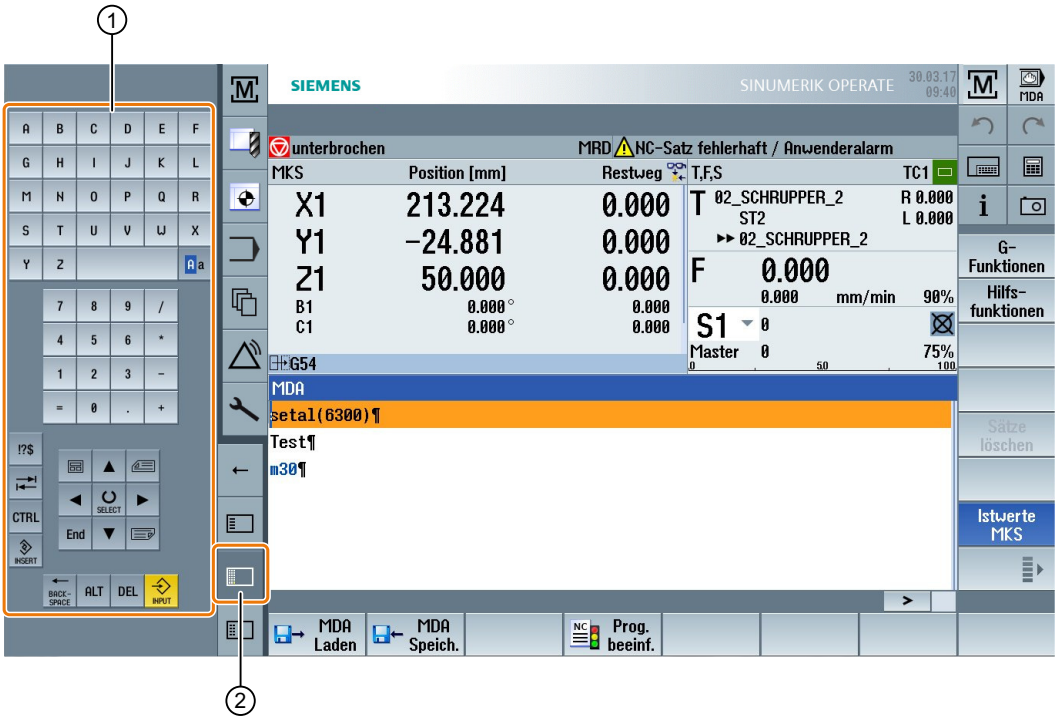
Not only standard widgets but also pages with ABC keyboards and machine control panels can be configured in the sidescreen of a multitouch panel.

Configure ABC keyboard and MCP

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

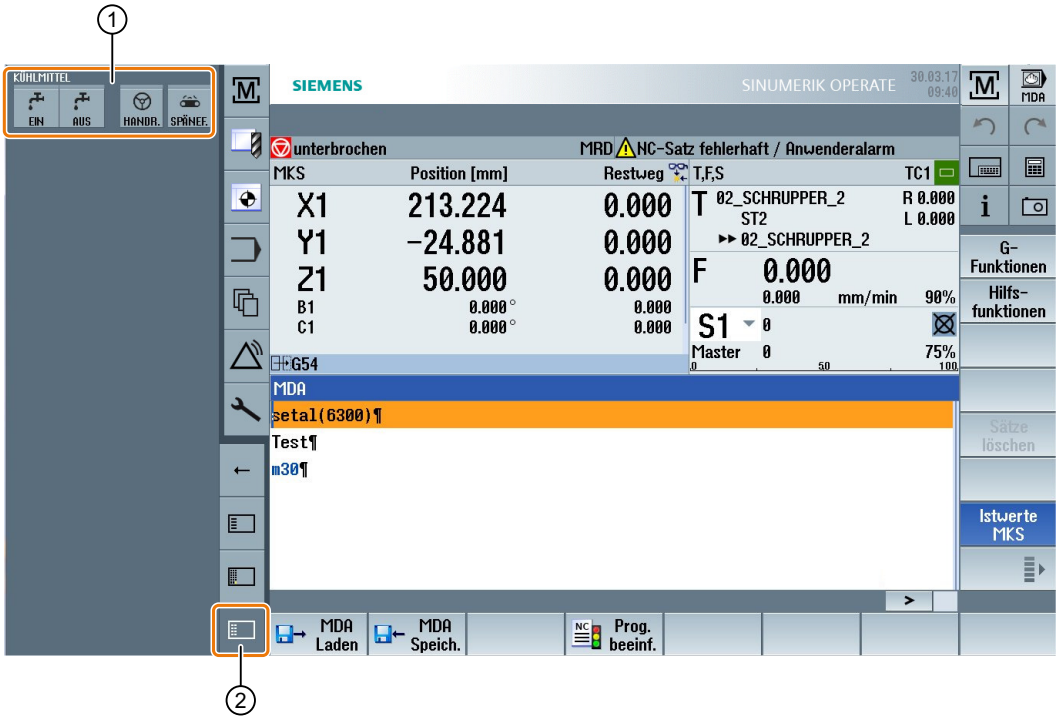
Operator control	Function
	Display of standard widgets in the sidescreen
	Display of an ABC keyboard on the sidescreen
	Display of a machine control panel on the sidescreen

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.

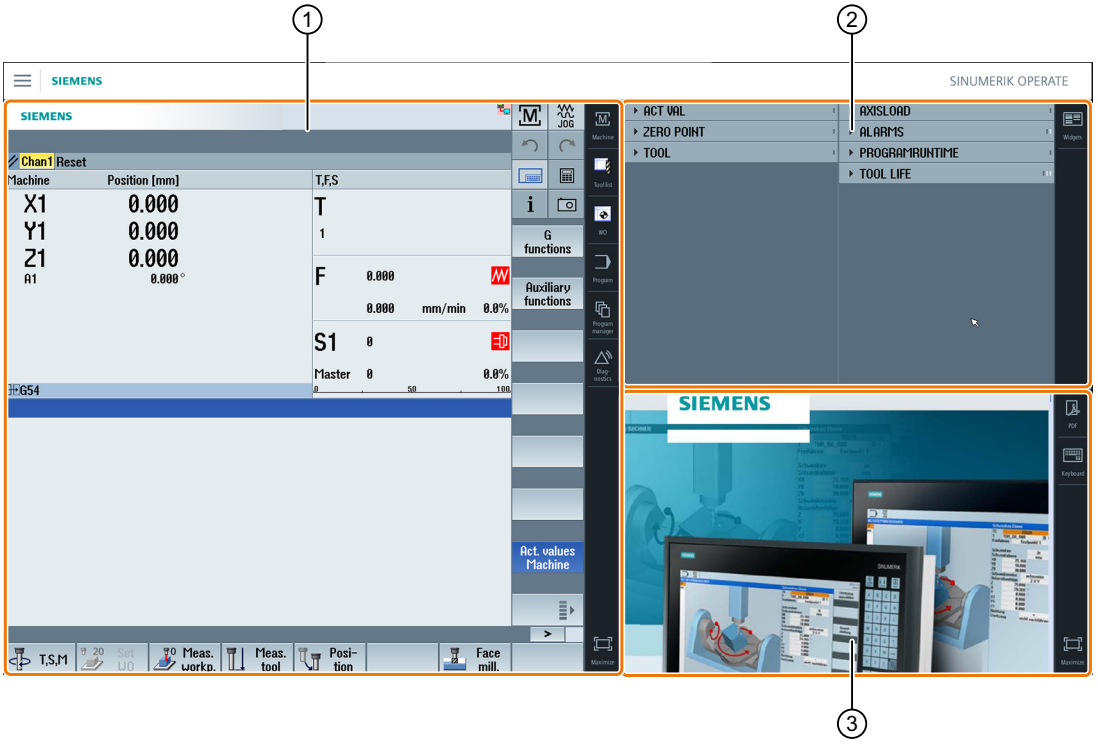
Additional information

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.

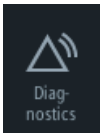


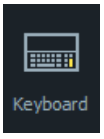
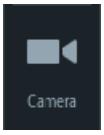
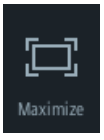
① SINUMERIK Operate with navigation bar for switchover of the operating area
② Display area for standard widgets
③ Display area for applications (e.g. PDF)

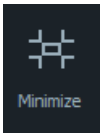
3.6.3 Operator controls

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 82) • Zero point (Page 83) • Tool (Page 84) • Axle load (Page 84) • Alarms (Page 83) • Program runtime (Page 85) • Service life (Page 85) • NC/PLC variables (Page 83)

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

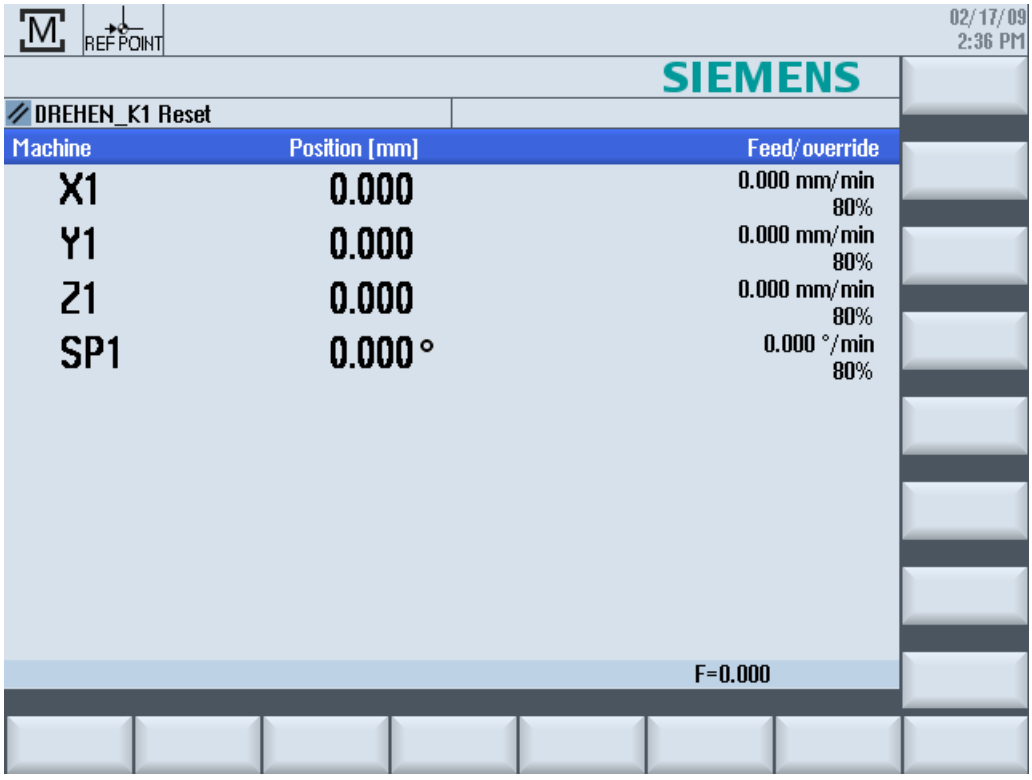
Widget "Camera 1" and "Camera 2" (Page 85)

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 General

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.

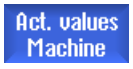
The machine coordinate system (MCS), in contrast to the workpiece coordinate system (WCS), does not take into account any zero offsets, tool offsets and coordinate rotation.

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.

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.

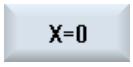
2. Press the "Set ZO" softkey.

- OR -

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

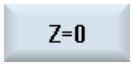
3. Enter the new required position value for X, Y or Z directly in the actual value display (you can toggle between the axes with the cursor keys) and press the "Input" key to confirm the entries.

- OR -

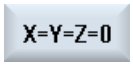


Press softkeys "X=0","Y=0" or "Z=0" to set the relevant position to zero.

...



- 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 Measure tool

4.5.1 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. After this, the controller independently calculates the individual tool path.

Drilling and milling tools

You can determine the tool offset data, i.e. the length and radius or diameter, either manually or automatically with tool probes.

Turning tools (only for milling/turning machine)

You can specify the tool offset data, i.e. the length, either manually or automatically using a tool probe.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Logging the measurement result

After you have completed the measurement, you have the option to output the displayed values in a log. You can define whether the log file that is generated is continually written to for each new measurement, or is overwritten.

See also

Logging tool measurement results (Page 118)

Settings for the measurement result log (Page 151)

4.5.2 Manually measuring drilling and milling tools

For manual measurement, move the tool manually to a known reference point to determine the tool length and the radius or diameter. The control system then calculates the tool offset data from the position of the tool carrier reference point and the reference point.

Reference point

When measuring the tool length you can either use the workpiece or a fixed point in the machine coordinate system, e.g. a mechanical test socket or a fixed point in combination with a distance gauge as the reference point.

When determining the radius/diameter, the workpiece is always used as the reference point

In the machine data, you define whether the radius or the diameter of the tool is to be measured.

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

Note

You can enter the position of the workpiece during measurement.

However, you must declare the position of the fixed point before the measurement.

4.5.3 Measuring drilling and milling tools with the workpiece reference point

Procedure

1. Insert the tool you want to measure in the spindle.
2. Select "JOG" mode in the "Machine" operating area.

3. Press the "Meas. tool" and "Length manual" softkeys.
   The "Length Manual" window opens.

4. Select the cutting edge number D and the number of the replacement tool ST of the tool.
5. Select the "Workpiece" reference point.
6. Approach the workpiece in the Z direction, scratch it with a turning spindle and enter the set position Z0 of the workpiece edge.

7. Press the "Set length" softkey.
   The tool length is calculated automatically and entered in the tool list.

Note

Tool measurement is only possible with an active tool.

4.5.4 Measuring drilling and milling tools with fixed reference point

Procedure

1. Insert the tool you want to measure in the spindle.
2.

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

3. Press the "Meas. tool" and "Length manual" softkeys.
   The "Length Manual" window opens.

...

4. Press the "Tool" softkey to open the tool list, select the desired tool and press the "In Manual" softkey.
   You return to the "Length Manual" window.

5. Select the cutting edge number D and the number of the replacement tool ST of the tool.

6. Select the "fixed point" reference point.

7. If you are measuring with a test socket, enter 0 for offset value "DZ" and approach the fixed point in the Z direction.
   Approaching is performed with a rotating spindle in the opposite direction of rotation. The test socket automatically indicates when the precise position is reached.
   - OR -

If you are using a distance gauge, travel as close to the fixed point as possible, measure the gap with the distance gauge and enter the value in "DZ".

The distance gauge is approached with the spindle stationary.

8. Press the "Set length" softkey.

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

4.5.5 Measuring radius or diameter

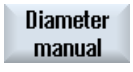
Procedure

1. Insert the tool you want to measure in the spindle.

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

2. Press the "Meas. tool" softkey.

3. Press the "Radius manual" or "Diam. manual" softkey.



4. Select the cutting edge number D and the the number of the replacement tool ST.
5. Approach the workpiece in the X or Y direction and perform scratching with the spindle rotating in the opposite direction.
6. Specify the setpoint position X0 or Y0 of the workpiece edge.

7. Press the "Set radius" or "Set diam." softkey.

The tool radius or diameter is calculated automatically and entered in the tool list.

Note

Tool measurement is only possible with an active tool.

4.5.6 Fixed point calibration

If you want to use a fixed point as the reference point in manual measurement of the tool length, you must first determine the position of the fixed point relative to the machine zero.

Test socket

You can use a mechanical test socket as the fixed point, for example. Mount the test socket on the machine table in the machining space of the machine. Enter zero as the distance.

Distance gauge

However, you can also use any fixed point on the machine in combination with a distance gauge. Enter the thickness of the plate as "DZ".

To calibrate the fixed point, use either a tool whose length is known (i.e. the tool length must be entered in the tool list) or the spindle directly.

The position of the fixed point may have already been determined by the machine manufacturer.

Machine manufacturer
Please refer to the machine manufacturer's specifications

Procedure

1. Traverse the tool or spindle to the fixed point.

2. Press the "Measure tool" softkey in the "JOG" mode.

3. Press the "Calibrate fixed point" softkey.

4. Enter a correction value for "DZ".
   If you have used a distance gauge, enter the thickness of the plate used.

5. Press the "Calibrate" softkey.

6. The distance between machine zero and fixed point is calculated and entered in the machine data.

4.5.7 Measuring the drilling and milling tool length with electrical tool probe

For automatic measurement, you determine the length and radius or diameter of the tool with the aid of a tool probe (table probe system). The control uses the known positions of the toolholder reference point and tool probe to calculate the tool offset data.

Use the softkey to select whether you want to measure the length, the radius or the diameter of the tool.

The corresponding windows can be adapted to the measurement tasks in order to automatically measure tools.

Adapting the user interface to calibrating and measuring functions

The following selection options can be switched-in or switched-out:

- Calibration plane, measurement plane
- Probe
- Calibration feedrate (measuring feedrate)

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

Requirements

- No function-related settings are necessary after the measuring cycles have been installed.
- Before the actual measurement, enter approximate values for length, radius or diameter of the tool in the tool list.
- Calibrate the probe first.

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

Tool offset

Some tool types require an offset for correct length measurement.

The following settings are available:

- Auto
  With a tool that is larger than the probe, the tool edge is set on the center of the probe.
  You can specify an offset correction in the ΔV input field. Select the direction and axis in the adjacent selection field.
- Yes
  The tool edge is positioned centrally on the probe.
  You can specify an offset correction in the ΔV input field. Select the direction and axis in the adjacent selection field.
- No
  The tool is positioned centrally on the probe.
  The fields for the offset correction and specification of the axis and direction are not available.

If the measurement shows that the length of the tool diameter is greater than the probe diameter, measurement is automatically performed with rotating spindle.

Individually checking teeth

Before or after machining you can check if any cutting edges of the milling tool have broken off.

If it is noticed during the check of the cutting edges that not all cutting edges or teeth are present, you will receive a corresponding message.

Procedure

1. Insert the tool that you want to measure.
2.

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

3.

Press the "Meas. tool" softkey.
4.

Press the "Length auto" softkey if you want to measure the length of the tool.

- OR -

Press the "Radius auto" or "Diam. auto", if you wish to measure the radius or diameter of the tool.

5.

Select the cutting edge number D and the number of the replacement tool ST.
6.
If you wish to check the cutting edges of the milling tool, then click "Yes" under "Check teeth individually".
7. If you require a tool offset, select "Yes" or "Auto" in "Tool offset".
8. Enter the tool offset "ΔV" and select the direction and the axis.
9.

Press the <CYCLE START> key.

   This starts the automatic measuring process. When you measure the tool radius or diameter, measurement is performed with a spindle rotating in the opposite direction.

   The tool length, respectively the tool radius or diameter, is calculated automatically and entered in the tool list.

Note

Tool measurement is only possible with an active tool.

4.5.8 Calibrating the electrical tool probe

If you want to measure your tools automatically, you must first determine the position of the tool probe on the machine table with reference to the machine zero.

Tool probes are typically shaped like a cube or a cylindrical disk. Install the tool probe in the working area of the machine (e.g on the machine table) and align it relative to the machining axes.

You must use a mill-type calibration tool to calibrate the tool probe. You must enter the length and radius/diameter of the calibration tool in the tool list beforehand.

Calibration of a tool probe with rotation

To compensate radial eccentricity of the spindle or position deviations of the calibrating tool when calibrating the tool probe, you have the option to calibrate the tool probe with rotation. This entails more exact calibration values of the tool probe and thus more exact measuring values.

During calibration, the probe is withdrawn after the first probing, the spindle is rotated by 180° and probing repeated. A mean value of two values is then determined and entered.

Note

Setting the protection level

The "Calibrate probe" function is only available if an adequate level of protection is set.

Please refer to the machine manufacturer's specifications.

Procedure

1. Move the calibration tool until it is approximately over the center of the measuring surface of the tool probe.

2. Select operating mode "JOG" in the "Machine" operating area and press the "Measure tool" softkey.

3. Press the "Calibrate probe" softkey.

4. Choose whether you want to calibrate the length only, or the length and the diameter.

5. Click in the selection field "Spindle rotation" entry "Yes" if you want to perform the "Calibration with rotation".

6. Press the <CYCLE START> key.
   Calibration is automatically executed at the measuring feedrate. The distance measurements between the machine zero and tool probe are calculated and stored in an internal data area.

4.5.9 Manually measuring a turning tool (for milling/turning machine)

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

Reference point

The workpiece edge is used as the reference point when measuring length X and length Z. The chuck of the main or counterspindle can also be used when measuring in the Z direction.

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

Note

Milling/turning machines with B axis

For milling/turning machines with a B axis, execute the tool change and alignment in the T, S, M window before performing the measurement.

Procedure

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

2. Press the "Meas. tool" softkey.

3. Press the "Manual" softkey.

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

5. Select the tool that you wish to measure.
   The tool edge position must already have been entered in the tool list.

6. Press the "OK" softkey.
   The tool is transferred into the window "Measure: Length manual".

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

8. Scratch the required edge using the tool.

9. If you do not wish to keep the tool at the workpiece edge, then press the "Save position" softkey.
   The tool position is saved and the tool can be retracted from the workpiece. For instance, this can be practical if the workpiece diameter still has to be subsequently measured.
   If the tool can remain at the workpiece edge, then after scratching you can directly continue with step 11.
10. Enter the position of the workpiece edge in X0 or Z0.
    If no value is entered for X0 or Z0, the value is taken from the actual value display.

11. Press the "Set length" softkey.
    The tool length is calculated automatically and entered in the tool list. Whereby the cutting edge position and tool radius or diameter are automatically taken into consideration as well.

Note

Tool measurement is only possible with an active tool.

4.5.10 Manually measuring a turning tool using a tool probe (for milling/turning machine)

During automatic measuring, you determine the tool dimensions in the X and Z axes using a probe.

You have the possibility of measuring a tool using a tool holder that can be oriented (tool carrier, swivel).

The function "Measure with orientable tool carrier" is implemented for lathes with a swivel axis around Y and associated tool spindle. The swivel axis can be used to align the tool at the X/Z level. The swivel axis can assume any position around Y to measure turning tools. Multiples of 90° are

permitted for milling and drilling tools. Multiples of 180° are possible when positioning the tool spindle.

Note

Milling/turning machines with B axis

For milling/turning machines with a B axis, execute the tool change and alignment in the T, S, M window before performing the measurement.

Adapting the user interface to the calibrating and measuring function

The tool offset data is calculated from the known position of the tool carrier reference point and the probe.

You can adapt the corresponding windows to the measurement tasks in order to automatically measure tools.

The following selection options can be switched in or switched out:

- Calibration plane, measurement plane
- Probe
- Calibration feedrate (measuring feedrate)

More information

More information on milling/turning machines with B-axis can be found in the SINUMERIK Operate Commissioning Manual.

Requirements

- If you wish to measure your tools with a tool probe, the machine manufacturer must parameterize special measuring functions for that purpose.
- Enter the cutting edge position and the radius or diameter of the tool in the tool list before performing the actual measurement. If the tool is measured using a tool carrier that can be orientated, then the cutting edge position must be entered into the tool list corresponding to the initial tool carrier position.
- Calibrate the probe first.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure

1. Insert the tool that you want to measure.

   If the tool is to be measured using a tool carrier that can be orientated, then at this position align the tool in the same way that it will be subsequently measured.

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

3. Press the "Meas. tool" and "Length auto" softkeys.

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

5. Manually position the tool in the vicinity of the tool probe in such a way that any collisions can be avoided when the tool probe is being traversed in the corresponding direction.

6. Press the <CYCLE START> key.

   This starts the automatic measuring process. The tool is traversed with the measurement feedrate to the probe and back again.

   The tool length is calculated and entered in the tool list. Whereby the cutting edge position and tool radius or diameter are automatically taken into consideration as well.

   When you measure turning tools with orientable tool carrier around Y using any positions (not multiples of 90°) of the swivel axis, then it should be taken into consideration that the turning tool is measured with the same tool position in both axes X/Z, assuming that this is possible.

4.5.11 Logging tool measurement results

After measuring a tool, you have the option to output the measured values to a log.

The following data are determined and logged:

- Date/time
- Log name with path
- Measuring version
- Input values
- Correction target
- Setpoints, measured values and differences

Note

Logging active

The measurement results can only be entered into a log once the measurement has been fully completed.

Procedure

1. You are in the "JOG" mode and have pressed the "Measure tool" softkey.
   The "Measurement log" softkey cannot be used.
2. Insert the tool, select the measuring version and measure the tool as usual.
   The tool data are displayed once the measurement has been completed.

3. Press the "Measurement log" softkey to save the measurement data as log.
   The "Measurement log" softkey becomes inactive again.

4.6 Measuring the workpiece zero

4.6.1 Overview

The reference point for programming a workpiece is always the workpiece zero. You can determine the workpiece zero on the following workpiece elements:

- Edge (Page 127)
- Corner (Page 132)
- Pocket and hole (Page 135)
- Spigot (Page 138)

Measuring methods

You can measure the workpiece zero either manually or automatically.

Manual measurement

To measure the zero point manually, you need to traverse your tool manually up to the workpiece. You can use edge probes, probes, or dial gauges with known radii and lengths. You can also use any other tool of which you know the radius and length.

The tools used for measuring must not be electronic probes.

Automatic measurement

For automatic measurements, only use the electronic workpiece probe, tool type 710/712. You must calibrate the electronic workpiece probes beforehand.

In the case of automatic measuring, first position the workpiece probe manually. After starting using the <CYCLE START> key, the workpiece probe is automatically extended to the workpiece with the measuring feedrate. Retraction motion from the measuring point is realized as a function of the setting data with the rapid traverse velocity or a user-specific positioning velocity.

Logging measurement results

After you have completed the measurement, you have the option to output the displayed values to a log. You can define whether the log file that is generated is continually written to for each new measurement, or is overwritten.

Measuring with rotation

Under the function "Measuring with rotation" you have the option to measure without prior calibration and without entry of a calibration data set to be used.

To do this, you will need a positionable spindle as well as an electronic 3D workpiece probe. The radius of the probe ball of the electrical probe must be determined once by calibration and entered in the tool data.

Spindle orientation of the probe in the measuring direction

In order to obtain extremely precise measurement results, you have the option to orientate the electronic 3D probe in the measuring direction to avoid deviations regarding the rotation-symmetric shifting characteristics. Orienting the probe is performed by positioning the working spindle into which the probe is clamped.

To do this, you will need a positionable spindle as well as an electronic 3D workpiece probe.

Probe in a machining spindle that cannot be positioned or fixed at the machine

You also have the option to measure workpieces using electronic probes on machines without an SPOS-capable spindle.

To do this, you will need a 3D probe (multi probe type 710). This measurement method requires that the measuring task does not require that the spindle is positioned.

Adapting the user interface to calibrating and measuring functions

Activate the following selection options using setting data:

- Calibration plane, measurement plane
- Calibration feedrate (measuring feedrate)
- Work offset as basis for the measuring process
- Number of the probe calibration data set
- Offset target, adjustable work offset
- Offset target, basis reference
- Offset target, global basis work offset
- Offset target, channel-specific basis work offset
- Standard measuring method
- Measuring with spindle reversal
- Align probe
- The measurement method depends on the probe

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

Note

"Measuring only" for automatic measuring

If "Measuring only" is selected as offset target, then instead of the "Set WO" softkey, the "Calculate" softkey is displayed.

The measuring versions "Set edge", "Rectangular pocket", "Rectangular spigot", "1 circular spigot" and "1 hole" are an exception. For these single-point measurements, for "Measuring only" neither the "Set WO" softkey nor the "Calculate" softkey is listed.

Requirements

- The automatic measurement in the JOG mode is completely installed and functional in the default setting of the control.
- When tool type 710/712 is active, the automatic measuring functions are always executed in the JOG mode.
- Define the user-specific settings (e.g. positioning speed in the working plane or tool axis, length of the measuring distance) using the appropriate parameters.

Machine manufacturer

Please observe the information provided by the machine manufacturer.

More information

More information on user-specific settings when measuring in JOG mode can be found in the SINUMERIK Operate Commissioning Manual.

Selecting the measuring plane

The measuring plane (G17,18,19) can currently be selected to flexibly adapt to measuring tasks. If the measuring plane selection is not activated, then the measurement is performed based on the currently active measuring plane.

Selecting the probe number and the calibration data set number

Workpiece probe calibration data fields can be selected using this function. For different measuring situations, in order to guarantee a high measuring accuracy, it may be necessary to save the corresponding calibration data in different data fields, which can then be selected for the measuring tasks.

If the probe number selection is not activated, then probe number "one" is always used.

Entering the calibration feedrate

The actual calibration feedrate can be entered into this entry field. The calibration feedrate is stored in the calibration data and is used for the measurements.

If the entry field does not exist, then the calibration feedrate from a central parameter is used.

Selecting the work offset as basis for the measurement

A work offset can be selected as measurement basis to flexibly adapt to the measuring tasks.

If the work offset selection as measurement basis is not activated, then the measurement refers to the currently active work offset.

Measuring sequence

To obtain the desired measurement results, you must keep to the measuring point sequence shown in the help displays.

You can reject measuring points and then measure them again. This is done by pressing the softkey that is currently active (measured value).

Measuring only

If you "only" want to measure the workpiece zero, the measured values are calculated and displayed without changing the coordinate system.

Work offset

You usually store the measured workpiece zero in a work offset. The HMI allows rotations and offsets to be measured.

Zero point

The measurement values for the offsets are stored in the coarse offset and the relevant fine offsets are deleted. If the zero point is stored in a non-active work offset, an activation window is displayed in which you can activate this work offset directly.

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

Aligning

Alignment can be carried out either by rotating the coordinate system or by rotating the workpiece with the aid of a rotary axis or, in the case of active swiveling, by swiveling the plane to the basic position. If your machine is equipped with two rotary axes and the "swivel" function is set up, you can also align an inclined plane.

Rotary axes

If your machine has rotary axes, you can include these rotary axes in the measurement and setup procedure. If you store the workpiece zero in a work offset, rotary axis positioning may be necessary in the following cases.

- Correcting the work offset requires you to position the rotary axes to align the workpiece parallel with the coordinate system, e.g. for "Align edge".
- Correcting the work offset rotates the workpiece coordinate system, which should align the tool perpendicular to the plane, e.g. for "Align plane".

One or two activation windows support positioning of the rotary axes.

You can only select "Rotary axis <name of rotary axis>" for the "Angle corr." parameter if your machine has rotary axes.

They must also be assigned to geometry axes via the machine data.

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

See also

Logging measurement results for the workpiece zero (Page 146)

4.6.2 Sequence of operations

To measure the workpiece zero, the workpiece probe must always be located or set perpendicular to the measuring plane (machining plane) (e.g. using "Align plane").

For the measuring versions "Set edge", "Distance 2 edges", "Rectangular pocket" and "Rectangular spigot", the workpiece must first be aligned parallel to the coordinate system.

To do this, it may be necessary to perform the measuring process in several steps.

Possible step sequences

1. "Align plane" (to align the workpiece probe perpendicular to the plane)

2. "Align edge" (to align the workpiece parallel to the coordinate system)

3. "Set edge" or "Distance 2 edges" of "Rectangular pocket" or

"Rectangular spigot", to define the workpiece zero.

- OR -

1. "Align plane" (to align the workpiece probe perpendicular to the plane)

2. "Corner" or "2 holes" or "2 spigots", to align the the coordinate system parallel to the workpiece and to determine the workpiece zero)

Pre-positioning

If you want to preposition a rotary axis before measuring with "Align edge", move the rotary axis so that your workpiece is approximately parallel to the coordinate system.

Set the relevant rotary axis angle to zero with "Set WO". Measurement with "Align edge" will then correct the value for rotary axis offset or include it in the coordinate rotation and align the workpiece edge precisely.

If you want to preposition your workpiece with "Align plane" prior to measurement, you can set the required angular values under "Manual swivel". With "Set zero plane"
you transfer the resulting rotations into the active work offset.

The measurement with "Align plane" will then correct the value for the coordinate rotations and precisely align the workpiece.

If the function "Swivel Manual" is set up on your machine, we recommend that you perform swivel to zero before starting measurement. In that way, you will ensure that the rotary axis positions match the actual coordinate system.

4.6.3 Examples with manual swiveling (swiveling in JOG mode)

Two typical examples demonstrate the interaction and the use of "Measure workpiece" and "Manual swivel" when measuring and aligning workpieces.

First example

The following steps are required when remachining a cylinder head with 2 holes on an inclined plane.

1. Clamp the workpiece
2. T,S,M
   Load the probe and activate the desired work offset.
3. Pre-position the workpiece
   Manually rotate the rotary axes until the inclined surface is almost perpendicular to the tool axis.
4. Manual swivel
   Select "direct" swivel, press the "Teach rotary axes" softkey and press <CYCLE START> key.
5. Manual swivel
   Apply "Set zero plane" to store the resulting rotations in the work zero.
6. Measure workpiece
   Apply "Align plane" to correct the alignment of the workpiece.
7. Measure workpiece
   Apply "2 holes" to define the rotation and offset in the XY plane.
8. Measure workpiece

   Apply "Set edge Z" to define the offset in Z.
9. Start part program to remachine under AUTO.
   Start the program with swivel zero.

Second example

Measuring workpieces in swiveled states. The workpiece is to be probed in the X direction even though the probe cannot approach the workpiece in the X direction because of an obstructing edge (e.g. due to clamping elements). However, with a swivel movement, the measurement in the X direction can be replaced by a measurement in the Z direction.

1. Clamp the workpiece.
2. T,S,M
   Load the probe and activate the desired work offset.
3. Manual swivel
   With "direct" swiveling enter the required rotary axis positions or with "axis by axis" the required rotations (e.g. Y=-90) and <CYCLE START>.
4. Measure workpiece
   Apply "Set edge Z": The measured offset in Z is converted and entered as an X value in the chosen work offset.

4.6.4 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

You can insert any tool in the spindle for scratching when measuring the workpiece zero manually.

- OR -

An electronic workpiece probe is inserted in the spindle and activated when measuring the workpiece zero automatically.

Procedure

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

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

3. Select "Measuring only" if you only want 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 an settable zero offset.

In the window "Zero Offset – G54 ... G599", select a zero offset, in which the zero point should be saved and 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. Select the measuring direction (+ or -) you want to approach the workpiece in.

   For Z0, the workpiece is always approached in the Z minus direction.

7. 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.

8. Traverse the workpiece probe close to the workpiece edge that you wish to measure and press the <CYCLE START> key in order to measure the workpiece zero automatically.

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.

4.6.5 Edge measurement

The following options are available to you when measuring an edge:

Align edge

The workpiece lies in any direction, i.e. not parallel to the coordinate system on the work table. By measuring two points on the workpiece reference edge that you have selected, you determine the angle to the coordinate system.

Distance 2 edges

The workpiece lies parallel to the coordinate system on the work table. You measure distance L of two parallel workpiece edges in one of the axes (X, Y, or Z) and determine its center.

Requirement

You can insert any tool in the spindle for scratching when measuring the workpiece zero manually.

- OR -

An electronic workpiece probe is inserted in the spindle and activated when measuring the workpiece zero automatically.

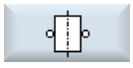
Procedure

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

2. Press the "Workpiece zero" softkey.

3. Press the "Align edge" softkey.

- OR -



Press the "Distance between 2 edges" softkey.

- OR -

If these softkeys are not listed, press any vertical softkey (with the exception of "Set edge") and select the desired measurement variant in the drop-down list.

4. Select "Measuring only" if you only want to display the measured values.

- OR -

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

- OR -

Select work offs.

Press the "Select work offs." softkey to select an settable work offset.

In the window "Work offset – G54 ... G599", select a work offset, in which the zero point should be saved and press the "In manual" softkey.

You return to the measurement window.

G54... G599

In manual

SELECT

6. **Distance 2 edges**:

   Under Meas. dir. P1, select the desired measuring axis in which you want to approach the workpiece and the measuring direction (+ or -) for points of measurement P1 and P2.

   Align edge:

   Under "Measuring axis", select the axis in which you want to approach the workpiece, and the measuring direction (+ or -).

7. **Align edge**:

   If the measuring axis is the tool axis, also select a reference axis in which "P2" is offset to measure the point.

   If the measuring axis is an axis of the plane, no selection is required because the other axis of the plane is automatically the reference axis.

8. **Align edge**:

   After you have set up the rotary axes on the machine and selected a work offset for the correction in step 5, select between offset into a rotation of the coordinate system and into the shift of a rotary axis in the "Angle offs." field.

9. **Distance 2 edges**:

   Enter the reference point (setpoint) for the center of the distance between the 2 edges in the measuring axis.

   Align edge:

   Enter the setpoint angle between reference axis and workpiece edge. The setpoint angle indicates the desired angle between the positive direction of the reference axis and the edge of the workpiece.

   If the setpoint angle=0, the edge is aligned parallel to the reference axis after correction.

10. Traverse the tool to the workpiece edge.

Store P1

11. Press the "Save P1" softkey.

Store P2

12. Reposition the tool and repeat the measuring procedure to measure the second point, and then press the "Save P2" softkey.

13. Press the "Calculate" softkey.

Distance 2 edges

The length or the distance between the 2 edges as well as the center point in the measuring axis between the 2 edges are calculated and displayed.

Align edge

The angle between the workpiece edge and reference axis is calculated and displayed.

- OR -

Press the "Set ZO" softkey.

Distance 2 edges

With "Set WO" the center point in the measuring axis between the 2 edges now corresponds to the selected reference point. The calculated shift is stored in the translation of the work offset.

Align edge

With "Set ZO", the workpiece edge now corresponds to the setpoint angle.

The calculated angle is stored in the rotation of the coordinate system of the work offset or in the shift of a rotary axis. When correcting for the shift of the rotary axis, the actual position of the rotary axis is taken into account in addition to the angle between reference axis and workpiece edge and the setpoint angle. The correction value is calculated so that the alignment of the edge to the setpoint angle can be achieved by positioning the rotary axis at 0 degrees.

Note

Settable work offsets

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.

Automatic measurement

1. Prepare the measurement (see steps 1 to 9 above).
2. Traverse the workpiece probe close to the workpiece edge on which you wish to measure and press the <CYCLE START> key.

This starts the automatic measuring process. The position of measuring point 1 is measured and stored.

The "P1 stored" softkey becomes active.

3. Repeat the operation to measure and store P2.

4. Press the "Calculate" softkey.

Distance 2 edges

The length or the distance between the 2 edges as well as the center point in the measuring axis between the 2 edges are calculated and displayed.

Align edge

The angle between the workpiece edge and reference axis is calculated and displayed.

- OR -

Press the "Set ZO" softkey.

Distance 2 edges

With "Set WO" the center point in the measuring axis between the 2 edges now corresponds to the selected reference point. The calculated shift is stored in the translation of the work offset.

Align edge

With "Set ZO", the workpiece edge now corresponds to the setpoint angle.

The calculated angle is stored in the rotation of the coordinate system of the work offset or in the shift of a rotary axis. When correcting for the shift of the rotary axis, the actual position of the rotary axis is taken into account in addition to the angle between reference axis and workpiece edge and the setpoint angle. The correction value is calculated so that the alignment of the edge to the setpoint angle can be achieved by positioning the rotary axis at 0 degrees.

Align the edge by positioning the rotary axis

After you have performed the angle correction of the workpiece edge in the shift of a rotary axis, the edge of the workpiece can be aligned to the selected setpoint angle by positioning the rotary axis. The rotary axis is positioned to the position of 0 degrees.

Note

Retract the probe to a safe position before the rotary axis is due to move.

The activation window asking whether you want to "Position rotary axis X to align?" is open.

1. Select "Yes" if you want to position the rotary axis.

   An input field for the feedrate and the softkey "Rapid traverse" are displayed.

2. Press the "Rapid traverse" softkey to enter the feedrate in rapid traverse.

   - OR -

   Enter the desired feedrate into input field "F".

3. Press the <CYCLE START> key.

   The rotary axis is repositioned.

Aligning the edge by swiveling

If a swivel data record (tool carrier) is active, you can align the edge to the setpoint angle by swiveling. The basic position of the swivel data record is approached.

The activation window with the query "Align edge?" is opened.

1. Select "Yes" if you want to align the edge by swiveling to the basic position.
   If it has been defined in the swivel data record, corresponding prompts appear for retracting before swiveling, the preferred direction for swiveling, and tracking of the tool tip when swiveling.
   Answer the queries.

2. Select the relevant parameters.

3. Press the <CYCLE START> key.
   After the axes have been retracted, the edge of the workpiece is aligned with the setpoint angle by swiveling into the basic position.

4.6.6 Measuring a corner

You have the option to measure workpiece corners, which are defined by a right angle (90°) or any inner angle.

Measuring a right-angled corner

The workpiece corner to be measured has a 90° inner angle and is clamped to the worktable in any position. By measuring 3 points you can determine the corner point (point of intersection of the angle side) in the working plane and angle α between the workpiece reference edge (line through P1 and P2) and the reference axis in the working plane (1st geometry axis of the working plane).

Measuring any corner

The workpiece corner to be measured has any (not right-angled) inner angle and is clamped at any position on the worktable. By measuring four points you can determine the corner point (point of intersection of the angle sides) in the working plane and angle α between the workpiece reference edge (line through P1 and P2) and the reference axis in the working plane (1st geometry axis of the working plane) and inner angle β of the corner.

Note

The coordinate system shown in the help displays is always in relation to the currently set workpiece coordinate system.

Please be aware of this if you have swiveled or changed the WCS in any other form.

Precondition

You can insert any tool in the spindle for scratching when measuring the workpiece zero manually.

- OR -

An electronic workpiece probe is inserted in the spindle and activated when measuring the workpiece zero automatically.

Procedure

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

2. Press the "Workpiece zero" softkey.

3. Press the "Right-angled corner" softkey if the workpiece has a right-angled corner.

   - OR -

   Press the "Any corner" softkey, if you want to measure a corner not equal to 90°.

   - OR -

   If these softkeys are not listed, press any vertical softkey (with the exception of "Set edge") and in the drop-down list, select the desired measurement version.

4. Select "Measuring only" if you only want to display the measured values.

   - OR -

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

   - OR -

   Press the "Select WO" softkey to select a settable work offset.

   In the window "Work offset – G54 ... G599", select a work offset, in which the zero point should be saved and press the "In manual" softkey.

   You return to the measurement window.

6. Select the corner (inside corner or outside corner) that you wish to measure and its position (position 1... position 4).

   The position of the measuring points appear in the help display.

7. Specify the setpoint of the workpiece corner (Z0, X0) you want to measure.
8. Traverse the tool (acc. to help display) to the first measuring point P1 if you are measuring manually.

9. Press the "Save P1" softkey.

   The coordinates of the first measuring point are measured and stored.

10. Reposition the spindle holding the tool each time, approach measuring points P2 and P3 and press the "Save P2" and "Save P3" softkeys.

11. Repeat the procedure to measure the fourth measuring point when you measure any corner.

12. Press the "Calculate" softkey.

    The corner point and angle α are calculated and displayed.

    - OR -

13. Press the "Set WO" softkey.

    The corner point now corresponds to the setpoint position. The calculated offset is stored in the work offset.

Note

Settable work offsets

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 refer to the machine manufacturer's specifications.

Automatic measurement

1. Prepare the measurement (see steps 1 to 6 above).

2. Approach measuring point P1 with the workpiece probe and press the <CYCLE START> key.

   This starts the automatic measuring process. The position of measuring point 1 is measured and stored.

   The "P1 stored" softkey becomes active.

3. Repeat the operation to measure and store points P2 and P3.

P4 stored

If you are measuring a corner not equal to 90°, repeat the procedure to measure and store point P4.

Calculate

4. Press the "Calculate" softkey.

The corner point and angle α are calculated and displayed.

- OR -

Set WO

Press the "Set WO" softkey.

The corner point now corresponds to the setpoint position. The calculated offset is stored in the offset target that you have selected.

4.6.7 Measuring a pocket and hole

You can measure rectangular pockets and one or more holes and then align the workpiece.

Measuring a rectangular pocket

The rectangular pocket must be aligned at right-angles to the coordinate system. By automatically measuring four points inside the pocket, its length, width and center point can be determined.

Measuring one hole

The workpiece with the hole to be measured is clamped to the work table in any position. In the hole, four points are automatically measured, and from this measurement, the diameter and center point of the hole are determined.

Measuring two holes

The workpiece with the two holes to be measured is clamped to the work table in any position. Four points are automatically measured in both holes and the hole centers are calculated from them. Angle α is calculated from the connecting line between both center points and the reference axis, and the new zero point that corresponds to the center point of the 1st hole is determined.

Measuring three holes

The workpiece with the three holes to be measured is clamped to the work table in any position. Four points are automatically measured in the three holes and the hole centers are calculated from them. A circle is placed through the three center points. The center point and the diameter are determined from this circle. This center point represents the new workpiece zero to be determined. When an angular offset is selected, the base angle of rotation α can also be determined.

Measuring four holes

The workpiece with the four holes to be measured is clamped to the work table in any position. Four points are automatically measured in the four holes and the hole centers are calculated from them. Two hole center points are diagonally connected in each case. The point of intersection is determined from the two lines that are obtained. This point of intersection

represents the new workpiece zero to be determined. When an angular offset is selected, the base angle of rotation α can also be determined.

Note

"Measuring only" for automatic measuring

If "Measuring only" is selected as offset target, then instead of the "Set WO" softkey, the "Calculate" softkey is displayed.

The measuring versions "Rectangular pocket" and "1 hole" are an exception. For these single-point measurements, for "Measuring only" neither the "Set WO" softkey nor the "Calculate" softkey is listed.

Note

You can only measure 2, 3, and 4 holes automatically.

Precondition

You can insert any tool in the spindle for scratching when measuring the workpiece zero manually.

- OR -

An electronic workpiece probe is inserted in the spindle and activated when measuring the workpiece zero automatically.

Procedure

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

2. Press the "Workpiece zero" softkey.

3. Press the "Rectangular pocket" softkey.

- OR -

Press the "1 hole" softkey.

- OR -

If these softkeys are not listed, press any vertical softkey (with the exception of "Set edge") and in the drop-down list, select the desired measurement version.

4. Select "Measuring only" if you only want to display the measured values.

- OR -

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

- OR -

Press the "Select WO" softkey to select a settable work offset.

In the window "Work Offset – G54 ... G599", select a work offset, in which the zero point should be saved and press the "In manual" softkey.

You return to the measurement window.

6. Specify the position setpoints (X0/Y0) of the pocket center point or hole center point.
7. Traverse the tool to the first/next measuring point if you are measuring manually.

8. Press the "Save P1" softkey.

   The point is measured and stored.

9. Repeat steps 6 and 7 to measure and save measuring points P2, P3 and P4.

...

10. Press the "Calculate" softkey.

    The length, width, and center point of the rectangular pocket or diameter and center point of the hole are calculated and displayed.

    - OR -

Press the "Set WO" softkey.

The setpoint position of the center point is stored as a new zero point with "Set WO". The tool radius is automatically included in the calculation.

Note

Settable work offsets

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 refer to the machine manufacturer's specifications.

4.6.8 Measuring a spigot

You have the option to measure and align rectangular spigots, and one or more circular spigots.

Measuring a rectangular spigot

The rectangular spigot should be aligned at right-angles to the coordinate system. By measuring four points at the spigot you can determine the length, width, and center point of the spigot.

Please note that the straight lines between points P1 and P2 or P3 and P4 must intersect with one another, in order that a measurement result is displayed.

Measuring one circular spigot

The workpiece is located anywhere on the work table and has a circular spigot. You can determine the diameter and center point of the spigot with four measuring points.

Measuring two circular spigots

The workpiece is located anywhere on the work table and has 2 spigots. Four points are automatically measured at the two spigots and the spigot centers are calculated from them. The angle α is calculated from the connecting line between both center points and the reference axis, and the new zero point that corresponds to the center point of the first spigot is determined.

Measuring three circular spigots

The workpiece is located anywhere on the work table and has three spigots. Four points are automatically measured at the three spigots and the spigot centers are calculated from them. A circle is placed through the three center points and the circle center and circle diameter are determined.

When an angular offset is selected, the base angle of rotation α can also be determined.

Measuring four circular spigots

The workpiece is located anywhere on the work table and has four spigots. four points are automatically measured at the four spigots and the spigot centers are calculated from them. Two spigot center points are each connected diagonally and the intersection point of the two lines is then determined. When an angular offset is selected, the base angle of rotation α can also be determined.

Note

"Measuring only" for automatic measuring

If "Measuring only" is selected as offset target, then instead of the "Set WO" softkey, the "Calculate" softkey is displayed.

The measuring versions "Rectangular spigot" and "1 circular spigot" are an exception. For these single-point measurements, for "Measuring only" neither the "Set WO" softkey nor the "Calculate" softkey is listed.

Note

You can only measure 2, 3, and 4 circular spigots automatically.

Precondition

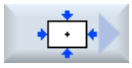
You can insert any tool in the spindle for scratching when measuring the workpiece zero manually.

An electronic workpiece probe is inserted in the spindle and activated when measuring the workpiece zero automatically.

Procedure

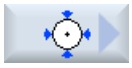
1. Select the "Machine" operating area and press the <JOG> key.

2. Press the "Workpiece zero" softkey.



3. Press the "Rectangular spigot" softkey.

- OR -



Press the "1 circular spigot" softkey.

- OR -

If these softkeys are not listed, press any vertical softkey (with the exception of "Set edge") and in the drop-down list, select the desired measurement version.

4. Select "Measuring only" if you only want to display the measured values.

- OR -

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

- OR -

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

You return to the "1 Circular Spigot" window.

The selection of work offsets can differ.

Please refer to the machine manufacturer's specifications.

5. Specify the position setpoints (X0/Y0) of the spigot center point P0.
6. Traverse the tool to the first measuring point.

7. Press the "Save P1" softkey.
   The point is measured and stored.

8. Repeat steps 6 and 7 to measure and save measuring points P2, P3 and P4.

9. Press the "Calculate" softkey.
   The diameter and center point of the spigot are calculated and displayed.
   - OR -

   Press the "Set WO" softkey.
   The setpoint position of the center point is stored as a new zero point with "Set WO". The tool radius is automatically included in the calculation.

Note

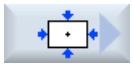
Settable work offsets

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).

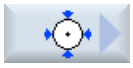
Please refer to the machine manufacturer's specifications.

Automatic measurement

1. Select the "Measure workpiece zero" function (see steps 1 and 2 above).
2. Press the "Rectangular spigot" softkey.



   - OR -



   Press the "1 circular spigot" softkey.

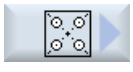
   - OR -

   Press the "2 circular spigots" softkey.

   - OR -

   Press the "3 circular spigots" softkey.

   - OR -



   Press the "4 circular spigots" softkey.

   - OR -

   If these softkeys are not listed, press any vertical softkey (with the exception of "Set edge") and in the drop-down list, select the desired measurement version.

3. Traverse the workpiece probe to approximately the center above the rectangular or circular spigot, or for several, above the first spigot to be measured.
4. Specify whether you want "Measurement only" or in which work offset you want to store the zero point.

Rectangular spigot

5.
- Enter the infeed value in "DZ" to determine the measuring depth.
- In field "L" enter the length (1st geometry axis of the working plane) and in field "W" the width (2nd geometry axis of the working plane) of the spigot.

- OR -

1 circular spigot

- Enter the approximate diameter of the spigot into "Øspigot".
- Enter an angle in "Probe angle". With the probe angle you can turn the traversing direction of the probe through any angle.

- OR -

2 circular spigots

- Enter the approximate diameter of the spigot into "Øspigot".
- Enter the infeed value in "DZ" to determine the measuring depth.
- Under "Angle offs.", select entry "Coor. rotation" or "Rotary axis A, B, C".
- Enter the setpoint angle.
- Enter the position setpoints (Z0/X0) for the center point of the first spigot.

The setpoint angle refers to the 1st axis of the working plane (X/Y plane).

The input fields for the setpoint positions are only active if you selected the angle offset via coordinate rotation.

- OR -

3 circular spigots

- Enter the approximate diameter of the spigot into "Øspigot".
- Enter the infeed value in "DZ" to determine the measuring depth.
- In "Angle offs.", select entry "No", or in "Angle offs." select entry "Yes" if you want to align using coordinate rotation.
- Specify the setpoint angle if you selected entry "Yes" for "Angle offs."
- Enter the setpoint positions Z0 and X0 to determine the center point of the circuit on which the center points of three spigots lie.

The setpoint angle refers to the 1st axis of the working plane (X/Y plane). This input field only appears if you specified "Yes" for "Angle offs."

- OR -

4 circular spigots

- Enter the approximate diameter of the spigot into "Øspigot".
- Enter the infeed value in "DZ" to determine the measuring depth.
- In "Angle offs.", select entry "Yes" if you want to align using coordinate rotation or select in "Angle offs." the entry "No".
- Enter the setpoint angle.
- Enter the setpoint positions X0 and Y0 to determine the point of intersection of the connecting lines between the spigot center points.

The setpoint angle refers to the 1st axis of the working plane (X/Y plane). This input field only appears if you specified "Yes" for "Angle offs."

4. Press the <CYCLE START> key.

   This starts the automatic measuring process. The tool automatically measures four points in succession around the rectangular or spigot outer wall or the outer wall of the first spigot if several spigots are to be measured.

   After the measurement has been successfully completed, the center of the spigot is determined and the "P1 stored" softkey becomes active.

5. If you are measuring several spigots, then move the tool approximately to the center of the second, third, and fourth spigot and press the <CYCLE START> key.

   After the measurement has been successfully completed, P2, P3 and P4 are stored and the softkeys "P2 stored", "P3 stored", and "P4 stored" become active.

...

6. Press the "Calculate" or "Set WO" softkey.

Rectangular spigot

The length, width, and center point of the rectangular spigot are calculated and displayed.

For "Set WO", the setpoint position of the center point is stored as new zero point. The tool radius is automatically included in the calculation.

1 spigot

The diameter and center point of the spigot are calculated and displayed.

For "Set WO", the setpoint position of the center point is stored as new zero point. The tool radius is automatically included in the calculation.

2 spigots

The angle between the line connecting the center points and the reference axis is calculated and displayed.

For "Set WO", the center point of the first spigot now corresponds to the position setpoint. The calculated rotation is stored in the work offset.

3 spigots

The center point and the diameter of the circle on which the three spigot center points lie are calculated and displayed. If you have selected "Yes" in "Coor. rot.", angle α is additionally calculated and displayed.

For "Set WO", the center point of the circle now corresponds to the position setpoint. The calculated rotation is stored in the work offset.

4 spigots

The spigot center points are connected diagonally and the intersection point of the two connecting lines calculated and displayed. If you have selected "Yes" in "Coor. rot.", angle α is additionally calculated and displayed.

For "Set WO", the intersection point now corresponds to the position setpoint. The calculated rotation is stored in the work offset.

4.6.9 Aligning the plane

You can measure an inclined plane of a workpiece in space and determine rotation angles α and β. By subsequently performing coordinate rotation, you can align the tool axis perpendicular to the workpiece plane.

In order to determine the position of the plane in space, three different points are measured along the tool axis. To vertically align the tool axis, the "Swivel" function or the 5 axis transformation (TRAORI) must be set-up at the machine.

In order to be able to measure the plane, the surface must be flat.

Requirement

You can insert any tool in the spindle for scratching when measuring the workpiece zero manually.

An electronic workpiece probe is inserted in the spindle and activated when measuring the workpiece zero automatically.

Procedure

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

2. Press the "Meas. zero" and "Align plane".
   The "Align plane" window opens.

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

   - OR -

   Select the desired work 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 appropriate measurement window.

You return to the "Align plane" window.

The selection of zero offsets can differ.

Please refer to the machine manufacturer's specifications.

4. Traverse the tool to the first measuring point that you want to determine.

5. Press the "Save P1" softkey.

6. Then traverse the tool to the second and third measuring point and press the "Save P2" and "Save P3" softkeys.

7. If you selected "Measuring only", press the "Calculate" softkey.

Angles α and β are calculated and displayed.

8. Press the "Set WO" softkey to store the angle offset in the work offset.

The query "Activate work offset Gxxx now?" opens.

9. Press the "OK" softkey to activate the corrected work offset.

Aligning and retracting the tool

If you stored the measured workpiece zero in a work offset and subsequently activate it again, rotation of the workpiece coordinate system may make it necessary to realign the tool with the plane.

The activation window asking whether you want to "Position measuring probe perpendicular to plane?" opens.

1. Select "Yes" if you want to swivel into the plane.

The query "Positioning by swiveling! Retract?" is displayed.

2. Select the relevant parameters.

3. Press the <CYCLE START> key.

   When the axis has been retracted the tool is realigned with the help of the swivel cycle.

   You can now measure again.

Parameter	Description	
Retraction	No	No retraction before swiveling
	Tool direction, inc.	Incremental retraction in tool direction The retraction path is entered into parameter ZR.
	Tool direction, max.	Maximum retraction in tool direction
	Z	Retraction in the direction of machine axis Z.
	Z XY	Retract towards the machine axis Z and then in the direction X, Y
ZR	Retraction path - (only for incremental retraction in the tool direction)	
Selection	Preferred direction of rotation for two alternatives • +: Larger angle of the axis on the scale of the swivel head / swivel table • -: Smaller angle of the axis on the scale of the swivel head / swivel table	
Tool	Tool tip position when swiveling	
		Correct The position of the tool tip is maintained during swiveling.
		No tracking The position of the tool tip changes during swiveling.

4.6.10 Defining the measurement function selection

The measurement versions "Set edge", "Align edge", "Right-angled corner", "1 hole" and "1 circular spigot" are listed in the "Measure workpiece zero" in the associated vertical softkey bar.

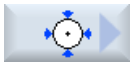
You have the option of replacing these by softkeys with other measurement versions.

"Set edge" softkey

The "Set edge" softkey cannot be assigned the softkey of another measurement version.

Procedure

1. The "Measure workpiece zero" function is selected.



2. Press the softkey that you wish to assign to a new measurement version, e.g. "1 circular spigot".

   The "1 Circular Spigot" window opens.

3. Open the list of measurement versions, select the desired measurement version using the <Cursor down> and the <Input> keys.

- OR -

Using the <Select> key, in the drop down list box, select the desired measurement version, e.g. "Align plane".

The "Align plane" window opens.

4. Enter the required parameter in order to make the measurement as usual.

   - OR -

Press the "Back" softkey.

The selected softkey is assigned the new measurement version, in this case, "Align plane".

4.6.11 Logging measurement results for the workpiece zero

When measuring the workpiece zero, you have the option to output the values that have been determined to a log.

The following data are determined and logged:

- Date/time
- Log name with path
- Measuring version
- Input values
- Correction target
- Setpoints, measured values and differences

You have the option to output the log as a text file (*.txt) or in a tabular format (*.csv).

Note

Processing the measurement results

The tabular format is a format that can be imported by Excel (or other spreadsheet programs). This allows the measurement result logs to be statistically processed.

Note

Logging active

The measurement results can only be entered into a log once the measurement has been fully completed.

Procedure

1. You are in the "JOG" mode and have pressed the "Workpiece zero" softkey.
   The "Measurement log" softkey cannot be used.
2. Select the required measurement version and measured the workpiece zero as usual.
   The measured values are displayed once the measurement has been completed.

3. Press the "Measurement log" softkey to save the measurement data as log.
   The "Measurement log" softkey becomes inactive again.

See also

Settings for the measurement result log (Page 151)

4.6.12 Calibrating the electronic workpiece probe

4.6.12.1 Calibration of length and radius or diameter

When the electronic probes are attached to the spindle, clamping tolerances usually occur. This can lead to measurement errors.

In addition, you need to determine the trigger points of the probe relative to the spindle center (trigger points).

Therefore, you need to calibrate the electronic probe. The radius or diameter is calibrated in a setting ring (calibration ring) or a hole, the length is calibrated on a surface. The diameter of the setting ring and the dimension of the surface in the Z direction (for G17) must be precisely known and this is entered into the corresponding entry field when calibrating the probe. The diameter of the workpiece probe ball and its length 1 must be stored in the tool list.

Procedure

1. Load the workpiece probe into the spindle.
2. Enter the approximate length and radius or diameter in the tool data.

Calibration of radius / diameter

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

4. Move the workpiece probe into the hole and position it at the approximate center of the hole.

5. Press the "Workpiece zero" and "Probe calibration" softkeys.
   The window "Calibration: Probe" is opened.

6. Press the "Radius" or "Diameter" softkey.

- OR -

Note

The "Diameter" softkey must be set up by the machine manufacturer.

7. In ∅, enter the calibration bore corresponding to the diameter.

8. Press the <CYCLE START> key.
   Calibration starts.
   When calibrating the radius, the exact hole center point is determined first. Then the four trigger points on the inside wall of the hole are approached.
   This procedure is carried out automatically twice: First with 180° (to the starting position of the working spindle) and then in its starting position.

Calibration of length

9. Move the workpiece probe over the reference surface.
10. Press the "Meas. workp." and "Calibrate probe" softkeys.
    The "Calibration: Probe" window opens.

11. Press the "Length" softkey.
12. Specify reference point Z0 of the surface, e.g. of the workpiece or the machine table.
    The length of the workpiece probe is determined.
13. In the "Adjust tool length" selection box, choose "no" to save the determined length difference in the calibration data set.
    - OR -
    Choose "yes" to calculate the length difference into the probe tool data.

Note

Please observe the information provided by the machine manufacturer.

14. Press the <CYCLE START> key.
    Calibration starts.

Note

User-specific defaults

- "Diameter, reference piece"
  For the entry field "Diameter, reference piece", fixed values can be separately entered at parameters for each probe number (calibration data set number). If these parameters are assigned, the values saved there are displayed in the entry field "Diameter of the reference piece"; however, they can no longer be changed there.
- "Height, reference piece"
  For the entry field "Height, reference piece", fixed values can be separately entered at parameters for each probe number (calibration data set number). If these parameters are assigned, the values saved there are displayed in the entry field "Height of the reference piece"; however, they can no longer be changed there.

Note

Please observe the information provided by the machine manufacturer.

4.6.12.2 Calibrate on sphere

When the electronic probes are attached to the spindle, clamping tolerances usually occur. This can lead to measurement errors.

Using this measuring method, a workpiece probe can be calibrated at any position in space. This has a special meaning in conjunction with swivel functions and transformations.

Procedure

1. Load the workpiece probe into the spindle.
2. Enter the diameter and length approximately in the tool data.
3. Select the "JOG" mode in the "Machine" operating area.

4. Move the workpiece probe approximately over the center of the sphere.
5. Press the "Meas. workp." and "Calibrate probe" softkeys.
   The window "Calibration: Probe" is opened.

6. Press the "Comp. to sphere" softkey.
   The "Calibration: Probe on sphere" window opens.
7. In the "Calib. in infeed axis" selection box, choose "yes" if you want to calibrate the probe in the plane and infeed axis.
   - OR -
   Choose "no" if you want to calibrate the probe on a reference surface.
8. If you choose "yes", enter the upper edge of the calibration sphere in "ZS" if you want to calibrate the infeed axis.
   The length of the workpiece probe is determined.
9. In the "Adjust tool length" selection box, choose "no" to save the determined length difference in the calibration data set.
   - OR -
   Choose "yes" to calculate the length difference into the probe tool data.
   Note
   Please observe the information provided by the machine manufacturer.
10. Enter the sphere diameter in "Ø" and the probing angle in "α0".

11. Press the <CYCLE START> key.
    Calibration starts.
    All reference data are determined in three passes round the reference sphere and stored in the selected data set.

4.7 Settings for the measurement result log

Make the following settings in the "Settings for measurement log" window:

- Log format
  - Text format
    The log in the text format is based on the display of the measurement results on the screen.
  - Tabular format
    When selecting the tabular format, the measurement results are saved so that the data can be imported into a spreadsheet program (e.g. Microsoft Excel). This allows the measurement result logs to be statistically processed.
- Log data
  - new
    The log of the actual measurement is created under the specified name. Existing logs with the same name are overwritten.
  - attach
    The log created is attached to the previous log.
- Where the log is saved
  The log created is saved in a specified directory.

Procedure

1. Select the "Machine" operating area.

2. Press the <JOG> key.

3. Press the menu forward key and the "Settings" softkey.

4. Press the "Measurement log" softkey.
   The "Settings for measurement log" window is opened.

5. Position the cursor to the log format field and select the required entry.

6. Position the cursor to the log data field and select the required entry.

7. Position the cursor to the log archive field and press the softkey "Select directory".
8. Navigate to the desired directory for the log archive.

9. Press the "OK" softkey and enter the name for the log file.

See also

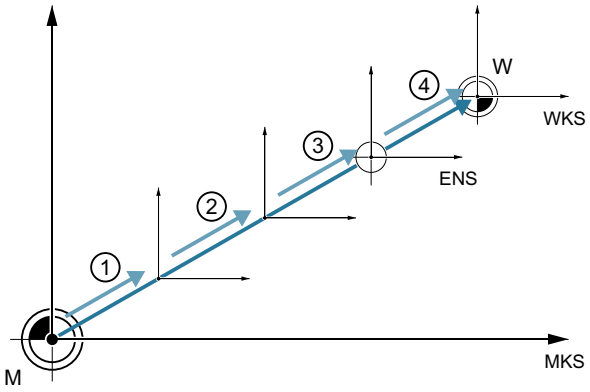
Logging measurement results for the workpiece zero (Page 146)
Logging tool measurement results (Page 118)

4.8 Zero offsets

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

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

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



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

Figure 4-1 Work offsets

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

Base offset

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

Coarse and fine offsets

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

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

Note

Deselect fine offset

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

See also

Actual value window (Page 43)

4.8.1 Display active zero offset

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

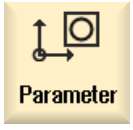
- Zero offsets, for which active offsets are included, or for which values are entered.
- Settable zero offsets
- Total zero offset

This window is generally used only for monitoring.

The availability of the offsets depends on the setting.

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

Procedure



1. Select the "Parameter" operating area.

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

Note

Further details on zero offsets

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

4.8.2 Displaying the zero offset "overview"

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

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

This window is generally used only for monitoring.

Display of active work offsets

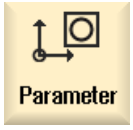
Work offsets	
MCS actual value	Display of the actual value in the Machine Coordinate System.
Kin. trans. workpiece	Displays the additional work offsets programmed with $P_TRAFRAME_P.
Kin. trans. tool	Displays the additional work offsets programmed with $P_TRAFRAME_T.
DRF	Displays the handwheel axis offset.
$AA_OFF overlay	Displays the overlaid movement programmed with $AA_OFF.
Basic reference	Displays the additional work offsets programmed with $P_SETFRAME. Access to the system offsets is protected via a keyswitch.
External WO frame	Displays the additional work offsets programmed with $P_EXTFRAME.
Total base WO	Displays all effective basis offsets.
G500	Displays the work offsets activated with G54 - G599. Under certain circumstances, you can change the data using "Set WO", i.e. you can correct a zero point that has been set.
Tool reference	Displays the additional work offsets programmed with $P_TOOLFRAME.
Workpiece reference	Displays the additional work offsets programmed with $P_WPFRAME.
Transformer reference	Displays the additional work offsets programmed with $P_TRAFRAME.
Programmed WO	Displays the additional work offsets programmed with $P_PFRAME.

Work offsets	
Cycle reference	Displays the additional work offsets programmed with $P_CYCFRAME.
Total WO	Displays the active work offset, resulting from the total of all work offsets.
T:	Displays the active tool.
WCS actual value	Displays the actual value in the Workpiece Coordinate System.

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

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

Procedure



1. Select the "Parameter" operating area.

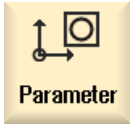
2 Press the "Work offset" and "Overview" softkeys.
The "Work Offsets - Overview" window opens.

4.8.3 Displaying and editing base zero offset

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

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

Procedure



1. Select the "Parameter" operating area.

2. Press the "Zero offset" softkey.

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

Note

Activate base offsets

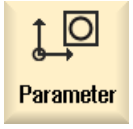
The offsets specified here are immediately active.

4.8.4 Displaying and editing settable zero offset

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

Rotation, scaling and mirroring are displayed.

Procedure



1. Select the "Parameter" operating area.

2. Press the "Work offset" softkey.

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

Note

Activate settable zero offsets

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

4.8.5 Displaying and editing details of the zero offsets

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

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

- Coarse and fine offsets
- Rotation
- Scaling
- Mirroring

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

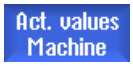
Note

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

Tool details

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

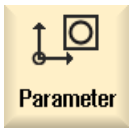
- TC
- Adapter dimension
- Length / length wear
- EC setup correction
- SC sum correction
- Total length
- Radius / radius wear



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

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

Procedure



1. Select the "Parameter" operating area.

2. Press the "Zero offset" softkey.

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

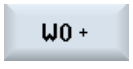
4. Place the cursor on the desired zero offset to view its details.

5. Press the "Details" softkey.

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

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

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



...

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

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

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

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

Press the "Back" softkey to close the window.

4.8.6 Deleting a zero offset

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

Procedure

1. Select the "Parameter" operating area.

2. Press the "Work offset" softkey.

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

...

4. Press the "Details" softkey.
5. Position the cursor on the work offset you would like to delete.

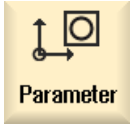
6. Press the "Clear offset" softkey.

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

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

4.8.7 Measuring the workpiece zero

Procedure



1. Select the "Parameters" operating area and press the "Zero offset" softkey.

2. Press the "G54...G599" softkey and select the zero offset in which the zero point is to be saved.

3. Press the "Meas. workp." softkey.

You change to the "Set Edge" window in the "JOG" mode.

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

...

5. Select the measuring direction (+ or -) you want to approach the workpiece in.

   The measuring direction cannot be selected for Z0.

6. In X0, Y0, or Z0, specify the setpoint position of the workpiece edge you are approaching.

   Traverse the tool up to the workpiece edge and press the "Set ZO" softkey to measure the workpiece zero.

4.9 Monitoring axis and spindle data

4.9.1 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.

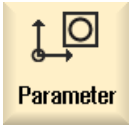
In this way, you are able to restrict the traversing range of the axes in addition to the limit switches.

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.

4.9.2 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

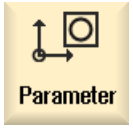
In field "Spindle speed limitation at G96", the programmed spindle speed limitation at constant cutting speed is displayed together with the permanently active limitations.

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.

4.10 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.

4.11 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.

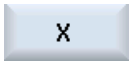
Procedure

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).

4.12 MDA

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

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

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

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

4.12.1 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.

4.12.2 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.

4.12.3 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

See also

Program control (Page 216)

4.12.4 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.

Execution in manual mode

5

5.1 General

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

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

5.2 Selecting a tool and spindle

5.2.1 T, S, M windows

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

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

Note

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

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

Parameter

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

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

Note

Spindle positioning

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

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

Additional parameters for milling/turning machines

For milling/turning machines, additional parameters are displayed to align the turning tools:

- if a turning tool was selected in field "T".
  - or -
- if the T field is empty, and a turning tool is currently active.

Parameter	Meaning		Unit
TC	Name of the swivel data record		
β	Angle of the tool to the axis of rotation		degrees
	←	Zero degrees	
	↓	90 degrees	
	Input value	The required angle can be freely entered	
Hirth gearing	◇	Round of β to the next Hirth gearing	
	^	Round of β to the next Hirth gearing	
	v	Round of β to the next Hirth gearing	
Tool	Tool tip position when swiveling		
		Tracking The position of the tool tip is maintained during swiveling.	
		No tracking The position of the tool tip is not maintained during swiveling.	
γ	Angle of rotation of the tool around itself		degrees
αC	Alignment of the plane of rotation in the pole position		degrees

5.2.2 Selecting a tool

Procedure

1. Select the "JOG" operating mode.

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

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

   - OR -

   Press the "Select tool" softkey.

   The tool selection window opens.

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

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

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

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

6. Press the <CYCLE START> key.

   The tool is loaded into the spindle.

5.2.3 Starting and stopping a spindle manually

Procedure

1. Select the "JOG" operating mode.

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

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

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

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

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

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

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

Note

Changing the spindle speed

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

5.2.4 Position spindle

Procedure

1. Select the "JOG" operating mode.

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

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

5. Press the <CYCLE START> key.

   The spindle is moved to the desired position.

Note

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

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

5.3 Traversing axes

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

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

Set the default feedrate

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

5.3.1 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.

5.3.2 Traversing axes by a variable increment

Procedure

1. Select the "Machine" operating area.

2. Press the <JOG> key.

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

5. Press the <Inc VAR> key.
6. Select the axis to be traversed.

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

5.4 Positioning axes

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

The feedrate / rapid traverse override is active during traversing.

Procedure

1. If required, select a tool.
2.

Select the "JOG" operating mode.
3.

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

   - OR -

   Press the "Rapid traverse" softkey.

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

Press the <CYCLE START> key.

   The axis is traversed to the specified target position.

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

5.5 Swiveling

Manual swivel in the JOG mode provides functions that make it far easier to setup, measure, and machine workpieces with swiveled surfaces.

If you want to create or correct an inclined position, the required rotations of the workpiece coordinate system around the geometry axes (X, Y, Z) are automatically converted into suitable positions of the machine kinematics.

Alternatively, you can program the swivel axes of the machine "directly" and generate a matching workpiece coordinate system for those swivel axis positions. After swiveling, the tool axis (for G17 Z) is always perpendicular to the working plane (for G17 XY).

The swiveled coordinates are maintained in the Reset status and after Power On, if the machine manufacturer has correspondingly set the machine data. With these settings, after a program interrupt, e.g. as a result of a retraction in the +Z direction, you can retract from an inclined hole.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Important parameters

- **TC - name of swivel data record**
  Here you can select the swivel data record.
- **Retraction**
  Before swiveling the axes you can move the tool to a safe retraction position. The retraction methods available to you are defined in the "Retraction position" parameter during set-up of the swivel data record.
  "Retraction" corresponds to Parameter _FR of CYCLE800.

Machine manufacturer

Please refer to the machine manufacturer's specifications. Due to additional swivel data records (e.g. for tool change) additional retraction versions can be enabled.

⚠ WARNING

Retraction position

Select a retraction position so that no collision can occur between the tool and workpiece when swiveling.

- **Swivel plane**
  You can start the swivel plane as "new" or "additive" to a swivel plane that is already active.

  Note

  "Additive" swivel plane

  With "additive" swivel plane, values should only be added to the swivel data set that is already active.

- **Swivel mode**
  Swiveling can be axis by axis or direct.
  - Axis-by-axis swiveling is based on the coordinate system of the workpiece (X, Y, Z). The coordinate axis sequence can be selected freely. Rotations are applied in the selected sequence. The rotation of the two rotary axes (A, B or C) is calculated from this.
  - For direct swiveling, the required positions of the rotary axes are specified. A suitable new coordinate system is calculated based on those values. The tool axis is aligned in the Z direction. You can derive the resulting direction of the X and Y axis by traversing the axes.

    Note

    The positive direction of each rotation for the different swivel methods is shown in the help displays.

- **Direction**
  "Direction" corresponds to the parameter _DIR of CYCLE800.
  For swivel systems with 2 rotary axes, a particular plane can be reached in two different ways. You can choose between these two different positions in the "Direction" parameter. The +/- corresponds to the larger or smaller value of a rotary axis. This may affect the working area. When the swivel data record is set up, the entries in the "Direction" parameter determine for which rotary axis you can select each of the two settings.
  If one of the two positions cannot be reached for mechanical reasons, the alternative position is automatically selected irrespective of the setting of the "Direction" parameter.

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

- **Correcting tool**
  "Tool" corresponds to the Parameter _ST=1x (correct tool tip) of CYCLE800.
  To avoid collisions, you can use the 5-axis transformation (software option) to retain the position of the tool tip during swiveling.
  When machine manufacturer commissions the function "Swivel Manually", "Track Tool" must be enabled.

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

- **Zero plane**
  The zero plane corresponds to the tool plane (G17, G18, G19) including the active zero offset (G500, G54, ...). Rotations of the active zero offset and the rotary axes are taken into account for manual swiveling.
  The "Swivel Manually" function only writes rotations either in the workpiece reference ($_WPFRAME) or in the active zero offset.
  You can use the "Swivel Manually" function not only for machining, but also for setting-up.
  - You can bring the machine into the initial position using the "Basic setting" softkey and the <CYCLE START> key. If the actual zero offset does not include a rotation, then the rotary axes of the swivel data record are moved to zero. The tool is located vertically to the machining plane.
    If you want to use the actual swiveled plane as the reference plane for setting up your workpiece, you must define this plane as the zero plane.
  - With "Set zero plane" the actual swivel plane in the active zero offset is stored as the zero plane. As a result, the rotations in the active zero offset are overwritten.
  - With "Delete zero plane", the rotations in the active zero offset are set to zero.

  Note

  The overall coordinate system does not change with "Set zero plane" or "Delete zero plane".

Machine manufacturer

Basic setting of the machine kinematics for "Swivel Manually" and "5-axis transformation".

Please refer to the machine manufacturer's specifications.

Procedure

1. Select the "Machine" operating area.

2. Press the <JOG> key

3. Press the "Swivel" softkey.

4. Enter the desired value for the parameter and press the <CYCLE START> key.
   The "Swivel" cycle is started.

Initial setting

5. Press the "Basic setting" softkey and the <CYCLE START> key to move the machine into the initial position.

   If the actual zero offset does not include a rotation, then the rotary axes of the swivel data record are moved to zero. The tool is located vertically to the machining plane.

   This is done, for example, to swivel the coordinate system back to its original orientation.

Set zero plane

6. Press the "Set zero plane" softkey to set the actual swivel plane to the new zero plane.

Delete 0-level

7. Press the "Delete zero plane" softkey to delete the actual swivel plane.

Parameter	Description		Unit
TC	Name of the swivel data record 0: Remove the swivel head, deselect the swivel data record No entry: No change to the set swivel data record		
Retraction	No	No retraction before swiveling	
	ink	Incremental retraction in tool direction The retraction path is entered into parameter ZR.	
	max	Maximum retraction in tool direction	
	Z	Retraction in the direction of machine axis Z	
	Z XY	Retract towards the machine axis Z and then in the direction X, Y	
ZR	Retraction path - (only for incremental retraction in the tool direction)		
Swivel plane	• New: New swivel plane • Additive: Additive swivel plane		
Swivel mode	• Axis by axis: Rotate coordinate system axis-by-axis • Direct: Directly position rotary axes Positions the rotary axes of the active swivel data record Angle of rotation in the plane around the tool axes		
Z	Angle of rotation in the plane (only in the "direct" swivel mode)		degrees
Axis sequence	Sequence of the axes which are rotated around: XYZ, XZY, YXZ, YZX, ZXY, ZYX		
X	Rotation around X		Degrees
Y	Rotation around Y		Degrees
Z	Rotation around Z		Degrees
Name of rotary axis 1	Axis angle for swivel, direct		Degrees
Name of rotary axis 2	Axis angle for swivel, direct		Degrees
Direction	Preferred direction of rotation for two alternatives (swiveling axis-by-axis)		
	+	Larger angle of the axis on the scale of the swivel head / swivel table	
	–	Smaller angle of the axis on the scale of the swivel head / swivel table	

Parameter	Description		Unit
Tool	Tool tip position when swiveling		
		Tracking The position of the tool tip is maintained during swiveling.	
		No tracking The position of the tool tip changes during swiveling.	

5.6 Manual retraction

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

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

The tool and/or the workpiece remain undamaged.

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

Note

Tapping

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

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

The machine OEM sets up the "Retract" function.

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

Procedure

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

3. Select the JOG operating mode.

4. Press the Menu forward key.

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

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

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

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

5.7 Simple face milling of the workpiece

You can use this cycle to face mill any workpiece. A rectangular surface is always machined.

Selecting the machining direction

In the "Direction" field, using the SELECT key, select the desired machining direction:

- Same direction of machining
- Alternating direction of machining

Selecting limits

You can select the limits using the appropriate softkeys:

Left

Bottom

Top

Right

Retraction plane / safety clearance

The retraction plane and safety clearance are set using the machine data $SCS_MAJOG_SAFETY_CLEARANCE or $SCS_MAJOG_RELEASE_PLANE.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Direction of spindle rotation

If the "ShopTurn/ShopMill" option is activated, the direction of spindle rotation is taken from the tool parameters entered in the tool list.

If the "ShopTurn/ShopMill" option is not set, select the direction of spindle rotation in the input screen.

See also

Face milling (CYCLE61) (Page 434)

Precondition

To carry out simple stock removal of a workpiece in manual mode, a measured tool must be in the machining position.

Procedure

1. Select the "Machine" operating area.

2. Press the <JOG> key.

3. Press the <Face milling> softkey.

4. Press the relevant softkey to specify the lateral limitations of the workpiece.

5. Select the machining type (e.g. roughing) in the "Machining" field.

6. Select the machining direction in the "Direction" field.

7. Enter all other parameters in the input screen.

8. Press the "OK" softkey.
   The parameter screen is closed.

9. Press the <CYCLE START> key.
   The face milling cycle is started.
   You can return to the parameter screen at any time to check and correct the inputs.

Note

You cannot use the "Repos" function while face milling.

Parameter	Description	Unit
T	Tool name	
D	Cutting edge number	
F	Feedrate	mm/min mm/rev
S / V	Spindle speed or constant cutting rate	rpm m/min
Spindle M function	Direction of spindle rotation (only when ShopMill is not active) • •	
Machining 	The following machining operations can be selected: • ∇ (roughing) • ∇∇∇ (finishing)	

Parameter	Description	Unit
Direction 	Same direction of machining • • Alternating direction of machining • •	
X0	Corner point 1 of surface in X direction (abs. or inc.)	mm
Y0	Corner point 1 of surface in Y direction (abs. or inc.)	mm
Z0	Height of blank (abs. or inc.)	mm
X1	Corner point 2 of surface in X direction (abs. or inc.)	mm
Y1	Corner point 2 of surface in Y direction (abs. or inc.)	mm
Z1	Height of finished part (abs. or inc.)	mm
DXY	Max. infeed in the XY plane (dependent on milling cutter diameter) Alternatively, you can specify the plane infeed as a %, as a ratio → plane infeed (mm) to milling cutter diameter (mm).	mm %
DZ	Max. infeed in Z direction - (only for roughing)	mm
UZ	Finishing allowance, depth	mm

Note

The same finishing allowance must be entered for both roughing and finishing. The finishing allowance is used to position the tool for retraction.

See also

Tool, offset value, feed and spindle speed (T, D, F, S, V) (Page 347)

5.8 Simple workpiece machining operations with milling/turning machines

5.8.1 Simple workpiece face milling (milling/turning machine)

You can use this cycle to face mill any workpiece. A rectangular surface is always machined.

Selecting the machining direction

In the "Direction" field, using the SELECT key, select the desired machining direction:

- Same direction of machining
- Alternating direction of machining

Selecting limits

You can select the limits using the appropriate softkeys:

Left

Bottom

Top

Right

Retraction plane / safety clearance

The retraction plane and safety clearance are set using the machine data $SCS_MAJOG_SAFETY_CLEARANCE or $SCS_MAJOG_RELEASE_PLANE.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Direction of spindle rotation

If the "ShopTurn/ShopMill" option is activated, the direction of spindle rotation is taken from the tool parameters entered in the tool list.

If the "ShopTurn/ShopMill" option is not set, select the direction of spindle rotation in the input screen.

Requirement

To carry out simple face milling of a workpiece in the manual mode, a measured tool must be in the machining position.

Procedure

1. Select the "Machine" operating area.

2. Press the <JOG> key.

3 Press the "Machining" and "Face milling" softkeys.

4. Press the relevant softkey to specify the lateral limitations of the workpiece.

5. Select the machining type (e.g. roughing) in the "Machining" field.

6. Select the machining direction in the "Direction" field.

7. Enter all other parameters in the input screen.

8. Press the "OK" softkey.
   The parameter screen is closed.

9. Press the <CYCLE START> key.
   The face milling cycle is started.
   You can return to the parameter screen at any time to check and correct the inputs.

Note

You cannot use the "Repos" function while face milling.

Parameter	Description	Unit
T	Tool name	
D	Cutting edge number	
F	Feedrate	mm/min mm/rev
S / V	Spindle speed or constant cutting rate	rpm m/min
Spindle M function	Direction of spindle rotation (only when ShopMill is not active) • •	

Parameter	Description	Unit
Machining	The following machining operations can be selected: • ∇ (roughing) • ∇∇∇ (finishing)	
Direction	Same direction of machining • • Alternating direction of machining • •	
X0	Corner point 1 of surface in X direction (abs. or inc.)	mm
Y0	Corner point 1 of surface in Y direction (abs. or inc.)	mm
Z0	Height of blank (abs. or inc.)	mm
X1	Corner point 2 of surface in X direction (abs. or inc.)	mm
Y1	Corner point 2 of surface in Y direction (abs. or inc.)	mm
Z1	Height of finished part (abs. or inc.)	mm
DXY	Max. infeed in the XY plane (dependent on milling cutter diameter) Alternatively, you can specify the plane infeed as a %, as a ratio → plane infeed (mm) to milling cutter diameter (mm).	mm %
DZ	Max. infeed in Z direction - (only for roughing)	mm
UZ	Finishing allowance, depth	mm

Note

The same finishing allowance must be entered for both roughing and finishing. The finishing allowance is used to position the tool for retraction.

5.8.2 Simple stock removal of workpiece (for milling/turning machine)

Some blanks do not have a smooth or even surface. For example, you can use the stock removal cycle to turn the face surface of the workpiece before machining actually takes place.

If you want to bore out a collet using the stock removal cycle, program an undercut (XF2) in the corner.

⚠ CAUTION

Risk of collision

The tool moves along a direct path to the starting point of the stock removal.

First move the tool to a safe position in order to avoid collisions during the approach.

Retraction plane / safety clearance

The retraction plane and safety clearance are set using the machine data $SCS_MAJOG_SAFETY_CLEARANCE or $SCS_MAJOG_RELEASE_PLANE.

Machine manufacturer

Please observe the information provided by the machine manufacturer.

Direction of spindle rotation

If the "ShopMill/ShopTurn" option is activated, the direction of spindle rotation is taken from the tool parameters entered in the tool list.

If the "ShopMill/ShopTurn" option is not set, select the direction of spindle rotation in the input screen.

Note

The "Repos" function cannot be used during simple stock removal.

Requirement

To carry out simple stock removal of a workpiece in manual mode, a measured tool must be in the machining position.

Procedure

1. Press the "Machine" operating area key

2. Press the <JOG> key.

3. Press the "Machining" and "Stock removal" softkeys.

4. Enter desired values for the parameters.

5. Press the "OK" softkey.

   The parameter screen is closed.

6. Press the <CYCLE START> key.

   The "Stock removal" cycle is started.

   You can return to the parameter screen form at any time to check and correct the inputs.

Parameters	Description		Unit
T	Tool name		
D	Cutting edge number		
TC	Name of the swivel data record		
β	Angle of the tool to the axis of rotation		degrees
	↓	β = 0°	
	←	β = 90°	
	Input value	The required angle can be freely entered	
	Hirth gearing		
	◇	Round of β to the next Hirth gearing	
	^	Round of β to the next Hirth gearing	
	v	Round of β to the next Hirth gearing	
Tool	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.		
γ	Angle of rotation of the tool around itself		degrees
αC	Alignment of the plane of rotation in the pole position		degrees
F	Feedrate		mm/rev
S / V	Spindle speed or constant cutting rate		rpm m/min
Spindle M function	Direction of spindle rotation (only when ShopMill is not active) • •		
Machining	• ∇ (roughing) • ∇∇∇ (finishing)		
Position	Machining position 		

Parameters	Description	Unit
Machining direction	• Face • Longitudinal	
X0	Reference point ∅ (abs)	mm
Z0	Reference point (abs)	mm
X1	End point X ∅ (abs) or end point X in relation to X0 (inc)	mm
Z1	End point Z (abs) or end point Z in relation to X0 (inc)	mm
FS1...FS3 or R1...R3	Chamfer width (FS1...FS3) or rounding radius (R1...R3)	mm
XF2	Undercut (alternative to FS2 or R2)	mm
D	Infeed depth (inc) – (for roughing only)	mm
UX	Final machining allowance in X direction (inc) – (for roughing only)	mm
UZ	Final machining allowance in Z direction (inc) – (for roughing only)	mm

5.9 Default settings for manual mode

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

Default settings

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

Procedure

1. Select the "Machine" operating area.

2. Press the <JOG> key.

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

Machining the workpiece

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 change-over to the "Machine" operating area occurs.

6.3 Testing a program

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.

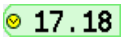
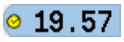
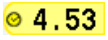
Display of current program

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

Setting for automatic mode (Page 281)

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 machining at a specific point

6.7.1 Use block search

If you only want to execute a particular section of a program on the machine, you do not need to start the program from the beginning. You can start the program from a specified program block.

Applications

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

Determining a search target

- User-friendly search target definition (search positions)
  - Direct specification of the search target by positioning the cursor in the selected program (main program)
    Note:
    In a block search, ensure that the correct tool is in the working position before starting program execution.
    ShopMill has automated this operation, i.e. any tool change that is required is performed automatically with this type of block search in ShopMill program steps.
    Please observe the information provided by the machine manufacturer.
  - 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 is selected that is in a subprogram. You can then change the program level up to the main program level and back to the level of the interruption point.

Note

If a rotary axis is traversed between a CYCLE800 call and the target block, problems will occur when calculating the block search.

If possible, place the search target in front of such a rotary axis motion.

- 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. After a search destination has been found, it is possible to continue cascading any number of times.

Note

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

Requirements

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

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

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.

More information

More information on the block search can be found in Basic Functions in the function manual.

See also

Selecting a program (Page 199)

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 Entering the search target via search pointer

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

Requirement

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

Screen form

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

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

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

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

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

Procedure

1. Press the "Block search" softkey.

2. Press the "Search pointer" softkey.

3. Enter the full path of the program as well as the subprograms, if required, in the input fields.

4. Press the "Start search" softkey.

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

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

5. Press the <CYCLE START> key twice.

   Processing is continued from the defined location.

Note

Interruption point

You can load the interruption point in search pointer mode.

6.7.6 Parameters for block search in the search pointer

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

6.7.7 Block search mode

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

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

Search variants

Block search mode	Meaning
With calculation - without approach	Is used to be able to approach a target position in any situation (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. **Note**: In a ShopMill program, the search is only performed on G code-blocks.
With calculation - skip extcall	This is used to speed-up a search with calculation when using EXTCALL programs: EXTCALL programs are not taken into account. **Notice**: Important information, e.g. modal functions, which are located in the EXTCALL program, are not taken into account. In this case, after the search target has been found, the program is not able to be executed. Such information should be programmed in the main program.
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.

Note

Search mode for ShopMill programs

- The search variant for the ShopMill machining step programs can be specified via MD 51024. This applies only to the ShopMill single-channel view.

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

More information

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

Procedure

1. Select the "Machine" operating area.

2. Press the <AUTO> key.

3. Press the "Block search" and "Block search mode" softkeys.
   The "Search Mode" window opens.

6.7.8 Block search for position pattern

It is possible performing a block search for the position pattern. You can define the number of the starting hole.

With ShopMill programs you can also define the technology with which you want to start.

Software option
You require the "ShopMill/ShopTurn" option for the block search for ShopMill machining step programs.

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

Procedure

The required ShopMill program or G code program is in the block display.

1. Press the "Block search" softkey.
2. Position the cursor to the position block.

3. Press the "Start search" softkey.
   The "Block search" window opens.

Define technology (only with ShopMill programs)

4. All of the technologies used in the position pattern are listed.
   Select the required technology and press the "OK" softkey
   The selected technology is displayed in the "Search" window.

Specify starting hole

5. Enter the number of the starting hole and press the "OK" softkey.
   Program processing starts with the specified technology (only for ShopMill programs) at the specified starting hole, and continues to all additional positions of this position pattern and all of the following position patterns.

   Note

   If you have hidden certain positions, then only the displayed positions are applicable for the starting hole number.

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.

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

With the editor, you are able to render, supplement, or change part programs.

Note

Maximum block length

The maximum block length is 512 characters.

Calling the editor

- The editor is started via the "Program correction" softkey in the "Machine" operating area. You can directly change the program by pressing the <INSERT> key.
- The editor is called via the "Open" softkey as well as with the <INPUT> or <Cursor right> key in the "Program manager" operating area.
- The editor opens in the "Program" operating area with the last executed part program, if this was not explicitly exited via the "Close" softkey.

  Note

  - Please note that the changes to programs saved in the NC memory take immediate effect.
  - If you are editing on a local drive or external drives, you can also exit the editor without saving, depending on the setting. Programs in the NC memory are always automatically saved.
  - Exit the program correction mode using the "Close" softkey to return to the "Program manager" operating area.

See also

Editor settings (Page 229)

Opening and closing the program (Page 778)

Correcting a program (Page 205)

Generating a G code program (Page 317)

6.10.1 Searching in programs

You can use the search function to quickly arrive at points where you would like to make changes, e.g. in very large programs.

Various search options are available that enable selective searching.

Search options

- Whole words
  Activate this option and enter a search term if you want to search for texts/terms that are present as words in precisely this form.
  If, for example, you enter the search term "Finishing tool", only single "Finishing tool" terms are displayed. Word combinations such as "Finishing tool_10" are not found.
- Exact expression
  Activate this option if you wish to search for terms with characters, which can also be used as place holders for other characters, e.g. "?" and "*".

Note

Search with place holders

When searching for specific program locations, you have the option of using place holders:

- "*": Replaces any character string
- "?": Replaces any character

Precondition

The desired program is opened in the editor.

Procedure

1. Press the "Search" softkey.
   A new vertical softkey bar appears.
   The "Search" window opens at the same time.
2. Enter the desired search term in the "Text" field.
3. Select "Whole words" if you want to search for whole words only.
   - OR -
   Activate the "Exact expression" checkbox if, for example, you want to search for place holders ("*", "?") in program lines.

4. Position the cursor in the "Direction" field and choose the search direction (forward, backward) with the <SELECT> key.

5. Press the "OK" softkey to start the search.
   If the text you are searching for is found, the corresponding line is highlighted.

6. Press the "Continue search" softkey if the text located during the search does not correspond to the point you are looking for.
   - OR -

Press the "Cancel" softkey when you want to cancel the search.

Further search options

Softkey	Function
Go to start	The cursor is set to the first character in the program.
Go to end	The cursor is set to the last character in the program.

6.10.2 Replacing program text

You can find and replace text in one step.

Precondition

The desired program is opened in the editor.

Procedure

1. Press the "Search" softkey.
   A new vertical softkey bar appears.

2. Press the "Find and replace" softkey.
   The "Find and Replace" window appears.
3. In the "Text" field, enter the term you are looking for and in the "Replace with" field, enter the text you would like to insert automatically during the search.

4. Position the cursor in the "Direction" field and choose the search direction (forward, backward) with the <SELECT> key.

5. Press the "OK" softkey to start the search.
   If the text you are searching for is found, the corresponding line is highlighted.

6. Press the "Replace" softkey to replace the text.

   - OR -

   Press the "Replace all" softkey to replace all text in the file that corresponds to the search term.

   - OR -

   Press the "Continue search" softkey if the text located during the search should not be replaced.

   - OR -

   Press the "Cancel" softkey when you want to cancel the search.

Note

Replacing texts

- Read-only lines (;*RO*)
  If hits are found, the texts are not replaced.
- Contour lines (;*GP*)
  If hits are found, the texts are replaced as long as the lines are not read-only.
- Hidden lines (;*HD*)
  If hidden lines are displayed in the editor and hits are found, the texts are replaced as long as the lines are not read-only. Hidden lines that are not displayed, are not replaced.

See also

Editor settings (Page 229)

6.10.3 Copying/pasting/deleting a program block

In the editor, you edit both basic G code as well as program steps such as cycles, blocks and subprogram calls.

Inserting program blocks

The editor responds depending on what type of program block you insert.

- If you insert a G code, then the program block is directly inserted where the write mark is located.
- If you insert a program step, then the program block is always inserted at the next block, independent of the position of the write mark within the actual line. This is necessary as a cycle call always requires its own line.
  This behavior is in all applications, irrespective of whether the program step is inserted with a screen form using "Accept" or "Insert" is used as editor function.

  Note

  Cutout program step and reinsert

  - If you cut out a program step at a specific location and you then directly reinsert it again, the sequence changes.
  - Press the shortcut (key combination) <CTRL> + <Z> to undo what you have cut out.

Precondition

The program is opened in the editor.

Procedure

1. Press the "Mark" softkey.

   - OR -

   Press the <SELECT> key.

2. Select the desired program blocks with the cursor or mouse.

3. Press the "Copy" softkey in order to copy the selection to the buffer memory.

4. Place the cursor on the desired insertion point in the program and press the "Paste" softkey.

   The content of the buffer memory is pasted.

   - OR -

   Press the "Cut" softkey to delete the selected program blocks and to copy them into the buffer memory.

   Note: When editing a program, you cannot copy or cut more than 1024 lines. While a program that is not on the NC is opened (progress display less than 100%), you cannot copy or cut more than 10 lines or insert more than 1024 characters.

Numbering the program blocks

If you have selected the "Automatic numbering" option for the editor, then the newly added program blocks are allocated a block number (N number).

The following rules apply:

- When creating a new program, the first line is allocated the "first block number".
- If, up until now, the program had no N number, then the program block inserted is allocated the starting block number defined in the "First block number" input field.
- If N numbers already exist before and after the insertion point of a new program block, then the N number before the insertion point is incremented by 1.
- If there are no N numbers before or after the insertion point, then the maximum N number in the program is increased by the "increment" defined in the settings.

Note:

After exiting the program, you have the option of renumbering the program blocks.

Note

The buffer memory contents are retained even after the editor is closed, enabling you to paste the contents in another program.

Note

Copy/cut current line

To copy and cut the current line where the cursor is positioned, it is not necessary to mark or select it. You have the option of making the "Cut" softkey only operable for marked program sections via editor settings.

See also

Opening additional programs (Page 228)

6.10.4 Renumbering a program

You can modify the block numbering of programs opened in the editor at a later point in time.

Precondition

The program is opened in the editor.

Procedure

1. Press the ">>" softkey.
   A new vertical softkey bar appears.

2. Press the "Renumber" softkey.
   The "Renumbering" window appears.
3. Enter the values for the first block number and the increment to be used for numbering.

4. Press the "OK" softkey.
   The program is renumbered.

Note

- If you only want to renumber a section, before the function call, select the program blocks whose block numbering you want to edit.
- When you enter a value of "0" for the increment size, then all of the existing block numbers are deleted from the program and/or from the selected range.

6.10.5 Creating a program block

In order to structure programs to achieve a higher degree of transparency, you have the option of combining several blocks (G code and/or ShopMill machining steps) to form program blocks.

Program blocks can be created in two stages. This means that additional blocks can be formed within a particular block.

You then have the option of opening and closing these blocks depending on your requirement.

Structuring programs

- Before generating the actual program, generate a program frame using empty blocks.
- By forming blocks, structure existing G code or ShopMill programs.

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 "Form block" softkey.
   The "Form New Block" window opens.

5. Enter a designation for the block and 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 using the mouse or with the cursor keys:

- <Cursor right> opens the block at the location where the cursor is
- <Cursor left> closes the block if the cursor is located 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.

6.10.6 Opening additional programs

You have the option of viewing and editing several programs simultaneously in the editor.

For instance, you can copy program blocks or machining steps of a program and paste them into another program.

Opening several programs

You have the option of opening up to ten program blocks.

1. In the program manager, select the programs that you wish to open and view in the multiple editor and then press on the "Open" softkey.
   The editor is opened and the first two programs are displayed.

2. Press the <NEXT WINDOW> key to change to the next opened program.

3. Press the "Close" softkey to close the actual program.

Note

Pasting program blocks

JobShop machining steps cannot be copied into a G code program.

Precondition

You have opened a program in the editor.

Procedure

1. Press the ">>" and "Open additional program" softkeys.

The "Select Additional Program" window is opened.

2. Select the program or programs that you wish to display in addition to the already opened program.

3. Press the "OK" softkey.

The editor opens and displays both programs next to each another.

See also

Copying/pasting/deleting a program block (Page 224)

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".
Show hidden lines	• Yes: Hidden lines marked with "*HD" (hidden) will be displayed. • No: Lines marked with ";*HD*" will not be displayed. **Note**: Only visible program lines are taken into account with the "Search" and "Search and Replace" functions.
Display block end as symbol	The "LF" (line feed) symbol ¶ is displayed at the block end.
Line break	• Yes: Long lines are broken and wrapped around. • No: If the program includes long lines, then a horizontal scrollbar is displayed. You can move the section of the screen horizontally to the end of the line.

Setting	Meaning
Line break also in cycle calls	• Yes: If the line of a cycle call becomes too long, then it is displayed over several lines. • No: The cycle call is truncated. The field is only visible if "Yes" is entered under "Line break".
Visible programs	• 1 - 10 Select how many programs can be displayed next to one another in the editor. • Auto Specifies that the number of programs entered in a job list or up to ten selected programs will be displayed next to each other.
Width per program w. focus	Here, you enter the width of the program that has the input focus in the editor as a percentage of the window width.
Save automatically	• Yes: The changes are saved automatically when you change to another operating area. • No: You are prompted to save when changing to another operating area. Save or reject the changes with the "Yes" and "No" softkeys. **Note**: Only for local and external drives.
Only cut after marking	• Yes: Parts of programs can only be cutout when program lines have been selected, i.e. the "Cutout" softkey only then is active. • No: The program line, in which the cursor is positioned, can be cut out without having to select it.
Determine machining times	Defines which program runtimes are determined in the simulation or in automatic mode: • Off Program runtimes are not determined. • Block-by-block: The runtimes are determined for each program block. • Non-modal: The runtimes are determined at the NC block level. **Note**: You also have the option of displaying the cumulative times for blocks. Please observe the information provided by the machine manufacturer. After the simulation or after executing the program, the required machining times are displayed in the editor.
Save machining times	Specifies how the machining times determined are processed. • Yes A subdirectory with the name "GEN_DATA.WPD" is created in the directory of the part program. There, the machining times determined are saved in an ini file together with the name of the program. The machining times are displayed again when the program or job list are reloaded. • No The machining times that have been determined are only displayed in the editor.

Setting	Meaning
Record tools	Defines whether the tool data is recorded. • Yes Recording takes place during processing. The data is stored in a TTD file (Tool Time Data). The TTD file is located in the directory of the associated part program. • No The tool data is not recorded.
Display cycles as machining step	• Yes: The cycle calls in the G code programs are displayed as plain text. • No: The cycle calls in the G code programs are displayed in the NC syntax.
Highlight selected G code commands	Defines the display of G code commands. • No All G code commands are displayed in the standard color. • Yes Selected G code commands or keywords are highlighted in color. Define the rules for the color assignment in the sleditorwidget.ini configuration file. **Note:** Please observe the information provided by the machine manufacturer. **Note** This setting also has an effect on the current block display.
Font size	Defines the font size for the editor and the display of the program sequence. • auto If you open a second program, then the smaller font size is automatically used. • normal (16) - character height in pixels Standard font size that is displayed with the appropriate screen resolution. • small (14) - character height in pixels More content is displayed in the editor. **Note** This setting also has an effect on the current block display.

Note

All entries that you make here are effective immediately.

Requirement

You have opened a program in the editor.

Procedure

1. Select the "Program" operating area.

2. Press the "Edit" softkey.

3. Press the ">>" and "Settings" softkeys.
   The "Settings" window opens.

4. Make the required changes.

5. Press the "Del. machining times" softkey if you wish to delete the machining times.
   The machining times that have been determined are deleted from the editor as well as from the actual block display. If the machining times are saved to an ini file, then this file is also deleted.

6. Press the "OK" softkey to confirm the settings.

See also

Replacing program text (Page 223)

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 as well as drilling positions directly in G-code and ShopMill programs.

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 following elements are read by the DXF reader:

- "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.

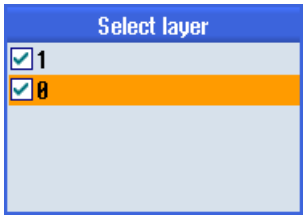
Requirement

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

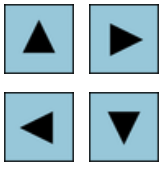
1. Press the "Details" and "Magnifying glass" softkeys.
   A magnifying glass in the shape of a rectangular frame appears.

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 in the editor

6.11.3.1 General procedure

- Creating and opening a G-code or ShopMill program
- Calling "Contour milling" cycles and creating a "New contour"
  - OR -
- Calling "Positions / position pattern" from the "Drill" cycle
- Importing a DXF file
- Select the contour or drilling positions in the DXF file or CAD drawing and click "OK" to accept the cycle
- Add program record with "Accept" to the G-code or ShopMill program

6.11.3.2 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.

6.11.3.3 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 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.

6.11.3.5 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".

6.11.3.6 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.

6.11.3.7 Transferring the drilling positions

Procedure

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

2. Press the "Drilling" softkey.

3. Press the "Positions" softkey.

4. Press the "Arbitrary positions" softkey.
   The "Positions" input window opens.

- OR -

Press the "Line" softkey.
The "Row of positions" input window opens.

- OR -

Press the "Grid" softkey.
The "Position grid" input window opens.

- OR

Press the "Frame" softkey.
The "Position frame" input window opens.

- OR -

Press the "Circle" softkey.
The "Position circle" input window opens.

- OR -

Press the "Pitch circle" softkey.
The "Position pitch circle" input window opens.

Selecting the drilling positions

Requirement

You have selected a position pattern.

Procedure

Opening a DXF file

1. Press the "Import from DXF" softkey.
2. Position the cursor on the desired DXF file in the storage directory.

You can use the search function to search directly for a DXF file in comprehensive folders and directories.

3. Press the "OK" softkey.
   The CAD drawing opens and the cursor takes the form of a cross.

Cleaning a file

4. Prior to selecting the drilling positions, you can select a layer and clean the file.

Specifying a reference point

5. If required, specify a zero point.

Specify clearance(s) (position pattern "Row"/"Arbitrary positions" and "Circle"/"Pitch circle"

6. Press the "Select element" softkey repeatedly to navigate through the orange selection symbol to the desired drilling position.

7. Press the "Accept element" softkey to transfer the position.
   Repeat steps 6 and 7 to specify other drilling positions for "Arbitrary positions".

Specify clearance with second clearance (for position pattern "Frame", "Grid")

Select element

8. Once the reference point has been specified, press the "Select element" softkey repeatedly to navigate to the desired drilling position in order to specify the clearance.

Accept element

9. Press the "Accept element" softkey.

   A rectangular cross-hair appears.

Select element

10. Press the "Select element" repeatedly to navigate to the desired drilling position on the displayed line.

    To determine the second clearance, drilling positions must be located on the line.

Accept element

11. Press the "Accept element" softkey.

    A frame or grid is displayed.

Size (position pattern "Row", "Frame", "Grid")

Select element

12. Once the reference point and clearances have been specified, press the "Select element" softkey repeatedly.

    All expansions of the frame or the grid are displayed.

Accept element

13. Press the "Accept element" softkey to confirm the selected frame or grid.

    If all elements for the position row or position frame and position grid are valid, the drilling positions are displayed with blue points.

Circle direction (circle and pitch circle)

Select element

Once the reference point and clearance have been specified, press the "Select element" softkey repeatedly.

The circle is shown in the possible orientations.

Accept element

Press the "Select element" softkey to confirm the selected circle or pitch circle.

If all elements of the circle or pitch circle are valid, the drilling positions are displayed with blue points.

Resetting actions

Revert

Undo can be used to reset the last actions.

Transfer drilling positions to the cycle and the program

OK

4. Press the "OK" softkey in order to accept the position values.

   You return to the associated parameter screen form.

Accept

Press the "Accept" softkey to transfer the drilling positions 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.

6.11.3.8 Accepting contours

Calling up the cycle

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

2. Press the "Mill 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 -

Element end	Press the "Element end" softkey to place the contour end at the end of the element.
	- OR -
Cursor	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

OK	Press the "OK" softkey. The selected contour is transferred to the contour input screen of the editor.
Accept contour	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 Importing shapes from CAD programs

Using the "Import from CAD" function, you have the option of transferring models, which can be processed using standard cycles, directly from a 3D model into the part program. This function is available for milling and drilling in the G code program as well as in the ShopMill and ShopTurn program.

Software option

You require software option "3D Job Shop" to be able to use this function.

NX server

You must access an NX server to evaluate the CAD data.

You configure access to the NX server in the window "Set up NX server", which you call using softkey "Set up NX server" in the "Setup" operating area.

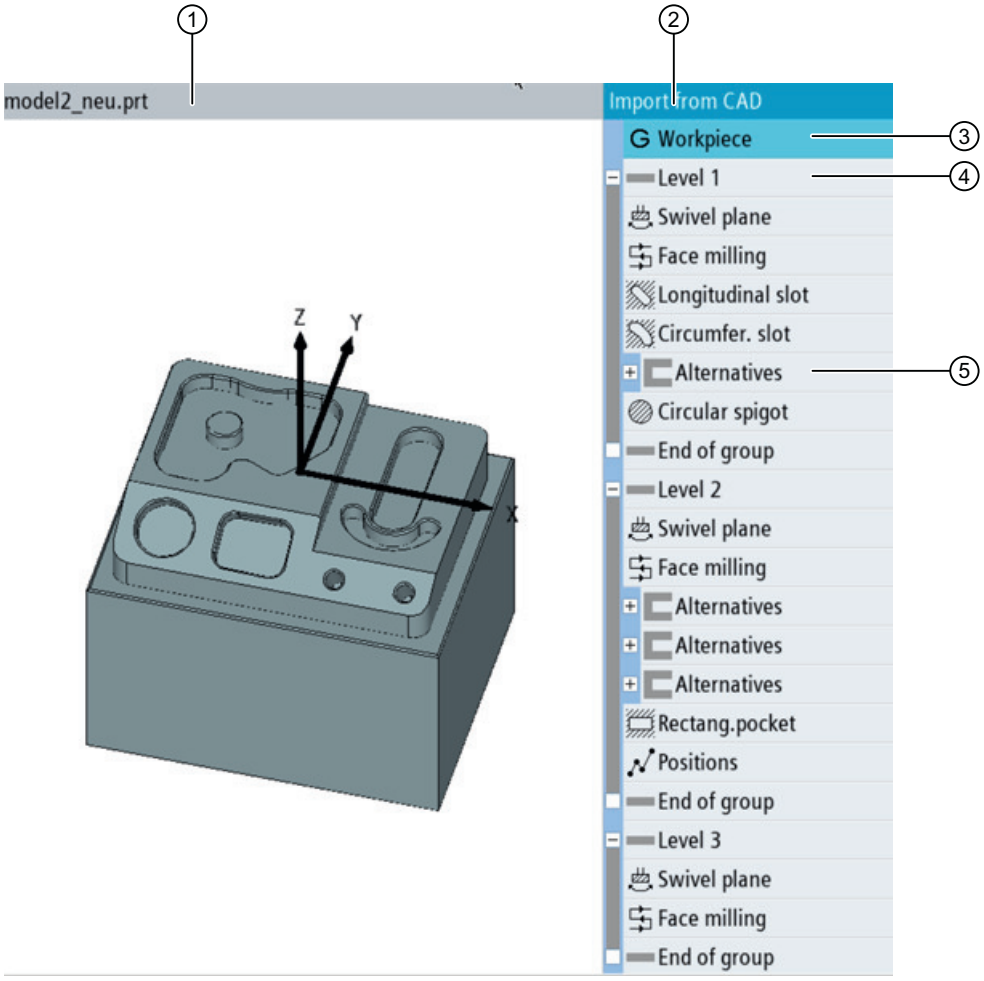
3D files

The following CAD file formats are supported:

- Step part and assembly files (*.stp, *.step)
- NX parts and assemblies (*.prt)

Screen structure: Overview

The 3D model of the workpiece and the suggested machining steps are displayed in the "Import from CAD" screen form.



① 3D model
② List of machining steps
③ "Workpiece" machining step
④ "Plane" block
⑤ "Alternatives" block

Every machining step list starts with a "Workpiece" step. This step contains the zero point of the complete workpiece.

Machining steps that can be used to create the same model are listed in the "Alternatives" block.

On swiveling machines, the models are additionally distributed across planes. A "Swivel plane" step is inserted for each plane.

The following models are recognized when evaluating the CAD file in the NX server:

- Hole
- Female thread

- Male thread
- Face milling
- Rectangular pocket
- Circular pocket
- Rectangular spigot
- Circular spigot
- Longitudinal groove
- Circumferential groove
- Contour pocket

6.12.1 Reading in CAD data into an editor and processing

6.12.1.1 General procedure

Proceed as follows to accept data from a CAD file into a G code program or a ShopMill/ShopTurn program using function "Import from CAD":

- Create and open a program
- Define a blank
- Import a 3D file
- Define the workpiece zero
- Define a zero point plane (only for swiveling machines)
- Select a machining step from the alternatives, add any missing parameters and insert into the program using "Apply".
- To accept additional machining steps, reopen the list of machining steps
- Make supplements/additions to the program (e.g. tool change, positioning blocks)

6.12.1.2 Import from CAD

To import models, select a CAD file.

In the machining steps, the selected file is uploaded to the NX server and analyzed. The results are converted into potential machining steps in SINUMERIK Operate, and displayed in the list of machining steps.

Requirement

- The connection to the NX server has been set up.
- The CAD file can be accessed via the Program Manager.
- A G code or ShopMill/ShopTurn program has been created, and opened in the program editor.

Procedure

1. Press the "Various" and "Import from CAD" softkeys.
   Window "Import from CAD" opens.

2. Navigate to the archive location, and position the cursor on the CAD file required.

3. Click "OK" to confirm the selection.
   The list of machining steps opens.
   The 3D model is displayed in the left-hand part of the window.
   The recognized models as well as alternative machining steps are listed in the right-hand part.

6.12.1.3 Defining reference points

Overview of zero points

You have the option of defining reference points so that the positions of the models are correctly calculated in the coordinate system. You save the workpiece zero in the "Workpiece" machining step. If swiveling is set up at your machine, you have the option of defining a dedicated zero point for each plane.

Position of the zero point

You have the option of defining the zero point using elements of the 3D model:

- Unchanged (for workpiece zero point)
  The selection "unchanged" takes the workpiece zero point from the 3D model.
- The same as the workpiece (for plane zero point)
  With selection "the same as the workpiece", the zero point of the plane remains unchanged, i.e. as defined in the workpiece.
  This selection is suitable for machining operations performed at the base plane.
- Point of intersection, 3 surfaces
- Point of intersection, 2 edges
- Based on the model
  The zero point is defined at the center point of the selected model.

You define the coordinates of the selected point by entering parameters X0, Y0 and Z0.

Alignment of the coordinate system

The Z axis is aligned vertically with respect to a freely selectable surface. The direction of the X axis is defined using an edge. The direction of the X axis can be adapted using parameter "Offset X". For instance, the direction of the X axis can be inverted by applying an offset of 180°.

Defining the workpiece zero

You have the option of defining a new reference point if the zero point from the 3D model deviates from the workpiece zero.

Procedure

1. The "Workpiece" machining step is selected.
2. Press the "Workpiece zero point" softkey.
   The "Workpiece zero" window opens.
3. Select how the zero point is defined.
4. Select the required references in field "Selection".
   - OR -
   Select the required references in the 3D model.
   The selected elements are highlighted in the 3D model and displayed in field "Selection".
5. To align the coordinate origin, select parameter "Direction Z", "Direction X" and "Offset X".
6. Confirm your selection with "Accept".

Defining the zero point of the plane

For each machining plane, you can define your own reference point.

Procedure

1. The "Swivel plane" machining step is selected.
2. Press softkey "Work offset plane".
   The "Work offset plane" window opens.
3. Select how the point of rotation for swiveling is defined.
4. In the "Select" fields, select the required surfaces or edges.
   - OR -
   Select the required references in the 3D model.
   The selected elements are highlighted in the 3D model and displayed in field "Selection".
5. If the point of rotation cannot be directly selected, enter an additional offset of the coordinates in parameters XD, YD and ZD.
6. Select how the zero point of the plane is defined, and select the required references.
7. To align the coordinate origin, select parameter "Direction X" and "Offset X".
   The alignment of the Z axis is defined by the current plane.
8. Confirm your selection with "Accept".

6.12.1.4 Accepting the machining steps

To accept the suggested data in the program, select the appropriate machining step in the list of machining steps, and if necessary, add additional parameters. The machining step is written to the program and the list of machining steps is closed.

Requirement

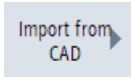
The list of machining steps is open.

Procedure

1. In the 3D model, select the required element, using either touch or mouse.
   The element is highlighted in the 3D model. The cursor jumps to the associated machining step in the list.
   - OR -
   Using the cursor keys, navigate in the list to the required machining step.
   The element is highlighted in color in the 3D model.
2. If several, alternative machining steps exist for the selected element, then these steps are listed in the "Alternatives" block.
   In this case, select the required technological function.

3. Press the "Accept" softkey.
   The parameter screen form of the selected function opens. The values that came from the CAD data are accepted. All other parameters are preassigned with default values.
4. Check the parameters, and if necessary, change the default values.

5. Confirm the data with "OK".
   The program view opens. The newly created machining step is marked.



6. If you wish to accept additional machining steps, then press the "Import from CAD" softkey.
   The list of machining steps opens again.

Note

Swivel after tool change

If swiveling is set-up at your machine, then step "Swivel plane" must be inserted after each tool change.

6.13 Display and edit user variables

6.13.1 Overview

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.

6.13.2 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

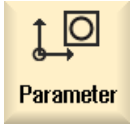
The number of global R parameters is defined in a machine data element.

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.

6.13.3 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

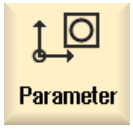
The number of channel-specific R parameters is defined in a machine data element.

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.

6.13.4 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)

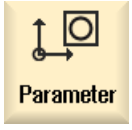
Example

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.

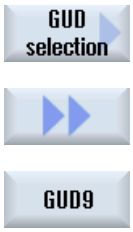
3. Press the "Global GUD" softkeys.

   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.

6.13.5 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

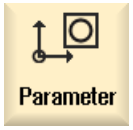
A channel-specific GUD variable is defined with the following:

- 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.

6.13.6 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.

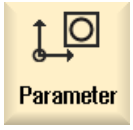
6.13.7 Displaying program user data (PUD)

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.

6.13.8 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.

6.14 Displaying G Functions and Auxiliary Functions

6.14.1 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.

6.14.2 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

6.14.3 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.

6.14.4 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.

6.15 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.

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 Mold making view

6.16.1 Overview

For large mold making programs such as those provided by CAD/CAM systems, you have the option to display the machining paths by using a fast view. This provides you with a fast overview of the program, and you have the possibility of correcting it.

Machine manufacturer

The mold making view has possibly been hidden.

Please observe the information provided by the machine manufacturer.

Checking the program

You can check the following:

- Does the programmed workpiece have the correct shape?
- Are there large traversing errors?
- Which program block hasn't been correctly programmed?
- How is the approach and retraction realized?

NC blocks that can be interpreted

The following NC blocks are supported for the mold making view:

- Types
  - Lines
    G0, G1 with X Y Z
  - Circles
    G2, G3 with center point I, J, K or radius CR, depending on the working plane G17, G18, G19, CIP with circular point I1, J1, K1 or radius CR
  - Absolute data AC and incremental data IC are possible
  - For G2, G3 and different radii at the start and end, an Archimedes spiral is used
- Orientation
  - Rotary axis programming with ORIAXES or ORIVECT using ABC for G0, G1, G2, G3, CIP, POLY
  - Orientation vector programming with ORIVECT using A3, B3, C3 for G0, G1, G2, G3, CIP
  - Rotary axes can be specified using DC
- G codes
  - Working planes (for circle definition G2, G3): G17 G18 G19
  - Incremental or absolute data: G90 G91

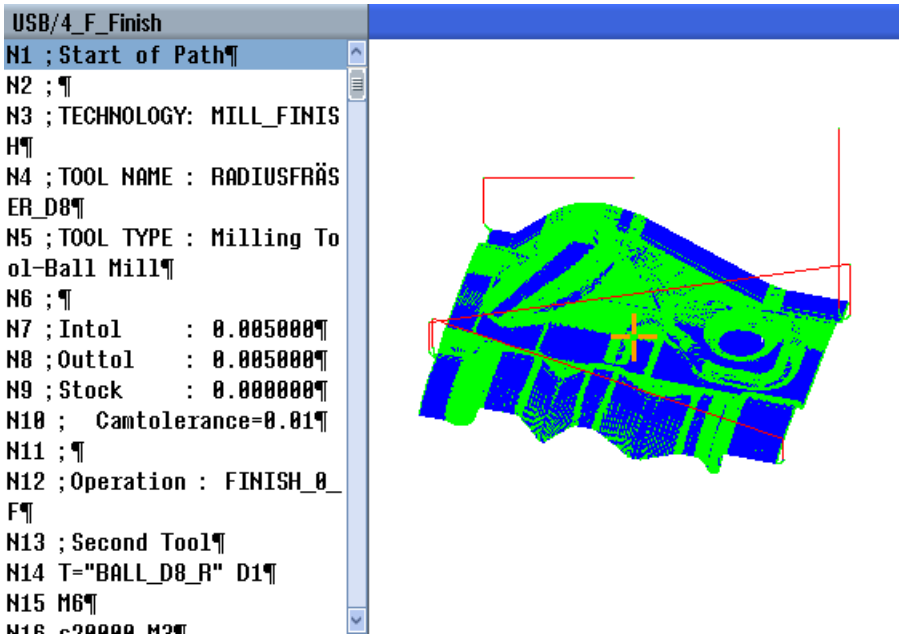
The following NC blocks are **not** supported for the mold making view:

- Helix programming
- Rational polynomials
- Other G codes or language commands

All NC blocks that cannot be interpreted are simply overread.

Simultaneous view of the program and mold making view

You have the option of displaying the mold making view next to the program blocks in the editor.



You can navigate back and forth between the NC blocks listed on the left and the associated points in the mold making view.

- On the left in the editor, if you place the cursor on an NC block with position data, then this NC block is marked in the graphic view.
- If you select a point on the right in the mold making view using the mouse, then conversely you mark the corresponding NC block on the left-hand side of the editor. This is how you jump directly to a position in the program in order to edit a program block for example.

Switch between the program window and the mold making view

Press the <NEXT WINDOW> key if you wish to toggle between the program window and the mold making view.

Changing and adapting the mold making view

Like simulation and simultaneous recording, you have the option of changing and adapting the mold making view in order to achieve the optimum view.

- Increasing or reducing the size of the graphic
- Moving the graphic
- Rotating the graphic
- Changing the section

See also

Specifically jump to the program block (Page 274)

Enlarging or reducing the graphical representation (Page 276)

Modifying the viewport (Page 277)

6.16.2 Starting the mold making view

Procedure

1. Select the "Program manager" operating area.
2. Select the program that you would like to display in the mold making view.

3. Press the "Open" softkey.

   The program is opened in the editor.

4. Press the ">>" and "Mold making view" softkeys.

   The editor splits up into two areas.

The G code blocks are displayed in the left half of the editor.

The workpiece is displayed in the mold making view on the right-hand side of the editor. All of the points and paths programmed in the part program are represented.

6.16.3 Adapting the mold making view

You can adapt the graphic in various ways to better assess the workpiece in the mold making view.

Preconditions

- The required program is opened in the mold making view.
- The "Graphic" softkey is active.

Procedure

1. Press the softkey "Hide G1/G2/G3" if you want to conceal the machining paths.

- OR -

2. Press the softkey "Hide G0" if you want to deactivate the approach and retraction paths.

- OR -

Press softkey "Hide points" to conceal all the points in the graphic.

Note:

You have the option of simultaneously hiding G1/G2/G3 and G0 lines.

In this case softkey "Hide points" is deactivated.

- OR -

Press the softkeys ">>" and "Vectors" to display all orientation vectors.

Note:

This softkey can only be operated if vectors are programmed.

- OR -

Press the softkeys ">>" and "Surface" to calculate the surface area of the workpiece.

- OR -

Press the softkeys ">>" and "Curvature".

The "Curvature" input window opens.

Enter the desired minimum and maximum value and press "OK" to confirm the entry and to highlight the curvature changes in color.

6.16.4 Specifically jump to the program block

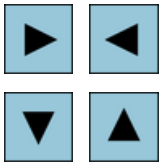
If you notice anything peculiar in the graphic or identify an error, then from this location, you can directly jump to the program block involved where you can edit the program.

Requirements

- The required program is opened in the mold making view.
- The "Graphic" softkey is active.

Procedure

1. Press the ">>" and "Select point" softkeys.
   Cross-hairs for selecting a point are shown in the diagram.



2. Using the cursor keys, move the cross-hairs to the desired position in the graphic.

3. Press the "Select NC block" softkey.
   The cursor jumps to the associated program block in the editor.

6.16.5 Searching for program blocks

Using the "Search" function, you can go specifically to program blocks where you can edit programs. You can find and replace text in one step.

Precondition

- The required program is opened in the mold making view.
- The "NC blocks" softkey is active.

Procedure

1. Press the "Search" softkey.
   A new vertical softkey bar appears.

See also

Searching in programs (Page 221)

Replacing program text (Page 223)

6.16.6 Changing the view

6.16.6.1 Enlarging or reducing the graphical representation

Precondition

- The mold making view has been started.
- The "Graphic" softkey is active.

Procedure

1. Press the <+> and <-> keys if you wish to enlarge or reduce the graphic display.
   The graphic display enlarged or reduced from the center.

...

- OR -

Press the "Details" and "Zoom +" softkeys if you wish to increase the size of the segment.

- OR -

Press the "Details" and "Zoom -" softkeys if you wish to decrease the size of the segment.

- OR -

Press the "Details" and "Auto zoom" softkeys if you wish to automatically adapt the segment to the size of the window.
The automatic scaling function "Fit to size" takes account of the largest expansion of the workpiece in the individual axes.

Note

Selected section

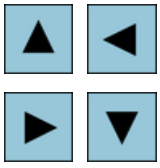
The selected sections and size changes are kept as long as the program is selected.

6.16.6.2 Moving and rotating the graphic

Precondition

- The mold making view has been started.
- The "Graphic" softkey is active.

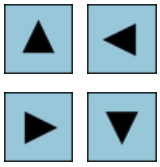
Procedure



1. Press one of the cursor keys to move the mold making view up, down, left or right.

- OR -

With the <SHIFT> key pressed, rotate the mold building view in the required direction using the cursor keys.



Note

Working with a mouse

Using the mouse, you have the option of rotating and shifting the mold making view.

- To do this, move the graphic with the left-hand mouse key pressed in order to reposition the mold making view.
- To do this, move the graphic with the right-hand mouse key pressed in order to rotate the mold making view.

6.16.6.3 Modifying the viewport

If you want to look at the details, you can shift and change the size of the mold building view section using a magnifying glass.

Using the magnifying glass, you can define your own segment and then increase or decrease its size.

Precondition

- The mold making view has been started.
- The "Graphic" softkey is active.

Procedure

1. Press the "Details" softkey.

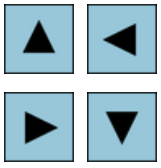
2. Press the "Zoom" softkey.
   A magnifying glass in the shape of a rectangular frame appears.

3. Press the "Magnify +" or <+> softkey to enlarge the frame.

   - OR -

   Press the "Magnify -" or <-> softkey to reduce the frame.

   - OR -



   Press one of the cursor keys to move the frame up, down, left or right.

4. Press the "Accept" softkey to accept the section.

6.17 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.

See also

Specifying the number of workpieces (Page 354)

6.18 Setting for automatic mode

Before machining a workpiece, you can test the program in order to identify programming errors at an early stage. Use the dry run feedrate for this purpose.

You have the option of additionally limiting the traversing speed so that when running-in a new program with rapid traverse, no undesirably high traversing speeds occur.

Dry run feedrate

If you selected "DRY run feedrate" under program control, then the value entered in "Dry run freedrate DRY" replaces the programmed feedrate when executing/machining.

Reduced rapid traverse

If you selected "RG0 reduced rapid traverse" under program control, then rapid traverse is reduced to the percentage value entered in "reduced rapid traverse RG0".

Displaying measurement results

Using an MMC command, measurement results can be displayed in a part program.

The following settings are possible:

- When it reaches the command, the control automatically jumps to the "Machine" operating area and the window with the measurement results is displayed
- The window with the measurement results is opened by pressing the "Measurement result" softkey

Recording machining times

To provide support when creating and optimizing a program, you have the option of displaying the machining times.

You define whether the time is determined while the workpiece is being machined (i.e. if the function is energized).

- Off
  Machining times are not determined when machining a workpiece. No machining times are determined.
- Non-modal
  The machining times are determined for each traversing block of a main program.
  Note: You also have the option of displaying the cumulative times for blocks.
  Please observe the information provided by the machine manufacturer.
- Block-by-block
  Machining times are determined for all blocks.

Note

Utilization of resources

The more machining times are displayed, the more resources are utilized.

More machining times are determined and saved with the non-modal setting than with the block-by-block setting.

Note

Please observe the information provided by the machine manufacturer.

Save machining times

You define how the machining times determined are processed.

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

Record tools

You define whether the tool data is recorded.

- Yes
  Recording takes place during processing. The data is stored in a TTD file (Tool Time Data). The TTD file is located in the directory of the associated part program.
- No
  The tool data is not recorded.

Procedure

1. Select the "Machine" operating area.

2. Press the <AUTO> key.

3. Press the menu forward key and the "Settings" softkey.
   The "Settings for Automatic Operation" window opens.

4. In "DRY run feedrate," enter the desired dry run speed.
5. Enter the desired percentage in the "Reduced rapid traverse RG0" field.
   RG0 has no effect if you do not change the specified amount of 100 %.

6. Select the required entry in the "Display measurement result" field:
   - "Automatic"
     The measurement result window opens automatically.
   - "manual"
     The measurement result window is to be opened by pressing the "Measurement result" softkey.

7. Select the desired entry in the fields "Record machining time", "Save machining times" and "Record tools".

Further information

Additional information on the display of measurement result display is provided in the Measuring Cycles Programming Manual.

Note

You have the option of changing the feed velocity during operation.

See also

Current block display (Page 47)

Simulating machining 7

7.1 Overview

During simulation, the current program is calculated in its entirety and the result displayed in graphic form. The result of programming is verified without traversing the machine axes. Incorrectly programmed machining steps are detected at an early stage and incorrect machining on the workpiece prevented.

Graphic display

The simulation represented on the screen uses the correct workpiece and tool proportions.

For simulation at milling machines, the workpiece is located, fixed in space. Only the tool moves, independent of the machine type.

Definition of a blank

The blank dimensions that are entered in the program editor are used for the workpiece.

The blank is clamped with reference to the coordinate system, which is valid at the time that the blank was defined. This means that before defining the blank in G code programs, the required output conditions must be established, e.g. by selecting a suitable work offset.

Programming a blank (example)

```
G54 G17 G90
CYCLE800(0,"TABLE", 100000,57,0,0,0,0,0,0,0,0,0,-1,100,1)
WORKPIECE(,,,"Box",112,0,-50,-80,00,155,100)
T="NC-SPOTDRILL_D16
```

Note

Blank offset for a changed work offset

The blank is always created in the work offset which is presently active.

If you select another work offset, then the coordinate system is converted, however, the display of the blank is not changed.

Note

Blank clamping

If your machine has various blank clamping options, enter the required clamping in the program header or in the blank screen.

Note also the machine manufacturer's instructions in this regard.

Display of the traversing paths

The traversing paths are displayed in color. Rapid traverse is red and the feedrate is green.

Depth display

The depth infeed is color-coded. The depth display indicates the depth level at which machining is currently taking place. "The deeper, the darker" applies for the depth display.

Machine references

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

In spite of this, unavoidable machine references are in the programming, such as for example, the tool change point in the machine, the retraction position when swiveling and the table components of a swivel kinematic. Depending on the current work offset - in the worst case - these machine references can mean that collisions are shown in the simulation that would not occur for a realistic work offset - or vice versa, collisions are not shown, which could occur for a realistic work offset.

Programmable frames

All frames and work offsets are taken into account in the simulation.

Note

Manually swiveled axes

Note that swivel movement in simulation and during simultaneous recording is also displayed when the axes are swiveled manually at the start.

Simulation display

You can choose one of the following types of display:

- Material removal simulation
  During simulation or simultaneous recording you can follow stock removal from the defined blank.
- Path display
  You have the option of including the display of the path. The programmed tool path is displayed.

  Note

  Tool display in the simulation and for simultaneous recording

  In order that workpiece simulation is also possible for tools that have either not been measured or have been incompletely entered, certain assumptions are made regarding the tool geometry.

  For instance, the length of a miller or drill is set to a value proportional to the tool radius so that cutting can be simulated.

Note

Thread turns not displayed

For thread and drill thread milling, the thread turns are not displayed in the simulation and for simultaneous recording.

Display variants

You can choose between three variants of graphical display:

- Simulation before machining of the workpiece
  Before machining the workpiece on the machine, you can perform a quick run-through in order to graphically display how the program will be executed.
- Simultaneous recording before machining of the workpiece
  Before machining the workpiece on the machine, you can graphically display how the program will be executed during the program test and dry run feedrate. The machine axes do not move if you have selected "no axis 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

The following views are available for all three variants:

- Top view
- 3D view
- Side views
- Turning view (milling/turning machine)
- Half-section (milling/turning machine)

Note

Simulation in half-section view

The "half-section" view in the simulation allows a more precise observation of the internal turning operations. This view was not conceived for the observation of milling operations. Display of the milling operations can lead to excessive simulation times.

Status display

The current axis coordinates, the override, the current tool with cutting edge, the current program block, the feedrate and the machining time are displayed.

In all views, a clock is displayed during graphical processing. The machining time is displayed in hours, minutes and seconds. It is approximately equal to the time that the program requires for processing including the tool change.

Software options

You require the "3D simulation of the finished part" option for the 3D view.

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

Determining the program runtime

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

Properties of simultaneous recording and simulation

Traversing paths

For the simulation, the displayed traversing paths are saved in a ring buffer. If this buffer is full, then the oldest traversing path is deleted with each new traversing path.

Optimum display

If simultaneous machining is stopped or has been completed, then the display is again converted into a high-resolution image. In some cases this is not possible. In this case, the following message is output: "High-resolution image cannot be generated".

Working zone limitation

No working zone limits and software limit switches are effective in the tool simulation.

Start position for simulation and simultaneous recording

During simulation, the start position is converted via the work offset to the workpiece coordinate system.

The simultaneous recording starts at the position at which the machine is currently located.

Restrictions

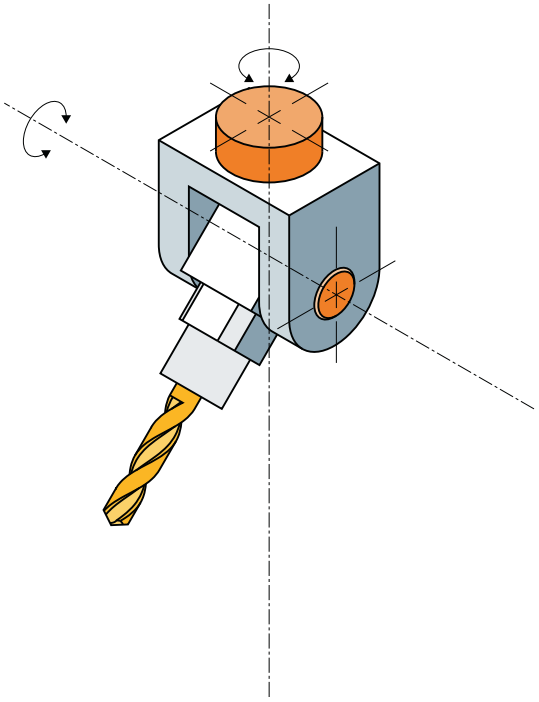
- Traori: 5-axis motion is linearly interpolated. More complex motion cannot be displayed.
- Referencing: G74 from a program run does not function.
- Alarm 15110 "REORG block not possible" is not displayed.
- Compile cycles are only partly supported.
- No PLC support.
- Axis containers are not supported.

Supplementary conditions

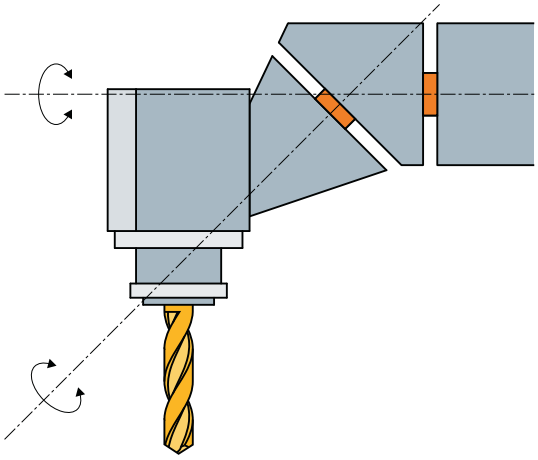
- All of the existing data records (tool carrier / TRAORI, TRACYL) are evaluated and must be correctly commissioned for correct simulation.
- The machine kinematics for TRAFOOF are not taken into consideration.
- Transformations with swiveled linear axis (TRAORI 64 - 69) as well as OEM transformations (TRAORI 4096 - 4098) are not supported.
- Changes to the tool carrier or transformation data only become effective after Power On.
- Transformation change and swivel data record change are supported. However, a real kinematic change is not supported, where a swivel head is physically changed.
- The simulation of mold making programs with extremely short block change times can take longer than machining, as the computation time distribution for this application is dimensioned in favor of the machining and to the detriment of simulation.

Examples

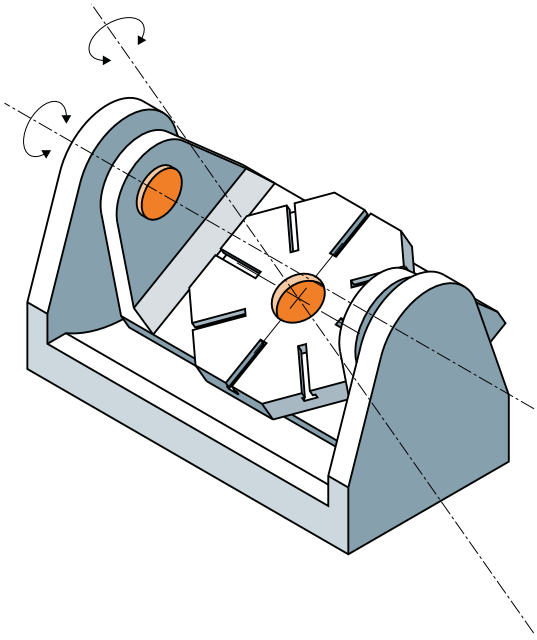
Several examples for machine types that are supported:



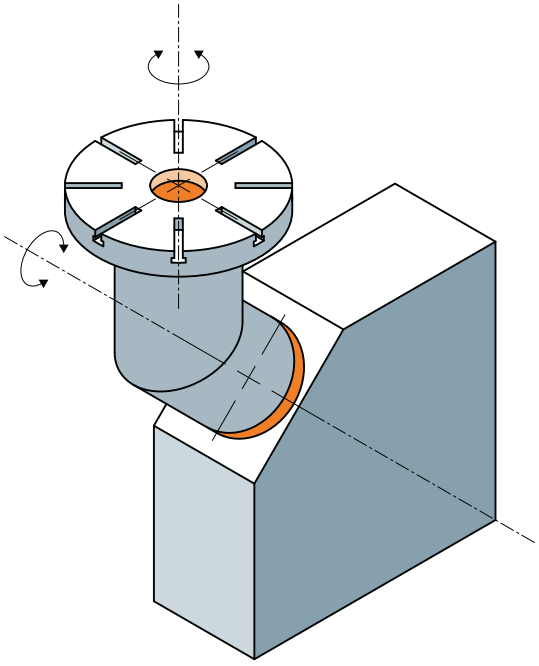
Swivel head 90°/90°



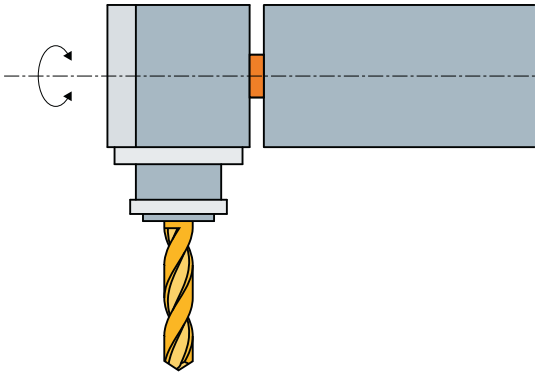
Swivel head 90°/45°

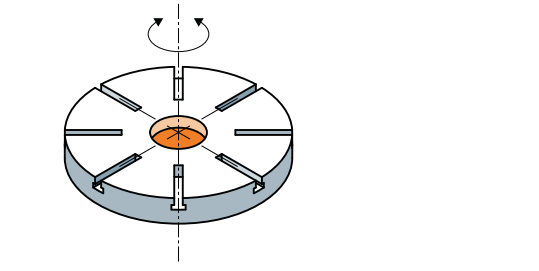


Swivel table 90°/90°

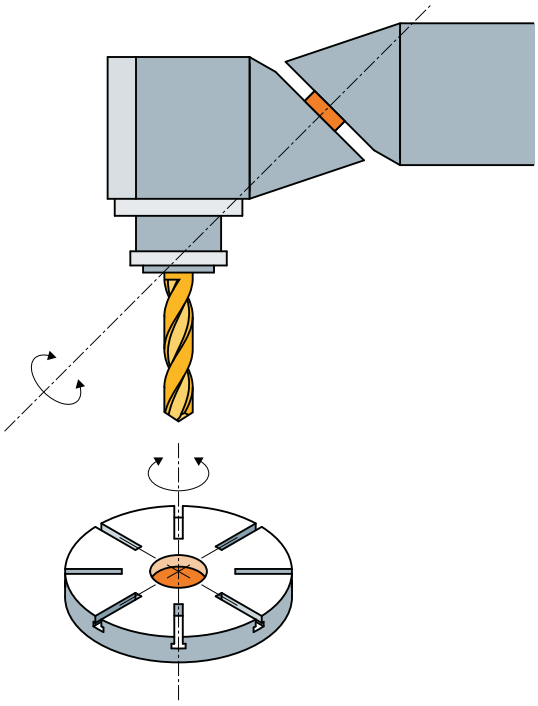


Swivel table 90°/45°





Swivel combination 90°/90°



Swivel combination 45°/90°

7.2 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 of 0 - 120%.

Procedure

1. Select the "Program Manager" operating area.

2. Select the storage location and position the cursor on the program to be simulated.

3. Press the <INPUT> or <Cursor right> key.

- OR -

Double-click the program.

The selected program is opened in the "Program" operating area.

4. Press the "Simulation" softkey.

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

5. Press the "Stop" softkey if you wish to stop the simulation.

- OR -

Press the "Reset" softkey to cancel the simulation.

6. Press the "Start" softkey to restart or continue the simulation.

Note

Operating area switchover

The simulation is exited if you switch into another operating area. If you restart the simulation, then this starts again at the beginning of the program.

7.3 Simultaneous recording before machining of the workpiece

Before machining the workpiece on the machine, you can graphically display the execution of the program on the screen to monitor the result of programming.

Software option

You require the option "Simultaneous recording (real-time simulation)" for simultaneous recording.

You can replace the programmed feedrate with a dry run feedrate to influence the speed of execution and select the program test to disable axis motion.

If you would like to view the current program blocks again instead of the graphical display, you can switch to the program view.

Procedure

1. Load a program in the "AUTO" mode.
2.

Press the "Prog. ctrl." softkey and activate the checkboxes "PRT no axis movement" and "DRY run feedrate".

   The program is executed without axis movement. The programmed feedrate is replaced by a dry run feedrate.
3.

Press the "Sim. rec." softkey.
4.

Press the <CYCLE START> key.

   The program execution is displayed graphically on the screen.
5.

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

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 on the screen.

Software option

You require the option "Simultaneous recording (real-time simulation)" for simultaneous recording.

Procedure

1. Load a program in the "AUTO" mode.

2. Press the "Sim. rec." softkey.

3. Press the <CYCLE START> key.

   The machining of the workpiece is started and graphically displayed on the screen.

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

Note

- If you switch on simultaneous recording after the blank information has already been processed in the program, only traversing paths and tool are displayed.
- If you switch off simultaneous recording during machining and then switch on the function again at a later time, the traversing paths generated in the meantime will not be displayed.

7.5 Different views of the workpiece

In the graphical display, you can choose between different views so that you constantly have the best view of the current workpiece machining, or in order to display details or the overall view of the finished workpiece.

The following views are available:

- Top view
- 3D view (with option)
- Side views
- Turn view (for milling/turning machine)
- Half cut view (for milling/turning machine)
- Machine space (with "collision avoidance" option)

Note

Simulation in half cut view

In the simulation, the "Half cut view" is used for more precise viewing of inside turning operations. This view is not designed for viewing milling operations. Displaying milling operations in this way can result in long simulation times.

7.5.1 Plan view

Display as a top view

1. Simultaneous recording or the simulation is started.
2. Press the "Top view" softkey.
   The workpiece is shown from above in the top view.

Changing the display

You can increase or decrease the size of the simulation graphic and move it, as well as change the segment.

7.5.2 3D view

Displaying the 3D view

1. Simultaneous recording or the simulation is started.
2. Press the "Other views" and "3D view" softkeys.

Software option

You require the option "3D simulation (finished part)" for the simulation.

Changing the display

You can increase or decrease the size of the simulation graphic, move it, turn it, or change the segment.

Displaying and moving cutting planes

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

See also

Defining cutting planes (Page 305)

7.5.3 Side view

Displaying further side views

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

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

- OR -

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

- OR -

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

- OR -

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

Changing the display

You can increase or decrease the size of the simulation graphic and move it, as well as change the segment.

7.5.4 Turning view

Display turning view (for milling/turning machine)

1. Start the simulation.
2. Press the "Further views" and "Turn view" softkeys.

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

Changing the display

You can increase or decrease the size of the simulation graphic and move it, as well as change the segment.

7.5.5 Half section

Display "half cut" view (for milling/turning machine)

1. Start the simulation.
2. Press the "Further views" and "Half cut view" softkeys.

The half cut view shows the workpiece cut in the Z-X plane.

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

Changing the display

You can increase or decrease the size of the simulation graphic and move it, as well as change the segment.

7.6 Editing the simulation display

7.6.1 Blank display

You have the option of replacing the blank defined in the program or to define a blank for programs in which a blank definition cannot be inserted.

Note

The unmachined part can only be entered if simulation or simultaneous recording is in the reset state.

Procedure

1. The simulation or the simultaneous recording is started.
2.
   Press the ">>" and "Blank" softkeys.
   The "Blank Input" windows opens and displays the pre-assigned values.

3. Enter the desired values for the dimensions.
4.

   Press the "Accept" softkey to confirm your entries. The newly defined workpiece is displayed.

7.6.2 Showing and hiding the tool path

The path display follows the programmed tool path of the selected program. The path is continuously updated as a function of the tool movement. The tool paths can be shown or hidden as required.

Procedure

1. The simulation or the simultaneous recording is started.
2.
   Press the ">>" softkey.
   The tool paths are displayed in the active view.
3.

   Press the softkey to hide the tool paths.
   The tool paths are still generated in the background and can be shown again by pressing the softkey again.
4.

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

7.7 Program control during the simulation

7.7.1 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%.

7.7.2 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.

7.8 Changing 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

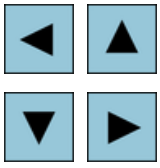
The selected sections and size changes are kept as long as the program is selected.

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.

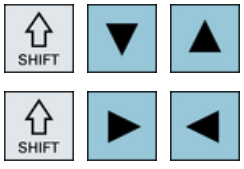
2. Press the "Rotate view" softkey.

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.

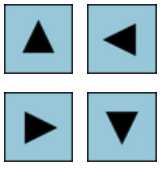
2. Press the "Magnifying glass" softkey.
   A magnifying glass in the shape of a rectangular frame appears.

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.

7.8.5 Defining cutting planes

In the 3D view, you have the option of "cutting" the workpiece and therefore displaying certain views in order to show hidden contours.

Precondition

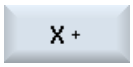
The simulation or the simultaneous recording is started.

Procedure

1. Press the "Details" softkey.

2. Press the "Cut" softkey.

The workpiece is displayed in the cut state.



3. Press the corresponding softkey to shift the cutting plane in the required direction.

...

7.9 Displaying simulation alarms

Alarms might occur during simulation. If an alarm occurs during a simulation run, a window opens in the operating window to display it.

The alarm overview contains the following information:

- Date and time
- Deletion criterion
  Specifies with which softkey the alarm is acknowledged
- Alarm number
- Alarm text

Precondition

Simulation is running and an alarm is active.

Procedure

1. Press the "Program control" and "Alarm" softkeys.
   The "Simulation Alarms" window is opened and a list of all pending alarms is displayed.

Press the "Acknowledge alarm" softkey to reset the simulation alarms indicated by the Reset or Cancel symbol.

The simulation can be continued.

- OR -

Press the "Simulation Power On" softkey to reset a simulation alarm indicated by the Power On symbol.

Generating a G code program

8

8.1 Graphical programming

Functions

The following functionality is available:

- Technology-oriented program step selection (cycles) using softkeys
- Input windows for parameter assignment with animated help screens
- Context-sensitive online help for every input window
- Support with contour input (geometry processor)

Call and return conditions

- The G functions active before the cycle call and the programmable frame remain active beyond the cycle.
- The starting position must be approached in the higher-level program before the cycle is called. The coordinates are programmed in a clockwise coordinate system.

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.

Program view

The program view in the editor provides an overview of the individual machining steps of a program.

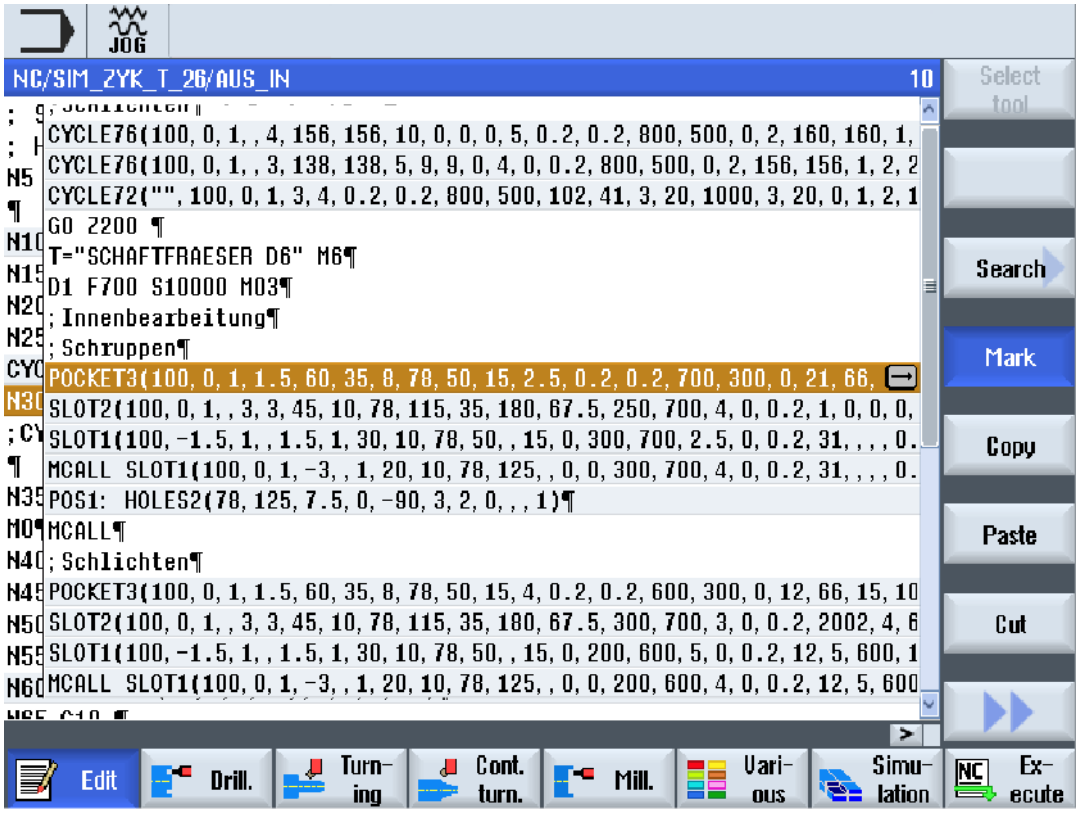
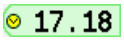
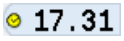
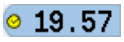
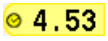


Figure 8-1 Program view of a G code program

Note

In the program editor settings, you define as to whether cycle calls are to be displayed as plain text or in NC syntax. You can also configure the recording of the machining times

Display of the machining times

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

Highlighting of selected G code commands or keywords

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

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

Machine manufacturer

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

Synchronization of programs on multi-channel machines

Special commands (e.g. GET and RELEASE) are used on multi-channel machines to synchronize the programs. These commands are marked with a clock symbol.

If the programs of several channels are displayed, the associated commands are displayed in one line.

Display	Meaning
	Synchronization command

In the program view, you can move between the program blocks using the <Cursor up> and <Cursor down> keys.

Parameter screen with help display

Press the <Cursor right> key to open a selected program block or cycle in the program view.

The associated parameter screen with help screen is then displayed.

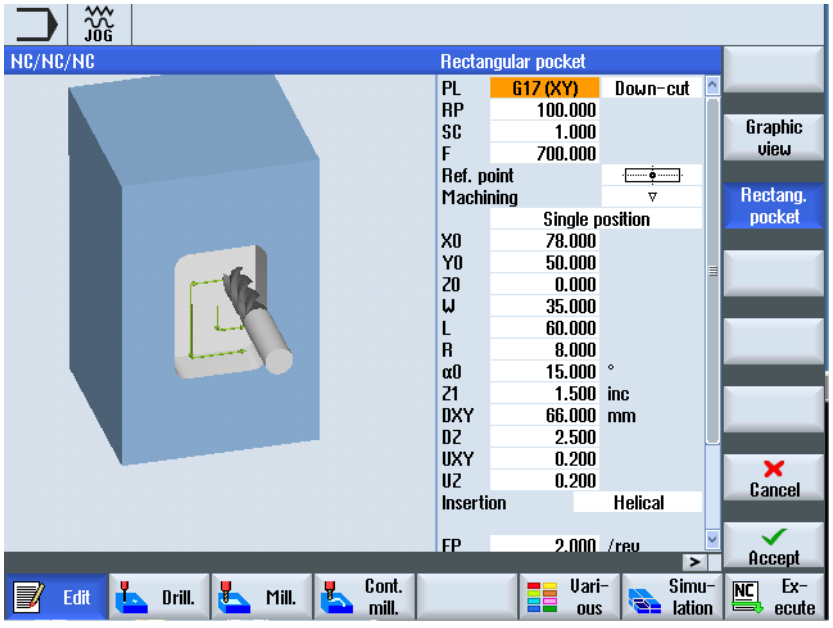


Figure 8-2 Parameter screen with help display

The animated help displays are always displayed with the correct orientation to the selected coordinate system. The parameters are dynamically displayed in the graphic. The selected parameter is displayed highlighted in the graphic.

The colored symbols

Red arrow = tool traverses in rapid traverse

Green arrow = tool traverses with the machining feedrate

Parameter screen with graphic view

Graphic view	Press the "Graphic view" softkey to toggle between the help screen and the graphic view.

Note

Switching between the help screen and the graphic view

The key combination <CTRL> + <G> is also available for the switchover between the help screen and the graphic view.

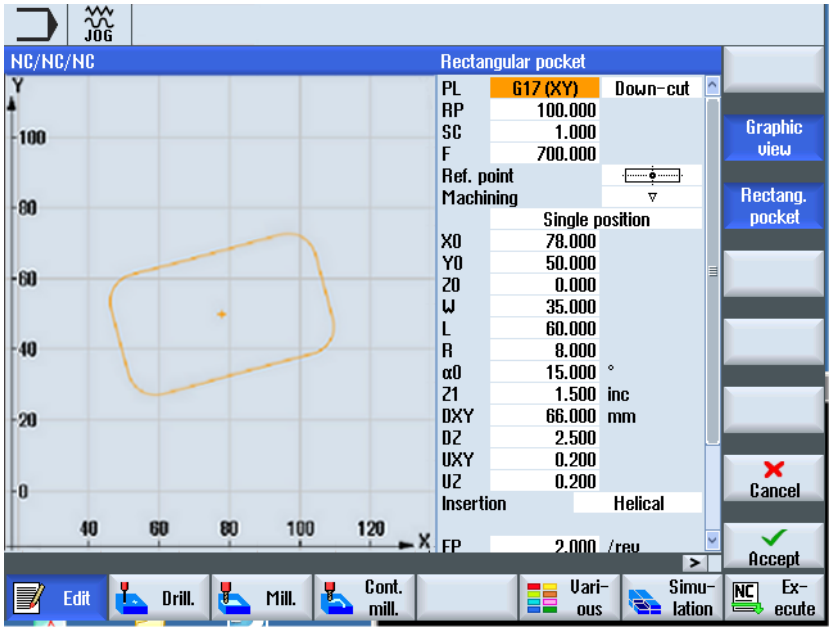


Figure 8-3 Parameter screen with a graphical view of a G code program block

See also

Editor settings (Page 229)

8.3 Program structure

G_code programs can always be freely programmed. The most important commands that are included in the rule:

- Set a machining plane
- Call a tool (T and D)
- Call a work offset
- Technology values such as feedrate (F), feedrate type (G94, G95,....), speed and direction of rotation of the spindle (S and M)
- Positions and calls, technology functions (cycles)
- End of program

For G code programs, before calling cycles, a tool must be selected and the required technology values F, S programmed.

A blank can be specified for simulation.

See also

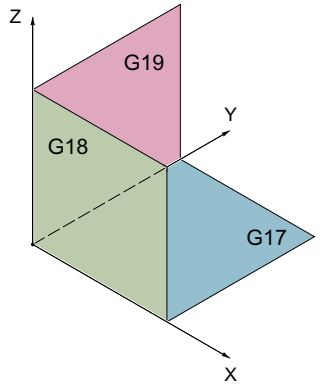
Blank input (Page 318)

8.4 Fundamentals

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 Current planes in cycles and input screens

Each input screen has a selection box for the planes, if the planes have not been specified by NC machine data.

- Empty (for compatibility reasons to screen forms without plane)
- G17 (XY)
- G18 (ZX)
- G19 (YZ)

There are parameters in the cycle screens whose names depend on this plane setting. These are usually parameters that refer to positions of the axes, such as reference point of a position pattern in the plane or depth specification when drilling in the tool axis.

For G17, reference points in the plane are called X0 Y0, for G18 they are called Z0 X0 - and for G19, they are called Y0 Z0. The depth specification in the tool axis for G17 is called Z1, for G18, Y1 and for G19, X1.

If the entry field remains empty, the parameters, the help screens and the broken-line graphics are displayed in the default plane (can be set via machine data):

- Turning: G18 (ZX)
- Milling: G17 (XY)

The plane is transferred to the cycles as new parameter. The plane is output in the cycle, i.e. the cycle runs in the entered plane. It is also possible to leave the plane fields empty and thus create a plane-independent program.

The entered plane only applies for this cycle (not modal)! At the end of the cycle, the plane from the main program applies again. In this way, a new cycle can be inserted in a program without having to change the plane for the remaining program.

8.4.3 Programming a tool (T)

Calling a tool

1. You are in the part program.

2. Press the "Select tool" softkey.

   The "Tool selection" window is opened.

3. Position the cursor on the desired tool and press the "To program" softkey.

   The selected tool is loaded into the G code editor. Text such as the following is displayed at the current cursor position in the G code editor:
   T="ROUGHINGTOOL100"

   - OR -

4. Press the "Tool list" and "New tool" softkeys.

5. Then select the required tool using the softkeys on the vertical softkey bar, parameterize it and then press the softkey "To program".

   The selected tool is loaded into the G code editor.

6. Then program the tool change (M6), the spindle direction (M3/M4), the spindle speed (S...), the feedrate (F), the feedrate type (G94, G95,...), the coolant (M7/M8) and, if required, further tool-specific functions.

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 "Workpiece" folder or under the "Part programs" folder.

Procedure

1. Select the "Program Manager" operating area.

2. Select the required archiving location.

Creating a new part program

3. Position the cursor on the folder "Part programs" and press the "New" softkey.

The "New G Code Program" window opens.

4. Enter the required name and press the "OK" softkey.

   The name can contain up to 28 characters (name + dot + 3-character extension). You can use any letters (except accented), digits or the underscore symbol (_).

   The program type (MPF) is set by default.

   The project is created and opened in the Editor.

Creating a new part program for a workpiece

5. Position the cursor on the folder "Workpieces" and press the "New" softkey.

The "New G Code Program" window opens.

6. Select the file type (MPF or SPF), enter the desired name of the program and press the "OK" softkey.

   The project is created and opened in the Editor.

7. Enter the desired G code commands.

See also

Changing a cycle call (Page 326)

Creating a new workpiece (Page 783)

8.6 Blank input

Function

The blank is used for the simulation and the simultaneous recording. A useful simulation can only be achieved with a blank that is as close as possible to the real blank.

Create a separate program for each new workpiece that you would like to produce. The program contains the individual machining steps that are performed to produce the workpiece.

For the blank of the workpiece, define the shape (cuboid, tube, cylinder, polygon or centered cuboid) and your dimensions.

Manually reclamping the blank

If the blank is to be manually reclamped from the main spindle to the counterspindle for example, then delete the blank.

Example

- Blank, main spindle, cylinder
- Machining
- M0 ; Manually reclamping the blank
- Blank, main spindle, delete
- Blank, counterspindle, cylinder
- Machining

The blank entry always refers to the work offset currently effective at the position in the program.

Note

Swiveling

For programs that use "Swiveling", a 0 swivel must first be made and then the blank defined.

Procedure

1. Select the "Program" operating area.

2. Press the "Misc." and "Blank" softkeys.
   The "Blank Input" window opens.

Parameter	Description	Unit
Data for	Selection of the spindle for blank • Main spindle • Counterspindle **Note:** If the machine does not have a counterspindle, then the entry field "Data for" is not applicable.	
Clamping	Selecting the clamping location of the blank • Table All clampings are mounted on a table **Note**: No turning cycles can be used in the program with the "Table" selection. • C1... All clampings are mounted on a rotary axis Note: please refer to the machine manufacturer's specifications.	
Blank	Selecting the blank • Cuboid • Tube • Cylinder • Polygon • Centered cuboid • Delete	
X0	1. Rectangular point X - (only for cuboid)	
Y0	1. Rectangular point Y - (only for cuboid)	
X1	2. Rectangular point X (abs) or 2nd rectangular point X referred to X0 (inc) - (only for cuboid)	
Y1	2. Rectangular point Y (abs) or 2nd rectangular point Y referred to Y0 (inc) - (only for cuboid)	
ZA	Initial dimension	
ZI	Final dimension (abs) or final dimension in relation to ZA (inc)	
ZB	Machining dimension (abs) or machining dimension in relation to ZA (inc)	
XA	Outside diameter – (only for tube and cylinder)	mm
XI	Inside diameter (abs) or wall thickness (inc) – (only for tube)	mm
N	Number of edges – (only for polygon)	
SW or L	Width across flats or edge length – (only for polygon)	
W	Width of the blank - (only for centered cuboid)	mm
L	Length of the blank - (only for centered cuboid)	mm

8.7 Machining plane, milling direction, retraction plane, safe clearance and feedrate (PL, RP, SC, F)

In the program header, cycle input screens have general parameters that are always repeated. You will find the following parameters in every input screen for a cycle in a G code program.

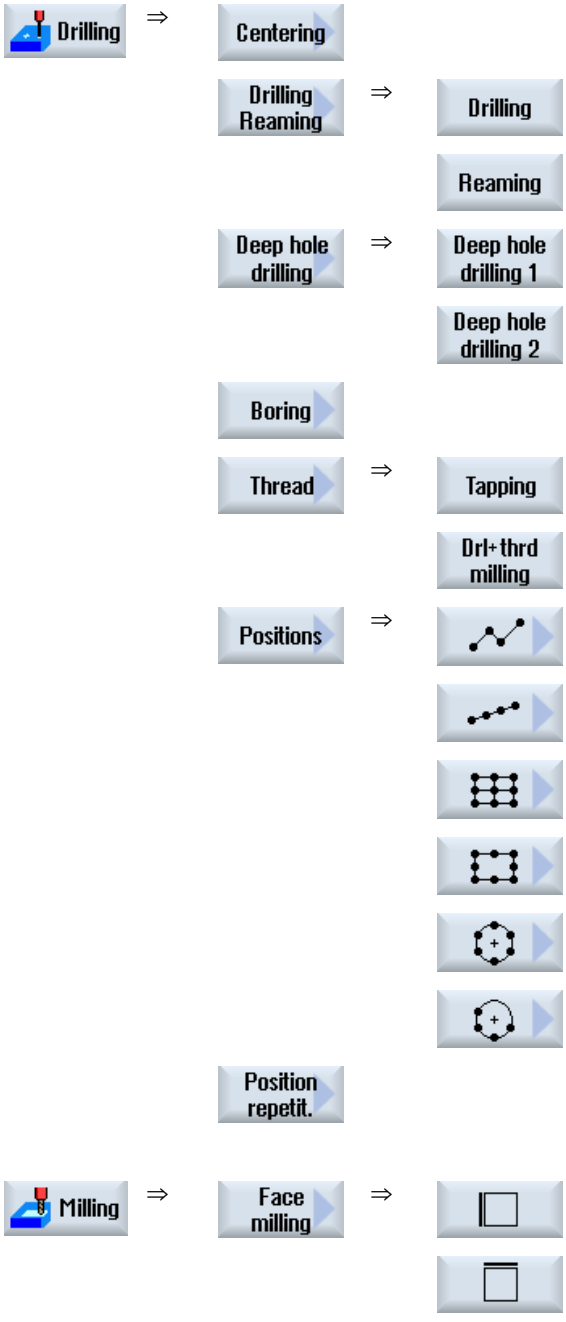
Parameter	Description	Unit
PL	Each input screen has a selection box for the planes, if the planes have not been specified by NC machine data. Machining plane: • G17 (XY) • G18 (ZX) • G19 (YZ)	
Milling direction	When milling, the machining direction of rotation (**climbing** or **conventional**) and the spindle direction of rotation in the tool list are taken into consideration. The pocket is then machined in a clockwise or counter-clockwise direction. During path milling, the programmed contour direction determines the machining direction.	
RP	Retraction plane (abs) During machining the tool travels in rapid traverse from the tool change point to the return plane and then to the safety clearance. The machining feedrate is activated at this level. When the machining operation is finished, the tool travels at the machining feedrate away from the workpiece to the safety clearance level. It travels from the safety clearance to the retraction plane and then to the tool change point in rapid traverse. The retraction plane is entered as an absolute value. Normally, reference point Z0 and retraction plane RP have different values. The cycle assumes that the retraction plane is in front of the reference point.	mm
SC	Safety clearance (inc) The safety clearance specifies from which clearance to the material rapid traverse is no longer used. The direction in which the safety clearance is active is automatically determined by the cycle. It is generally effective in several directions. The safety clearance is entered incrementally (without sign).	mm
F	Feedrate The feedrate F (also referred to as the machining feedrate) specifies the speed at which the axes move during machining of the workpiece. The unit of the feedrate (mm/min, mm/rev, mm/tooth etc.) always refers to the feedrate type programmed before the cycle call. The maximum feedrate is determined via machine data.	

8.8 Selection of the cycles via softkey

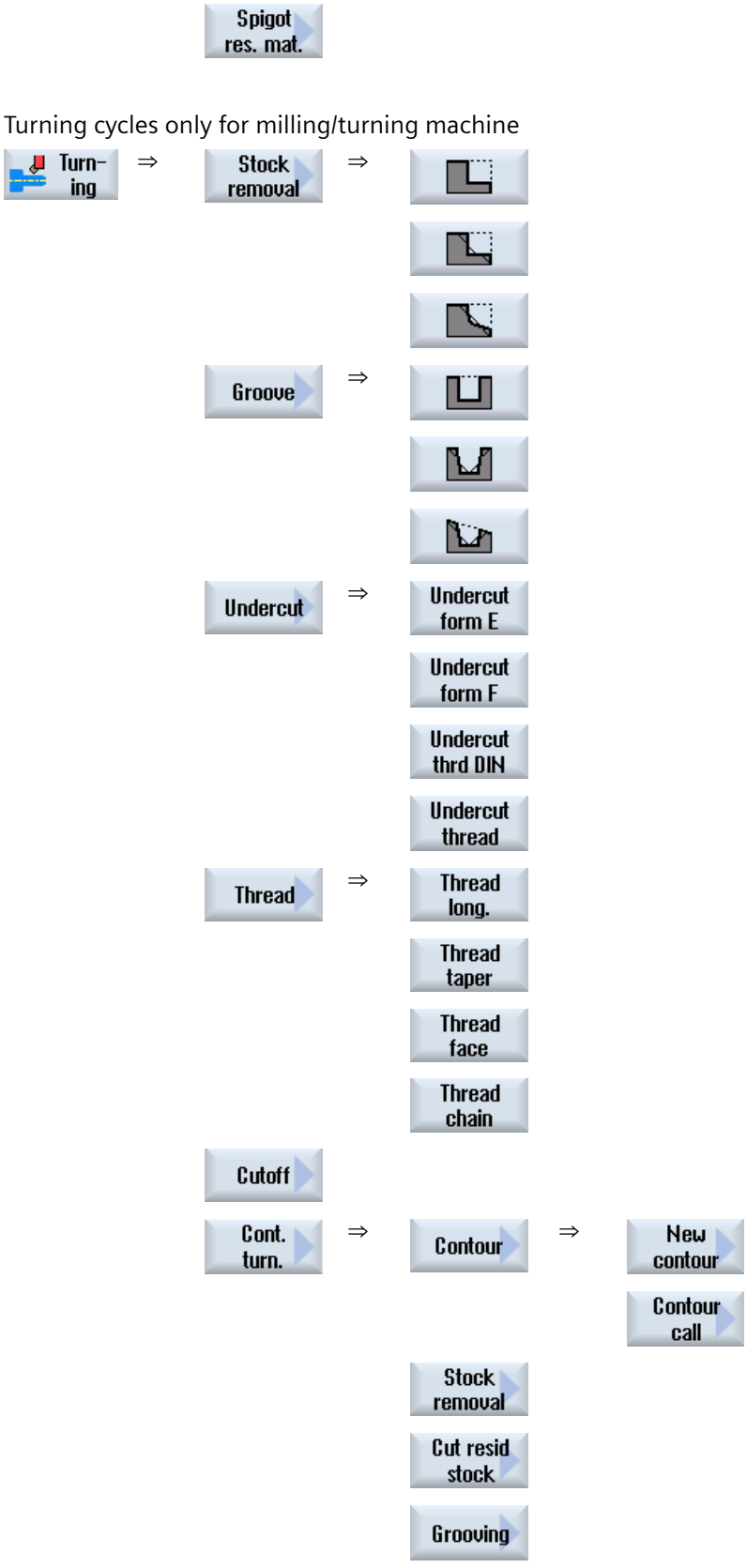
Overview of machining steps

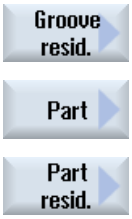
The following softkey bars are available to insert machining steps.

All of the cycles/functions available in the control are shown in this display. However, at a specific system, only the steps possible corresponding to the selected technology can be selected.

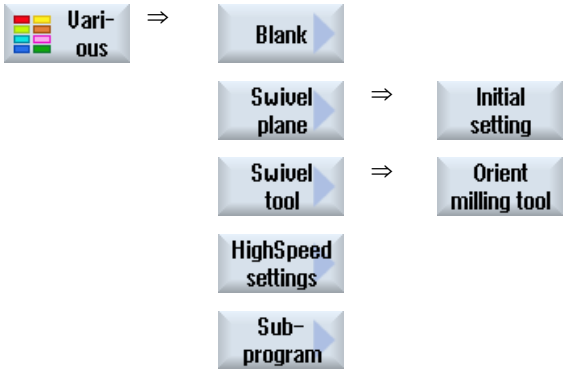


Pocket
⇒
Rectangle pocket
Circular pocket
Multi-edge spigot
⇒
Rectangle spigot
Circular spigot
Multi-edge
Slot
⇒
Longit. slot
Circumfer. slot
Open slot
Elongated hole
Thread milling
Engraving
Cont. mill.
⇒
Contour
⇒
New contour
Contour call
Path milling
Rough drill
⇒
Centering
Rough drill
Pocket
Pocket res.mat.
Spigot


Spigot res. mat.
Turning cycles only for milling/turning machine
Turn-ing
⇒
Stock removal
⇒
Groove
⇒
Undercut
⇒
Undercut form E
Undercut form F
Undercut thrd DIN
Undercut thread
Thread
⇒
Thread long.
Thread taper
Thread face
Thread chain
Cutoff
Cont. turn.
⇒
Contour
⇒
New contour
Contour call
Stock removal
Cut resid stock
Grooving



Note:
Please observe the information provided by the machine manufacturer.



⇒ **More information** on the available measuring variants of the measuring cycle function "Measure workpiece" can be found in the Measuring Cycles Programming Maunual.

⇒ **More information** on the available measuring variants of the measuring cycle function "Measure workpiece" can be found in the Measuring Cycles Programming Manual.

8.9 Calling technology functions

8.9.1 Hiding cycle parameters

The documentation describes all the possible input parameters for each cycle. Depending on the settings of the machine manufacturer, certain parameters can be hidden in the screens, i.e. not displayed. These are then generated with the appropriate default values when the cycles are called.

Cycle support

Example

1. Use the softkeys to select whether you want support for programming contours, drilling or milling cycles.

2. Select the desired cycle via the softkey.

3. Enter the parameters and press the "Accept" key.
   The cycle is transferred to the editor as G code.

8.9.2 Setting data for cycles

Cycle functions can be influenced and configured using machine and setting data.

More information on configuring the cycles can be found in the SINUMERIK Operate Commissioning Manual.

8.9.3 Checking cycle parameters

The entered parameters are already checked during the program creation in order to avoid faulty entries.

If a parameter is assigned an illegal value, this is indicated in the input screen and is designated as follows:

- The entry field has a colored background (background color, pink).
- A note is displayed in the comment line.
- If the parameter input field is selected using the cursor, the not is also displayed as tooltip.

The programming can only be completed after the incorrect value has been corrected.

Faulty parameter values are also monitored with alarms during the cycle runtime.

8.9.4 Programming variables

In principle, variables or expressions can also be used in the input fields of the screen forms instead of specific numeric values. In this way, programs can be created very flexibly.

Input of variables

Please note the following points when using variables:

- Values of variables and expressions are not checked since the values are not known at the time of programming.
- Variables and expressions cannot be used in fields in which a text is expected (e.g. tool name). An exception is the "Engraving" function , in which you can assign the desired text in the text field via a variable as "Variable text".
- Selection fields generally cannot be programmed with variables.

Examples

VAR_A

VAR_A+2*VAR_B

SIN(VAR_C)

8.9.5 Changing a cycle call

You have called the desired cycle via softkey in the program editor, entered the parameters and confirmed with "Accept".

Procedure

1. Select the desired cycle call and press the <Cursor right> key.
   The associated input screen of the selected cycle call is opened.

   - OR -

   Press the <SHIFT + INSERT> key combination.
   This starts the edit mode for this cycle call and you can edit it like a normal NC block. This means that it is possible to generate an empty block before the cycle is called. For instance, to insert something before a cycle that is located at the beginning of the program.
   Note: In edit mode, the cycle call can be changed in such a way that it can no longer be recompiled in the parameter screen.

   You exit the edit mode by pressing the <SHIFT + INSERT> key combination.

- OR -

You are in the edit mode and press the <INPUT> key.

A new line is created after the cursor position.

See also

Generating a G code program (Page 317)

8.9.6 Compatibility for cycle support

The cycle support is generally upwards compatible. This means that cycle calls in NC programs can always be recompiled with a higher software version, changed and then run again.

When transferring NC programs to a machine with a lower software version, it cannot be guaranteed, however, that the program will be able to be changed by recompiling cycle calls.

8.9.7 Additional functions in the input screens

Selection of units

If, for example, the unit can be switched in a field, this is highlighted as soon as the cursor is positioned on the element. In this way, the operator recognizes the dependency.

The selection symbol is also displayed in the tooltip.

Display of abs or inc

The abbreviations "abs" and "inc" for absolute and incremental values are displayed behind the entry fields when a switchover is possible for the field.

Help screens

2D and 3D graphics or sectional views are displayed for the parameterization of the cycles.

Online help

If you wish to obtain more detailed information about certain G code commands or cycle parameters, then you can call a context-sensitive online help.

8.10 Measuring cycle support

Measuring cycles are general subroutines designed to solve specific measurement tasks. They can be adapted to specific problems via parameter settings.

Software option
You require the "Measuring cycles" option to use "Measuring cycles".

You can find additional information on the use of measuring cycles in the Measuring Cycles Programming Manual.

Creating a ShopMill program

9

The program editor offers graphic programming to generate machining step programs that you can directly generate at the machine.

Software option

You require the "ShopMill/ShopTurn" option to generate ShopMill machining step programs.

Program loops

When you open a ShopMill program a program test is always executed. For larger program loops or nested program loops, this can result in performance problems in the editor. Therefore, always program program repetitions in the program end block.

Functions

The following functionality is available:

- Technology-oriented program step selection (cycles) using softkeys
- Input windows for parameter assignment with animated help screens
- Context-sensitive online help for every input window
- Support with contour input (geometry processor)

See also

Repeating program blocks (Page 352)

9.1 Program views

You can display a ShopMill program in various views:

- Work plan
- Graphic 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.

Work plan

The work plan in the editor provides an overview of the individual machining steps of a program.

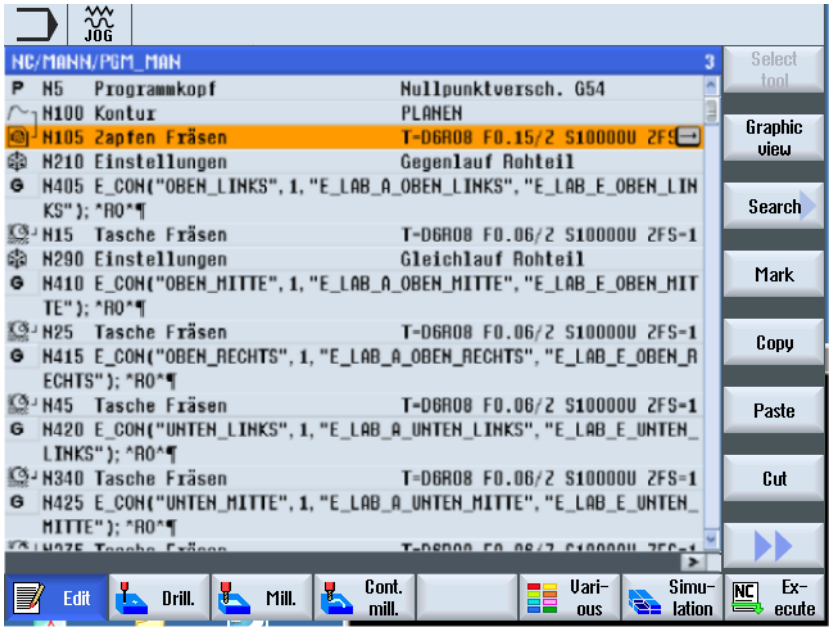
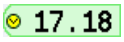
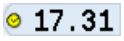
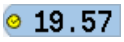
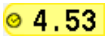


Figure 9-1 Work plan of a ShopMill program

Note

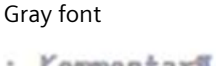
In the program editor settings, you can specify whether the machining times are to be recorded.

Display of the machining times

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

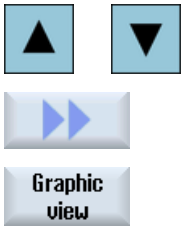
You can define further highlight colors in the "sleditorwidget.ini" configuration file.
Please observe the information provided by the machine manufacturer.

Synchronization of programs on multi-channel machines

Special commands (e.g. GET and RELEASE) are used on multi-channel machines to synchronize the programs. These commands are marked with a clock symbol.

If the programs of several channels are displayed, the associated commands are displayed in one line.

Display	Meaning
	Synchronization command



1. You can move between the program blocks in the work plan by pressing the <Cursor up> and <Cursor down> keys.
2. Press the ">>" and "Graphic view" softkeys to display the graphic view.

Note

Switching between the help screen and the graphic view

The key combination <CTRL> + <G> is also available for the switchover between the help screen and the graphic view.

Graphic view

The graphic view shows the contour of the workpiece as a dynamic graphic with broken lines. The program block selected in the work plan is highlighted in color in the graphic view.

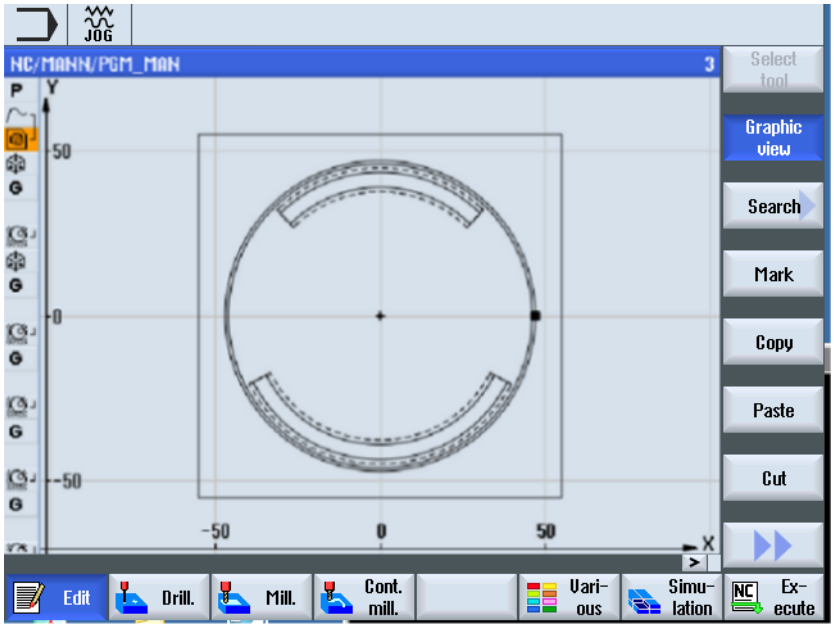


Figure 9-2 Graphic view of a ShopMill program

Parameter screen with help display

Press the <Cursor right> key to open a selected program block or cycle in the work plan.

The associated parameter screen with help screen is then displayed.

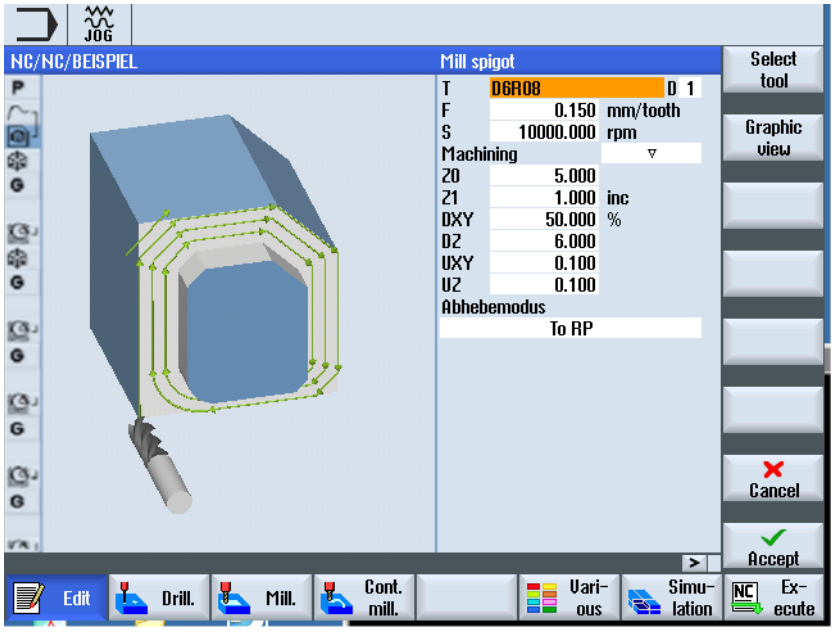


Figure 9-3 Parameter screen with help display

The animated help displays are always displayed with the correct orientation to the selected coordinate system. The parameters are dynamically displayed in the graphic. The selected parameter is displayed highlighted in the graphic.

The colored symbols

Red arrow = tool traverses in rapid traverse

Green arrow = tool traverses with the machining feedrate

Parameter screen with graphic view

Press the "Graphic view" softkey to toggle between the help screen and the graphic view in the screen.

Note

Switching between the help screen and the graphic view

The key combination <CTRL> + <G> is also available for the switchover between the help screen and the graphic view.

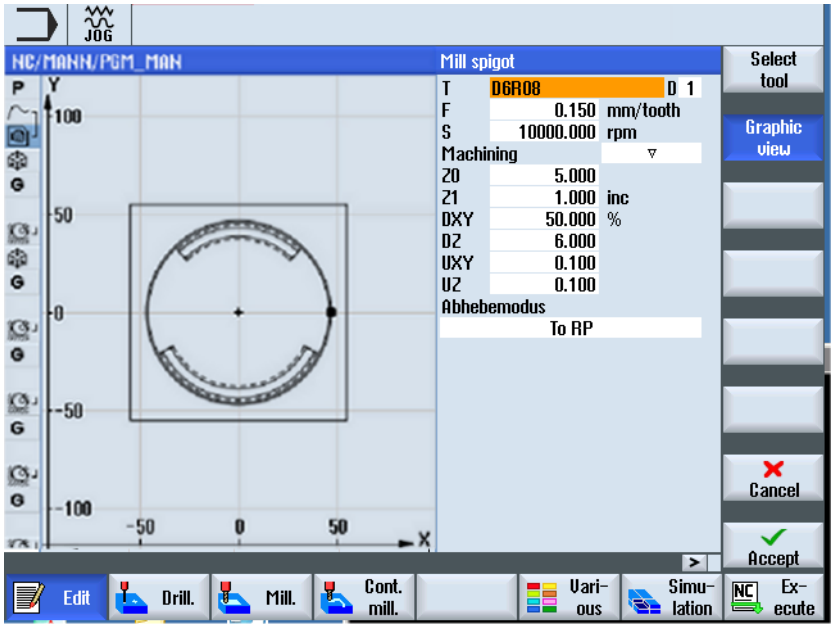


Figure 9-4 Parameter screen with graphic view

See also

Editor settings (Page 229)

9.2 Program structure

A machining step program is divided into three sub-areas:

- Program header
- Program blocks
- End of program

These sub-areas form a work plan.

Program header

The program header contains parameters that affect the entire program, such as blank dimensions or retraction planes.

Program blocks

You determine the individual machining steps in the program blocks. In doing this, you specify the technology data and positions, among other things.

Linked blocks

For the "Contour milling", "Milling", and "Drilling" functions, program the technology blocks and contours or positioning blocks separately. These program blocks are automatically linked by the control and connected by brackets in the work plan.

In the technology blocks, you specify how and in what form the machining should take place, e.g. centering first, and then drilling. In the positioning blocks, you define the positions for drilling or milling operations.

End of program

End of program signals to the machine that the machining of the workpiece has ended. Further, here you set whether program execute should be repeated.

Note

Number of workpieces

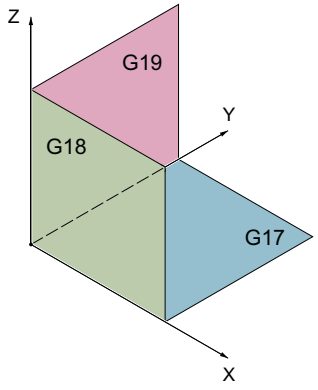
You can enter the number of required workpieces using the "Times, counters" window.

9.3 Fundamentals

9.3.1 Machining planes

A plane is defined by means of two coordinate axes. The third coordinate axis (tool axis) is perpendicular to this plane and determines the infeed direction of the tool (e.g. for 2½-D machining).

When programming, it is necessary to specify the working plane so that the control system can calculate the tool offset values correctly. The plane is also relevant to certain types of circular programming and polar coordinates.



Working planes

Working planes are defined as follows:

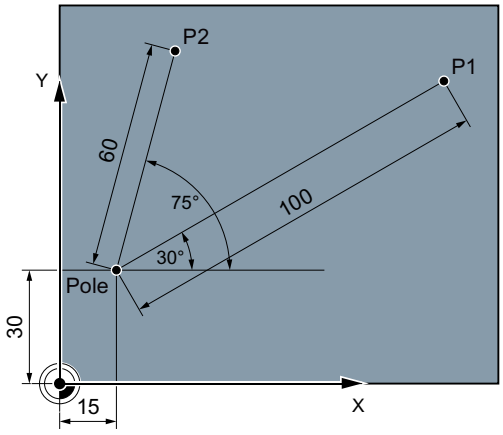
Plane		Tool axis
X/Y	G17	Z
Z/X	G18	Y
Y/Z	G19	X

9.3.2 Polar coordinates

The rectangular coordinate system is suitable in cases where dimensions in the production drawing are orthogonal. For workpieces dimensioned with arcs or angles, it is better to define positions using polar coordinates. This is possible if you are programming a straight line or a circle.

Polar coordinates have their zero point at the "pole".

Example



Points P1 and P2 can then be described – with reference to the pole – as follows:

P1: Radius =100 / angle =30°

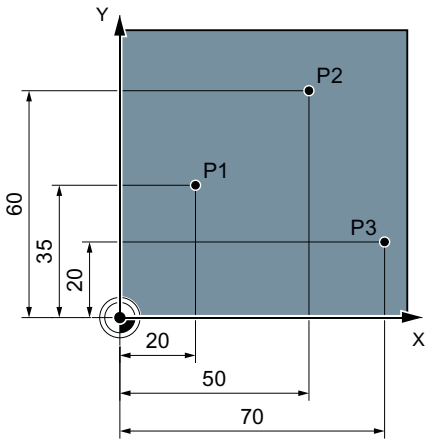
P2: Radius =60 / angle =75°

9.3.3 Absolute and incremental dimensions

Absolute dimensions

With absolute dimensions, all the position specifications refer to the currently valid zero point. Applied to tool movement this means: The absolute dimension data defines the position, to which the tool is to travel.

Example



The position data points P1 to P3 in absolute dimensions relative to the zero point are the following:

P1: X20 Y35

P2: X50 Y60

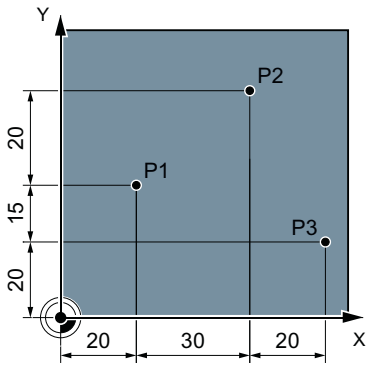
P3: X70 Y20

Incremental dimensions

In the case of production drawings in which dimensions refer to some other point on the workpiece rather than the zero point, it is possible to enter an incremental dimension.

When incremental dimensions are entered, each item of position data refers to a point programmed beforehand.

Example



The position data for points P1 to P3 in incremental dimensions are:

P1: X20 Y35 ;(referred to the zero point)

P2: X30 Y20 ;(referred to P1)

P3: X20 Y-35 ;(referred to P2)

9.4 Creating a ShopMill 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.

If you create a new program, a program header and program end are automatically generated.

ShopMill programs can be created in a new workpiece or under the folder "Part programs".

Procedure

1. Select the "Program Manager" operating area.

2. Select the desired storage location and position the cursor on the folder "Part programs" or under the folder "Workpieces" on the workpiece for which you wish to create a program.

3. Press the "New" and "ShopMill" softkeys.
   The "New machining step programming" 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 "ShopMill" program type is selected.
   The editor is opened and the "Program header" parameter screen is displayed.

Filling out the program header

5. Select a work offset and enter the dimensions of the blank and the parameters, which are effective over the complete program, e.g. dimension units in mm or inch, tool axis, retraction plane, safety clearance and machining direction.

6. Press the "Accept" softkey.
   The work plan is displayed. Program header and end of program are created as program blocks.
   The end of program is automatically defined.

See also

Creating a new workpiece (Page 783)

Changing program settings (Page 356)

9.5 Program header

In the program header, set the following parameters, which are effective for the complete program.

Parameter	Description	Unit
Dimension unit	The dimension unit (**mm** or **inch**) set in the program header only refers to the position data in the actual program. All other data, such as feedrate or tool offsets, is entered in the dimension unit that you have set for the entire machine.	mm inch
Work offset	The work offset in which the workpiece zero is saved. You can also delete the pre-setting of the parameter if you do not want to specify a work offset.	
Clamping	Selecting the clamping location of the blank for multiple clamping • Table All clampings are mounted on a table **Note**: No turning cycles can be used in the program with the "Table" selection • C1 ... All clampings are mounted on a rotary axis **Note**: Please refer to the machine manufacturer's specifications.	
Blank	Define the form and dimensions of the workpiece	
	• **Cylinder**	
∅A	Outer diameter ∅	mm
	• **Polygon**	
N	Number of edges	
SW / L	Width across flats Edge length	mm
	• **Centered cuboid**	
W	Width of blank	mm
L	Length of blank	mm
	• **Cuboid**	
X0	1. 1st corner point X	mm
Y0	1. 1st corner point Y	mm
X1	2. 2nd corner point X (abs) or 2nd corner point X referred to X0 (inc)	mm
Y1	2. 2nd corner point Y (abs) or 2nd corner point Y referred to Y0 (inc)	mm
ZA	Initial dimension	
ZI	Final dimension (abs) or final dimension in relation to ZA (inc)	
	• **Tube**	
∅A	Outer diameter ∅	mm
∅I	Inner diameter ∅ (abs) or wall thickness (inc)	mm
	• **Without** Without blank use	
HA - not for "Cuboid" and "Without" blanks	Initial dimension	mm

Parameter	Description	Unit
HI - not for "Cuboid" and "Without" blanks	Final dimension (abs) or final dimension in relation to HA (inc)	mm
PL	Machining plane G17 (XY) G18 (ZX) G19 (YZ) **Note**: The plane settings can already be defined. Ask the machine manufacturer in order that the selection box is available.	
Retraction plane RP Safety clearance SC:	Planes above the workpiece. During machining, the tool travels in rapid traverse from the tool change point to the return plane (RP) and then to the safety clearance (SC). The machining feedrate is activated at this height. When the machining operation is finished, the tool traverses at the machining feedrate away from the workpiece to the safety clearance height. It traverses from the safety clearance to the retraction plane and then to the tool change point in rapid traverse. The retraction plane is entered as an absolute value. The safety clearance must be entered as an incremental value (without sign).	
Machining direction	When machining a pocket, a longitudinal slot or a spigot, ShopMill takes the machining direction (**down-cut** or **up-cut**) and the spindle direction in the tool list into account. The pocket is then machined in a clockwise or counterclockwise direction. During path milling, the programmed contour direction determines the machining direction.	
Retraction position pattern	• **Optimized** When machining with optimized retraction, the tool travels across the workpiece at the machining feedrate depending on the contour and with a safety clearance (SC). • **To RP** When retracting to RP, the tool is retracted to the retraction plane when the machining step is complete and infeeds at the new position. Collisions with workpiece obstacles are thus prevented when the tool is retracted and fed in, e.g. when holes in pockets or grooves are machined at different levels and positions.	

See also

Program header setting, "Clamping" (Page 830)

9.6 Program header (for milling/turning machine)

In the program header, set the following parameters, which are effective for the complete program.

Parameter	Description	Unit
Dimension unit	The dimension unit (**mm** or **inch**) set in the program header only refers to the position data in the actual program. All other data, such as feedrate or tool offsets, is entered in the dimension unit that you have set for the entire machine.	mm inch
Work offset	The work offset in which the workpiece zero is saved. You can also delete the pre-setting of the parameter if you do not want to specify a work offset.	
Clamping	Selecting the clamping location of the blank • Table All clampings are mounted on a table **Note**: No turning cycles can be used in the program with the "Table" selection • C1 ... All clampings are mounted on a rotary axis **Note**: Please refer to the machine manufacturer's specifications.	
Blank	Define the form and dimensions of the workpiece	
	• **Cylinder**	
ØA	Outer diameter ∅	mm
	• **Polygon**	
N	Number of edges	
SW / L	Width across flats Edge length	mm
	• **Centered cuboid**	
W	Width of blank	mm
L	Length of blank	mm
	• **Cuboid**	
X0	1. corner point X	mm
Y0	1. corner point Y	mm
X1	2. corner point X (abs) or 2nd corner point X referred to X0 (inc)	mm
Y1	2. corner point Y (abs) or 2nd corner point Y referred to Y0 (inc)	mm
ZA	Initial dimension	mm
ZI	Final dimension (abs) or final dimension in relation to ZI (inc)	mm
	• **Tube**	
∅A	Outer diameter ∅	mm
ØI	Inner diameter ∅ (abs) or wall thickness (inc)	mm
	• **Without** Without blank use	
HA - not for "Cuboid" and "Without" blanks	Initial dimension	mm

Parameter	Description	Unit
HI - not for "Cuboid" and "Without" blanks	Final dimension (abs) or final dimension in relation to HA (inc)	mm
PL	Selecting the machining plane • Machining planes for milling G17 (XY) G18 (ZX) G19 (YZ) • Machining plane when turning G18 (ZX)	
Retraction, milling - only for "Cuboid" blank	Retraction planes above the workpiece. During machining, the tool travels in rapid traverse from the tool change point to the return plane (RP) and then to the safety clearance (SC). The machining feedrate is activated at this height. When the machining operation is finished, the tool traverses at the machining feedrate away from the workpiece to the safety clearance height. It traverses from the safety clearance to the retraction plane and then to the tool change point in rapid traverse. The retraction plane is entered as an absolute value. The safety clearance must be entered as an incremental value (without sign). **Note:** "Retraction milling" cannot be used in turning applications.	
Retraction turning - only for "Cylinder", "Tube", "Centered cuboid", "Polygon" and "Without" blanks	The retraction area indicates the area outside of which collision-free traversing of the axes must be possible.	
	• **Simple**	
XRA	Retraction plane X external ∅ (abs) or retraction plane X referred to HA (inc)	
XRI	- only for "Tube" blank Retraction plane X internal ∅ (abs) or retraction plane X referred to HI (inc)	
ZRA	Retraction plane Z front (abs) or retraction plane Z referred to HA (inc)	
	• **Extended** - not for a "Tube" blank	
XRA	Retraction plane X external ∅ (abs) or retraction plane X referred to HA (inc)	
XRI	Retraction plane X internal ∅ (abs) or retraction plane X referred to HI (inc)	
ZRA	Retraction plane Z front (abs) or retraction plane Z referred to HA (inc)	
	• **All**	
XRA	Retraction plane X external ∅ (abs) or retraction plane X referred to HA (inc)	
XRI	Retraction plane X internal ∅ (abs) or retraction plane X referred to HI (inc)	

Parameter	Description	Unit
ZRA ↻	Retraction plane Z front (abs) or retraction plane Z referred to HA (inc)	
ZRI ↻	Retraction plane Z rear	
	• **No** No turning cycles can be used.	
Tailstock - not for retraction: "No" ↻	• Yes • No	
XRR - not for retraction: "No"	Retraction plane tailstock – (only "Yes" for tailstock)	
S1 - not for retraction: "No"	Speed limit for G96 when turning	
PL ↻	Machining plane for milling: G17 (XY) G18 (ZX) G19 (YZ) **Note**: The plane settings can already be defined. Ask the machine manufacturer in order that the selection box is available. Machining plane when turning: G18 (ZX)	
SC	The safety clearance defines how close the tool can approach the workpiece in rapid traverse. **Note** Enter the safety clearance without sign into the incremental dimension.	
Machining direction ↻	When machining a pocket, a longitudinal slot or a spigot, ShopMill takes the machining direction (**down-cut** or **up-cut**) and the spindle direction in the tool list into account. The pocket is then machined in a clockwise or counterclockwise direction. During path milling, the programmed contour direction determines the machining direction.	
Retraction position pattern ↻	• **Optimized** When machining with optimized retraction, the tool travels across the workpiece at the machining feedrate depending on the contour and with a safety clearance (SC). • **To RP** When retracting to RP, the tool is retracted to the retraction plane when the machining step is complete and infeeds at the new position. Collisions with workpiece obstacles are thus prevented when the tool is retracted and fed in, e.g. when holes in pockets or grooves are machined at different levels and positions.	

9.7 Generating program blocks

After a new program is created and the program header is filled out, define the individual machining steps in program blocks that are necessary to machine the workpiece.

You can only create the program blocks between the program header and the program end.

Procedure

Selecting a technological function

1. Position the cursor in the work plan on the line behind which a new program block is to be inserted.
2.

Using the softkeys, select the desired function. The associated parameter screen is displayed.

...

3. First, program the tool, offset value, feedrate and spindle speed (T, D, F, S, V) and then enter the values for the other parameters.

Selecting a tool from the tool list

4.

Press the "Select tools" softkey if you wish to select the tool for parameter "T".

   The "Tool selection" window is opened.

5.

Position the cursor on the tool that you wish to use for machining and press the "To program" softkey.

   The selected tool is accepted into the parameter screen form.

   - OR -

Press the "Tool list" and "New tool" softkeys.

   Using the softkeys on the vertical softkey bar, select the required tool with the data and press the "To program" softkey.

   The selected tool is accepted into the parameter screen form.

   The process plan is displayed and the newly generated program block is marked.

9.8 Tool, offset value, feed and spindle speed (T, D, F, S, V)

Generally, the following parameters are entered for a program block.

Tool (T)

Each time a workpiece is machined, you must program a tool. Tools are selected by name, and the selection is integrated in all parameter screen forms of the machining cycles (with the exception of the straight line/circle).

The tool length offsets become active as soon as the tool is changed.

Tool selection is modal for the straight line/circle, i.e. if the same tool is used to perform several machining steps occur in succession, you only have to program one tool for the first straight line/circle.

Cutting edge (D)

In the case of tools with several cutting edges, there is a separate set of individual tool offset data for each edge. For these tools, you must select or specify the number of the cutting edge that you would like to use for machining.

NOTICE
Risk of collision Collisions may occur if you specify the wrong cutting edge number for some tools (e.g. a flat chamfering drill with guide spigot or step drill) and then traverse the tool. Always ensure that you enter the correct cutting edge number.

Tool length compensation

Tool length compensation takes effect as soon as the tool is loaded into the spindle. Different tool offsets can be assigned to each tool with multiple cutting edges.

The tool length compensation of the spindle tool remains active even after the program has been executed (RESET).

Radius compensation

The tool radius compensation is automatically included in the machining cycles except for path milling.

For path milling and straight line/circle, you have the option of programming the machining with or without radius compensation. The tool radius compensation is modal for straight lines/

circles, i.e. you have to deselect the radius compensation if you want to traverse without radius compensation.

- Radius compensation to right of contour
- Radius compensation to left of contour
- Radius compensation off
- Radius compensation remains as previously set

Feedrate (F)

The feedrate F (also referred to as the machining feedrate) specifies the speed at which the tool moves when machining the workpiece. The machining feedrate is entered in mm/min, mm/rev or in mm/tooth. The feedrate for milling cycles is automatically converted when switching from mm/min to mm/rev and vice versa.

It is only possible to enter the feedrate in mm/tooth during milling; this ensures that each cutting edge of the milling cutter is cutting under the best possible conditions. The feedrate per tooth corresponds to the linear path traversed by the milling cutter when a tooth is engaged.

With milling cycles, the feedrate for rough cutting is relative to the milling tool center point. This also applies to finish cutting, with the exception of concave curves where the feedrate is relative to the contact point between the tool and workpiece.

The maximum feedrate is determined via machine data.

Converting the feedrate (F) for drilling and milling

The feedrate entered for drilling cycles is automatically converted when switching from mm/min to mm/rev and vice versa using the selected tool diameter.

The feedrate entered for milling cycles is automatically converted when switching from mm/Z to mm/min and vice versa using the selected tool diameter.

Spindle speed (S) / cutting rate (V)

You have the option of either programming the spindle speed (S) or the cutting rate (V). You can toggle between them using the <SELECT> key.

In the milling cycles, the spindle speed is automatically converted to the cutting rate and vice versa.

- Spindle speed and cutting rate remain valid until you program a new tool.
- Spindle speeds are programmed in rpm.
- Cutting rates are programmed in m/min
- You can set the direction of rotation of a tool in the tool list.

Converting the spindle speed (S) / cutting rate (V) when milling.

9.9 Defining machine functions

You can switch-on the coolant or stop machining between the individual machining steps.

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

You have the option of defining machine functions as well as your own texts in the "Machine functions" window.

More information

More information on configuration options can be found in the SINUMERIK Operate Commissioning Manual.

Procedure

1. The ShopMill program to be edited has been created and you are in the editor.
2. Press the menu forward key and the "Straight Circle" softkey.

3. Press the "Machine functions" softkey.
   The "Machine functions" window opens.
4. Enter the desired parameters.

5. Press the "Accept" softkey.

See also

Starting and stopping a spindle manually (Page 174)

Parameters	Description	Unit
Spindle	Spindle M function, defines the spindle direction of rotation or spindle position • Spindle off • Spindle rotates clockwise • Spindle rotates counterclockwise • Spindle positions	
Stop position	Spindle stop position - (only for spindle M function SPOS)	Degrees
Other M function	Machine functions that are additionally provided by the machine manufacturer (e.g. "Close door").	

Parameters	Description	Unit
Coolant 1	Selects coolant (switches coolant 1 on or off) • with • without	
Coolant 2	Selects coolant (switches coolant 2 on or off) • with • without	
Tool-spec. function 1	User machine functions on/off	
Tool-spec. function 2	User machine functions on/off	
Tool-spec. function 3	User machine functions on/off	
Tool-spec. function 4	User machine functions on/off	
DT	Dwell time in seconds Time after which machining is continued.	s
Programmed stop	Programmed stop on Stops machining at the machine if, under Machine in the "Program control" window, the check box "Programmed stop" was activated.	
Stop	Stop on Stops machining at the machine.	

Note

Coolant after a block search

After a block search, the coolant specified for a tool in the tool management is always output after a block search even if a different coolant is programmed here.

In this case, the desired coolant can be activated with the "Overstore" function.

9.10 Call work offsets

You can call work offsets (G54, etc.) from any program.

You define work offsets in work offset lists. You can also view the coordinates of the selected offset here.

Procedure

1. Press the "Various", "Transformations" and "Work offset" softkeys.
   The "Work offset" window opens.

2. Select the desired work offset (e.g. G54).

3. Press the "Accept" softkey.
   The work offset is transferred into the work plan.

9.11 Repeating program blocks

If certain steps when machining a workpiece have to be executed more than once, it is only necessary to program these steps once. You have the option of repeating program blocks.

Note

Machining several workpieces

The program repeat function is not suitable to program repeat machining of parts.

In order to repeatedly machine the same workpieces (Page 354), program this using "end of program".

Start and end marker

You must mark the program blocks that you want to repeat with a start and end marker. You can then call these program blocks up to 200 times within a program. The markers must be unique, i.e. they must have different names. No names used in the NCK can be used.

You can also set markers and repeats after creating the program, but not within linked program blocks.

Note

You can use one and the same marker as end marker for preceding program blocks and as start marker for following program blocks.

Procedure

1. Position the cursor at the program block, behind which a program block that will be repeated should follow.

2. Press the "Various" softkey.

3. Press the ">>" and "Repeat progr." softkeys.

4. Press the "Set marker" and "Accept" softkeys.
   A start marker is inserted behind the actual block.

5. Enter the program blocks that you want to repeat later.

6. Press the "Set marker" and "Accept" softkeys again.
   An end marker is inserted after the actual block.

7. Continue programming up to the point where you want to repeat the program blocks.

8. Press the "Various" and "Repeat progr." softkeys.

9. Enter the names of the start and end markers and the number of times the blocks are to be repeated.

10. Press the "Accept" softkey.

    The marked program blocks are repeated.

9.12 Specifying the number of workpieces

If you wish to produce a certain quantity of the same workpiece, then at the end of the program, specify that you wish to repeat the program.

Control the numbers of times that the program is repeated using the "Times, counters" window. Enter the number of required workpieces using the target number. You can track the number of machined and completed workpieces in the actual counter window.

Controlling program repetition

End of program: Repeat	**Times, counter: Counts the workpieces**	
No	No	A CYCLE START is required for each workpiece.
No	Yes	A CYCLE START is required for each workpiece. The workpieces are counted.
Yes	Yes	The program is repeated without a new CYCLE START until the required number of workpieces have been machined.
Yes	No	Without a new CYCLE START, the program is repeated an infinite number of times. You can interrupt program execution with <RESET>.

Procedure

1. Open the "Program end" program block, if you want to machine more than one workpiece.
2. In the "Repeat" field, enter "Yes".
3. Press the "Accept" softkey.

   If you start the program later, program execution is repeated.

   Depending on the settings in the "Times, counters" window, the program is repeated until the set number of workpieces has been machined.

See also

Displaying the program runtime and counting workpieces (Page 279)

9.13 Changing program blocks

You can subsequently optimize the parameters in the programmed blocks or adapt them to new situations, e.g. if you want to increase the feedrate or shift a position. In this case, you can directly change all the parameters in every program block in the associated parameter screen form.

Procedure

1. Select the program that you wish to change in the "Program Manager" operating area.

2. Press the <Cursor right> or <INPUT> key.
   The work plan of the program is displayed.

3. Position the cursor in the work plan at the desired program block and press the <Cursor right> key.
   The parameter screen for the selected program block is displayed.
4. Make the desired changes.

5. Press the "Accept" softkey.

   - OR -

   Press the <Cursor left> key.

   The changes are accepted in the program.

9.14 Changing program settings

Function

All parameters defined in the program header, with the exception of the dimension unit, can be changed at any location in the program.

The settings in the program header are modal, i.e. they remain active until they are changed.

For the simulation and the simultaneous recording use a blank. A useful simulation can only be achieved with a blank that is as close as possible to the real blank.

For the blank of the workpiece, define the shape (cuboid, tube, cylinder, polygon or centered cuboid) and your dimensions.

The blank entry always refers to the work offset currently effective at the position in the program.

Procedure

1. Select the "Program" operating area.

2. Press the "Various" and "Settings" softkeys.
   The "Settings" input window opens.

Parameters	Description	Unit
Clamping	Selecting the clamping location of the blank • Table The blank is mounted on a table. • C1... The blank is mounted on a rotary axis. **Note**: Please refer to the machine manufacturer's specifications.	
Blank	Selecting the blank: • Cylinder • Tube • Centered cuboid • Cuboid • Polygon • Without	
	• **Cylinder**	
ØA	Outer diameter Ø	mm
	• **Tube**	

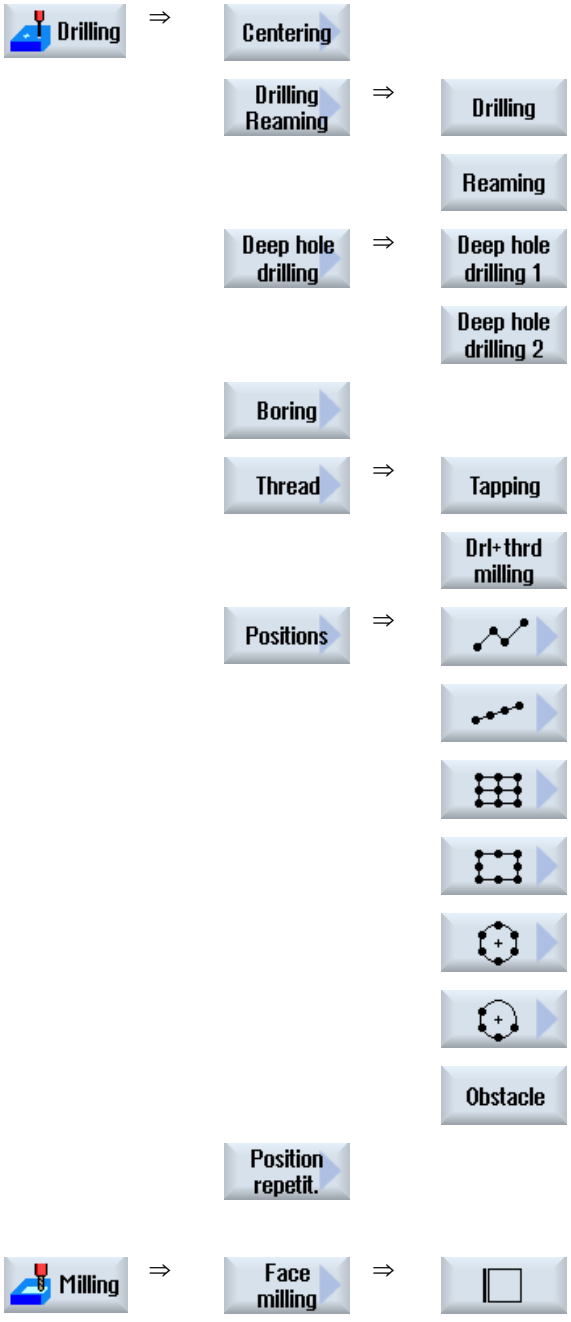
Parameters	Description		Unit
ØA		Outer diameter Ø	mm
ØI		Inner diameter Ø (abs) or wall thickness (inc)	mm
	• **Centered cuboid**		
W		Width of blank	mm
L		Length of blank	mm
	• **Cuboid**		
X0		1st corner point X	mm
Y0		1st corner point Y	mm
X1		2nd corner point X (abs) or 2nd corner point X referred to X0 (inc)	mm
Y1		2nd corner point X (abs) or 2nd corner point X referred to X0 (inc)	mm
ZA		Initial dimension	mm
ZI		Final dimension (abs) or final dimension in relation to ZA (inc)	mm
	• **Polygon**		
N		Number of edges	
SW L		Width across flats Edge length	mm
	• **Without** No blank use		
HA	Initial dimension		mm
HI	Final dimension (abs) or final dimension in relation to HA (inc)		mm
PL	Machining plane • G17 (XY) • G18 (ZX) • G19 (YZ)		
RP	Retraction plane (abs)		mm
SC	Safety clearance (inc) Acts in relation to the reference point. The direction in which the safety clearance is effective is automatically determined by the cycle.		mm
Machining direction	Milling direction • Down-cut • Up-cut		
Retraction position pattern	Lift mode before new infeed • To RP • Optimized		mm

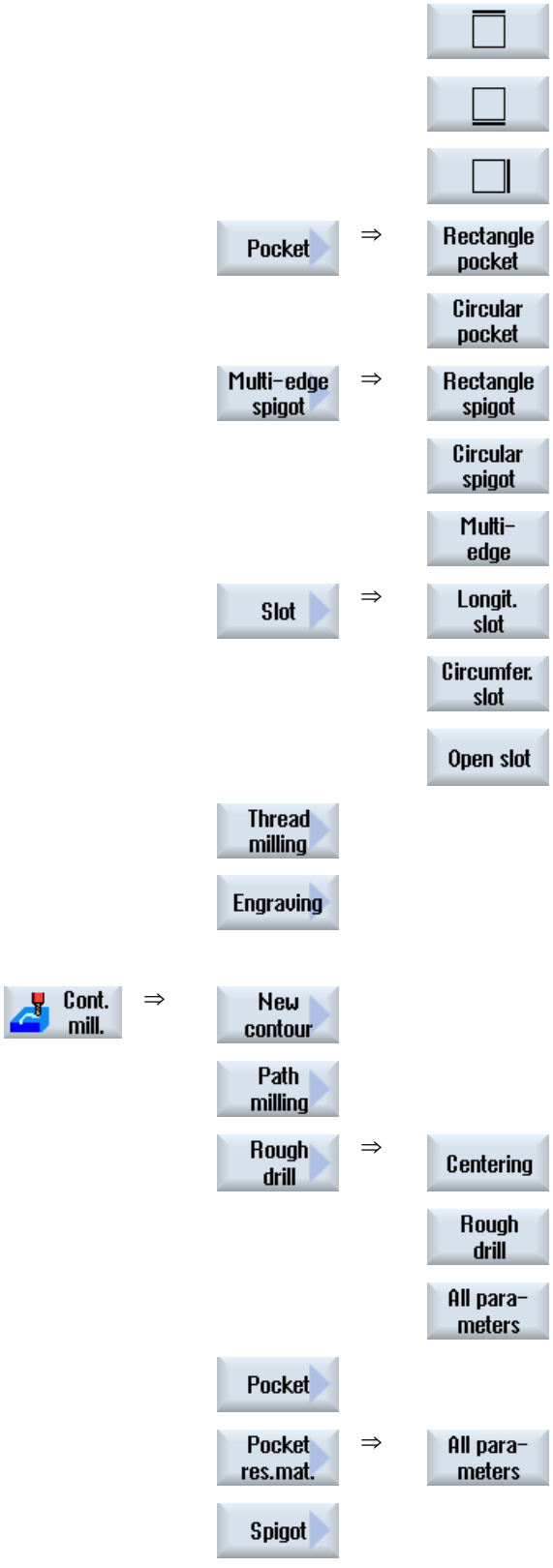
9.15 Selection of the cycles via softkey

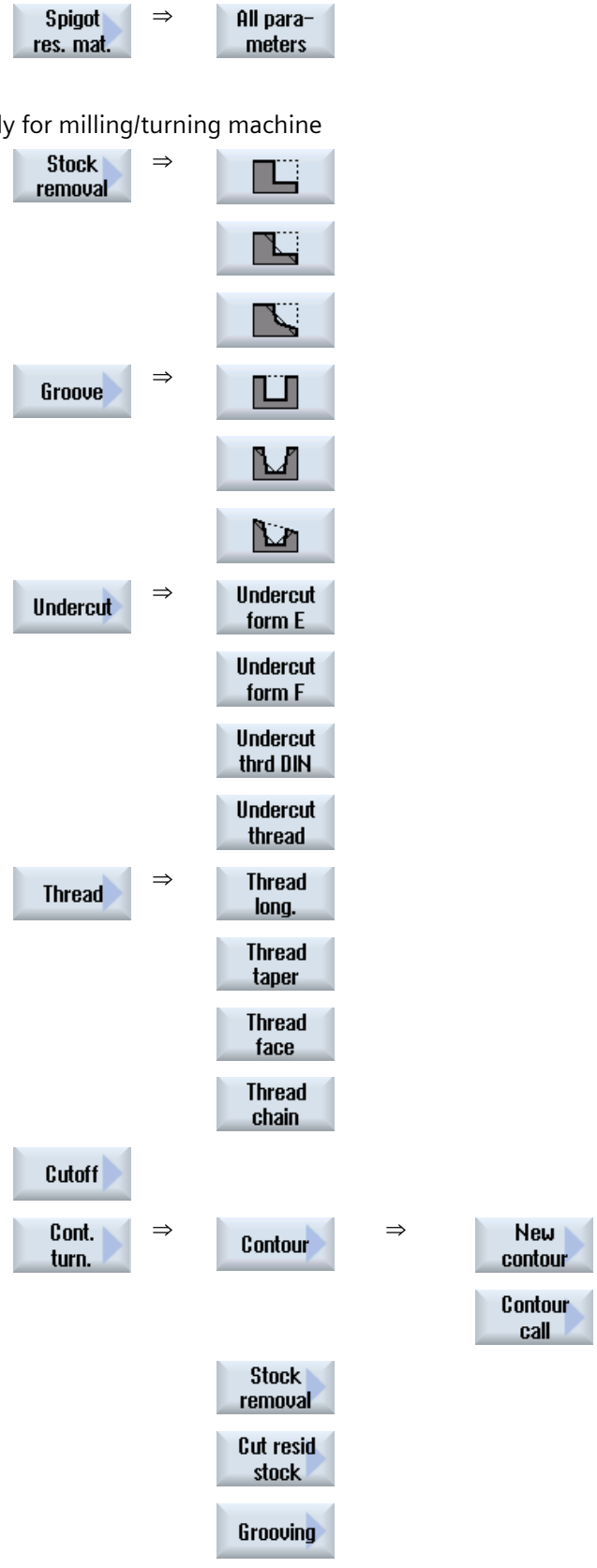
Overview of machining steps

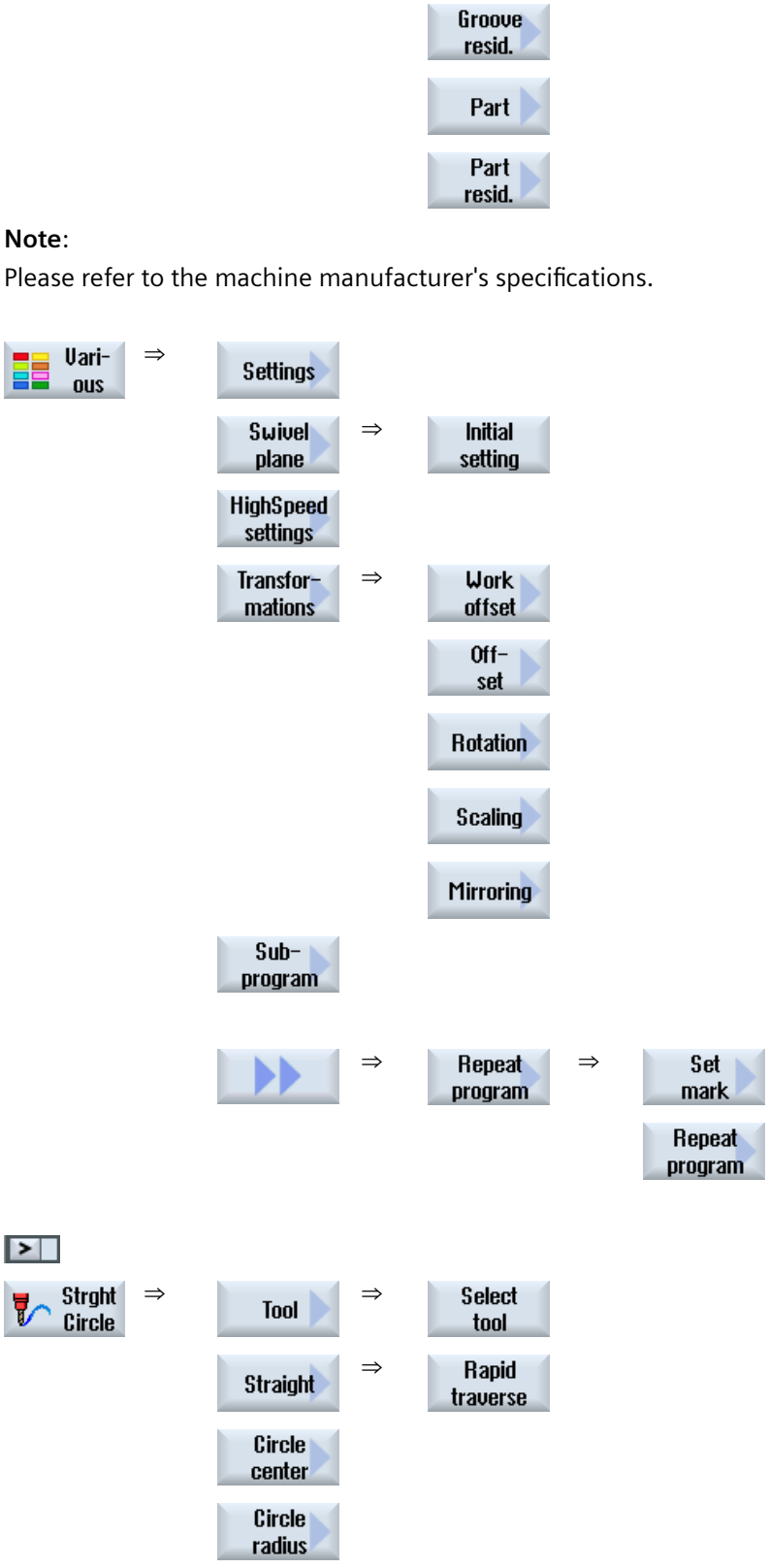
The following machining steps are available for insertion.

All of the cycles/functions available in the control are shown in this display. However, at a specific system, only the steps possible corresponding to the selected technology can be selected.

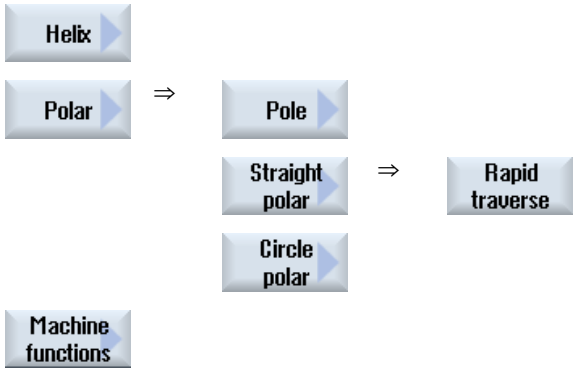



Pocket
⇒
Rectangle pocket
Circular pocket
Multi-edge spigot
⇒
Rectangle spigot
Circular spigot
Multi-edge
Slot
⇒
Longit. slot
Circumfer. slot
Open slot
Thread milling
Engraving
Cont. mill.
⇒
New contour
Path milling
Rough drill
⇒
Centering
Rough drill
All para-meters
Pocket
Pocket res.mat.
⇒
All para-meters
Spigot


Spigot res. mat.
⇒
All para-meters
Turning cycles only for milling/turning machine
Turn-ing
Stock removal
⇒
Groove
⇒
Undercut
⇒
Undercut form E
Undercut form F
Undercut thrd DIN
Undercut thread
Thread
⇒
Thread long.
Thread taper
Thread face
Thread chain
Cutoff
Cont. turn.
⇒
Contour
⇒
New contour
Contour call
Stock removal
Cut resid stock
Grooving



Note:
Please refer to the machine manufacturer's specifications.



⇒ **More information** on the available measuring variants of the measuring cycle function "Measure workpiece" can be found in the Measuring Cycles Programming Maunual.

⇒ **More information** on the available measuring variants of the measuring cycle function "Measure workpiece" can be found in the Measuring Cycles Programming Manual.

9.16 Calling technology functions

9.16.1 Additional functions in the input screens

Selection of units

If, for example, the unit can be switched in a field, this is highlighted as soon as the cursor is positioned on the element. In this way, the operator recognizes the dependency.

The selection symbol is also displayed in the tooltip.

Display of abs or inc

The abbreviations "abs" and "inc" for absolute and incremental values are displayed behind the entry fields when a switchover is possible for the field.

Help screens

2D and 3D graphics or sectional views are displayed for the parameterization of the cycles.

Online help

If you wish to obtain more detailed information about certain G code commands or cycle parameters, then you can call a context-sensitive online help.

9.16.2 Checking input parameters

When generating the program, the parameters that are entered are already checked in order to avoid making incorrect entries.

If a parameter is assigned an illegal value, this is indicated in the input screen and is designated as follows:

- The entry field has a colored background (background color, pink).
- A note is displayed in the comment line.
- If the parameter input field is selected using the cursor, the note is also displayed as tooltip.

The programming can only be completed after the incorrect value has been corrected.

Incorrect parameter values are also monitored with alarms during the cycle runtime.

9.16.3 Setting data for technological functions

Technological functions can be influenced and corrected using machine or setting data.

More information on configuring the cycles can be found in the SINUMERIK Operate Commissioning Manual.

9.16.4 Changing a cycle call

You have called the desired cycle via softkey in the program editor, entered the parameters and confirmed with "Accept".

Procedure

1. Select the desired cycle call and press the <Cursor right> key.
   The associated input screen of the selected cycle call is opened.

- OR -

Press the <SHIFT + INSERT> key combination.
This starts the edit mode for this cycle call and you can edit it like a normal NC block. This means that it is possible to generate an empty block before the cycle is called. For instance, to insert something before a cycle that is located at the beginning of the program.
Note: In edit mode, the cycle call can be changed in such a way that it can no longer be recompiled in the parameter screen.

You exit the edit mode by pressing the <SHIFT + INSERT> key combination.

- OR -

You are in the edit mode and press the <INPUT> key.
A new line is created after the cursor position.

9.16.5 Programming variables

In principle, variables or expressions can also be used in the input fields of the screen forms instead of specific numeric values. In this way, programs can be created very flexibly.

Input of variables

Please note the following points when using variables:

- Values of variables and expressions are not checked since the values are not known at the time of programming.
- Variables and expressions cannot be used in fields in which a text is expected (e.g. tool name). An exception is the "Engraving" function, in which you can assign the desired text in the text field via a variable as "Variable text".
- Selection fields generally cannot be programmed with variables.

Examples

VAR_A

VAR_A+2*VAR_B

SIN(VAR_C)

9.16.6 Compatibility for cycle support

The cycle support is generally upwards compatible. This means that cycle calls in NC programs can always be recompiled with a higher software version, changed and then run again.

When transferring NC programs to a machine with a lower software version, it cannot be guaranteed, however, that the program will be able to be changed by recompiling cycle calls.

9.17 Measuring cycle support

Measuring cycles are general subroutines designed to solve specific measurement tasks. They can be adapted to specific problems via parameter settings.

Software option
You require the "Measuring cycles" option to use "Measuring cycles".

You can find additional information on the use of measuring cycles in the Measuring Cycles Programming Manual.

9.18 Example, standard machining

General

The following example is described in detail as ShopMill program. A G code program is generated in the same way; however, some differences must be observed.

If you copy the G code program listed below, read it into the control and open it in the editor, then you can track the individual program steps.

Machine manufacturer
Under all circumstances, observe the machine manufacturer's instructions.

Tools

The following tools are saved in the tool manager:

Tool name	Tool diameter	Cutting materials	Number of teeth
Face miller	D80 mm	HM	Z = 8
End mill	D20 mm	HM	Z = 3
End mill	D10 mm	HM	Z = 3
End mill	D8 mm	HM	Z = 3
Centering tool (NC spotdrill)	D10 mm	HM	-
Twist drill	D10 mm	HSS	-

The correction (compensation) values for length and radius as well as the tip angle for drills and number of teeth for milling tools should be entered into the tool list. If you are working with ShopMill, in addition, enter the spindle direction of rotation and coolant.

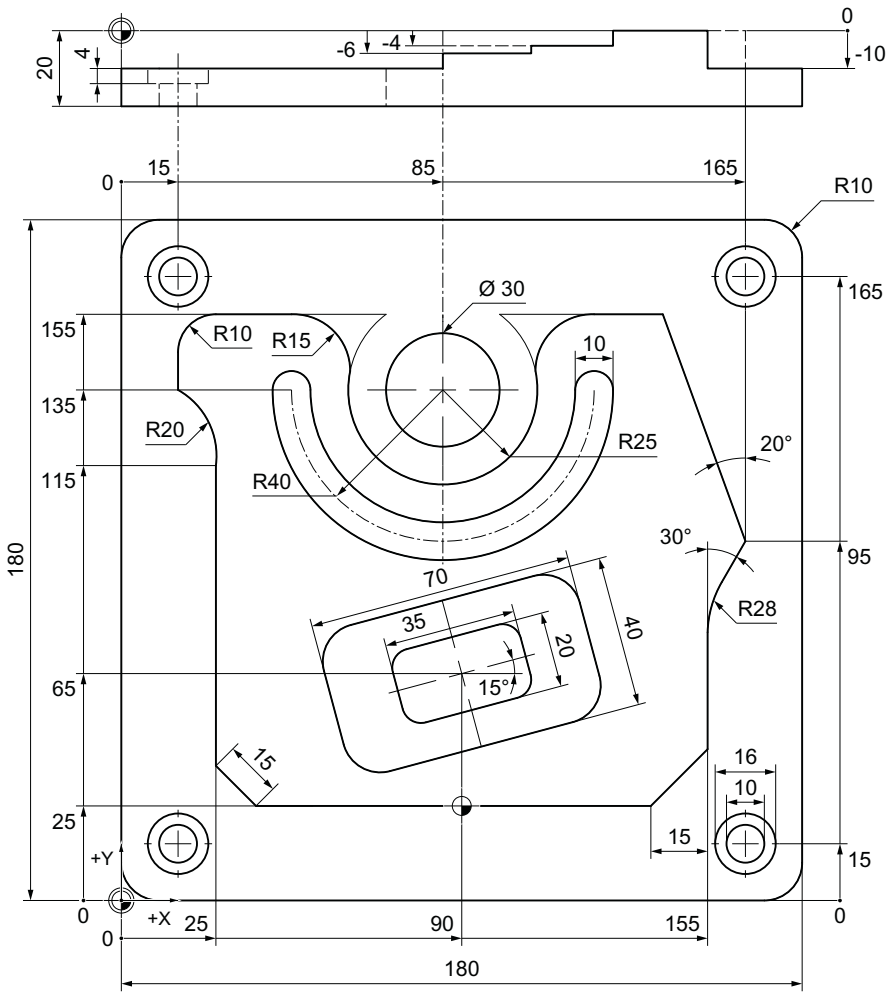
Adapt the cutting data to the tools used and the specific application conditions.

Blank

Dimensions: 185 x 185 x 50

Material: Aluminum

9.18.1 Workpiece drawing



9.18.2 Programming

1. Program header

1. Specify the blank.

Measurement unit mm	
Work offset	G54
Blank	Cuboid
X0	-2.5 abs
Y0	-2.5 abs
X1	182.5 abs

Y1	182.5 abs	
ZA	1 abs	
ZI	-50 abs	
PL	G17 (XY)	Plane selection, if MD 52005 = 0
RP	100	
SC	1	
Machining direction	Climbing	
Retraction position pattern	Optimized	

2. Press the "Accept" softkey.
   The work plan is displayed. Program header and end of program are created as program blocks.
   The end of program is automatically defined.

2. Rectangular spigots, face milling

1. Press the "Milling" and "Face milling" softkeys.

2. Enter the following technology parameters:
   T Facing tool_80mm **D1** **F** 0.10 mm/tooth **V** 750 m/min
3. Enter the following parameters:

Machining	Roughing (∇)
Direction	
X0	-2.5 abs
Y0	-2.5 abs
Z0	1 abs
X1	185 abs
Y1	185 abs
Z1	0 abs
DXY	80 %
DZ	2.0
UZ	0

4. Press the "Accept" softkey.

3. Outside contour of the workpiece

1. Press the "Milling", "Multi-edge spigot" and "Rectangular spigot" softkeys.
2. Enter the following technology parameters:
   T End_mill_20mm **D1** **F** 0.140 mm/tooth **V** 240 m/min
3. Enter the following parameters:

Position of reference point	
Machining	Roughing (∇)
Type of position	Single position
X0	0 abs
Y0	0 abs
Z0	0 abs
W1	185 (fictitious blank dimension)
L1	185 (fictitious blank dimension)
W	180 abs
L	180 abs
R	10 abs
α0	0 degrees
Z1	20 inc
DZ	5
UXY	0
UZ	0

4. Press the "Accept" softkey.

4. Outside contour islands

To simply machine the entire surface outside the island, define a contour pocket around the blank and then program the island. In this way, the entire surface area is machined and no residual material is left behind.

Outside contour of the pocket

1. Press the "Contour milling", "Contour" and "New contour" softkeys.
   The "New Contour" input window opens.

2. Enter the contour name (in this case: Part_4_POCKET).
   The contour calculated as NC code is written as an internal subprogram between a start and an end marker containing the entered name.

3. Press the "Accept" softkey.
   The "Starting Point" input window opens.
4. Enter the starting point of the contour.
   X -10 abs **Y** -1 abs
   0

5. Press the "Accept" softkey.

6. Enter the following contour elements and acknowledge using the "Accept" softkey.

6.1. **X** 190 abs

6.2. **Y** 190 abs

6.3. **X** -10 abs

6.4. Press the ">>" and "Close contour" softkeys, to close the contour.

7. Press the "Accept" softkey.

Outside contour of the island

1. Press the "Contour milling", "Contour" and "New contour" softkeys.
   The "New Contour" input window opens.

2. Enter the contour name (in this case: Part_4_ISLAND).
   The contour calculated as NC code is written as an internal subprogram between a start and an end marker containing the entered name.

3. Press the "Accept" softkey.
   The "Starting Point" input window opens.
4. Enter the starting point of the contour.
   X 90 abs **Y** 25 abs

5. Press the "Accept" softkey.

6. Enter the following contour elements and acknowledge using the "Accept" softkey.

6.1. **X** 25 abs **FS** 15

6.2. **Y** 115 abs **R** 20

6.3. **X** 15 abs **Y** 135 abs

6.4. **Y** 155 abs **R** 10

6.5. **X** 60 abs **R** 15

6.6. **Y** 135 abs **R** 20

7. Direction of rotation ↺

8. **R** 25 **X** 110 abs

9.1 **Y** 155 abs **R** 15

9.2 **R** 0

9.3 **X** 165 abs **Y** 95 abs **α1** 290 degrees **R** 0

9.4 **X** 155 abs **α1** 240 Degrees **R** 28

9.5 **FS** 0

9.6 **X** 140 abs **Y** 25 abs **α1** 225 degrees **R** 0

10. Press the ">>" and "Close contour" softkeys, to close the contour.

11. Press the "Accept" softkey.

Contour milling/solid machining

1. Press the "Contour milling" and "Pocket" softkeys.

2. Enter the following technology parameters:
   T End_mill_20mm **D1** **F** 0.1 mm/tooth **V** 240 m/min
3. Enter the following parameters:

Machining ∇		
Z0	0	abs
Z1	10	inc
DXY	40	%
DZ	3.5	
UXY	0	mm
UZ	0	
Starting point		Auto
Insertion		Helical
EP	1.0	
ER	2.0	
Lift mode		Select, e.g. to the retraction plane

4. Press the "Accept" softkey.

Note

- When selecting the milling tool, please make sure that the tool diameter is large enough to cut the intended pocket. A message will be displayed if you make a mistake.
- If you want to finish cut the pocket, you must assign parameters UXY and UZ accordingly and add a second solid machining cycle for finishing.

5. Milling a rectangular pocket (large)

1. Press the "Milling", "Pocket" and "Rectangular pocket" softkeys.
   The "Rectangular Pocket" input window opens.
2. Enter the following technology parameters:
   T End mill_10mm **D1** **F** 0.04 mm/tooth **V** 260 m/min
3. Enter the following parameters:

Reference point	
Machining	Roughing (∇)
Machining position	Single position
X0	90 abs
Y0	60 abs
Z0	0 abs
W	40
L	70
R	10
α0	15 degrees
Z1	4 inc
DXY	40 %
DZ	4
UXY	0
UZ	0
Insertion	Helical
EP	1
ER	2
Solid machining	Complete machining

4. Press the "Accept" softkey.

6. Milling a rectangular pocket (small)

1. Press the "Milling", "Pocket" and "Rectangular pocket" softkeys.
   The "Rectangular Pocket" input window opens.
2. Enter the following technology parameters:

T End mill_10mm **D1** **F** 0.04 mm/tooth **V** 260 m/min

3. Enter the following parameters:

Reference point	
Machining	Roughing (∇)
Machining position	Single position
X0	90 abs
Y0	60 abs
Z0	-4 abs
W	20
L	35
R	5
α0	15 degrees
Z1	2 inc
DXY	40 %
DZ	2
UXY	0
UZ	0
Insertion	Oscillation
EW	Maximum insertion angle
Solid machining	Complete machining

4. Press the "Accept" softkey.

7. Milling a circumferential slot

1. Press the "Milling", "Groove" and "Circ. groove" softkeys.
   The "Circumferential Groove" input window opens.
2. Enter the following technology parameters:
   T End_mill_8mm **D1** **F** 0.018 mm/tooth **FZ** 0.010 mm/tooth
   V 230 m/min
3. Enter the following parameters:

Machining	Roughing (∇)
Circular pattern	Pitch circle
X0	85 abs
Y0	135 abs
Z0	0 abs
N	1

R	40	
α0	180	degrees
α1	180	degrees
W	10	
Z1	3	inc
DZ	3	
UXY	0	mm

4. Press the "Accept" softkey.

8. Drilling/centering

1. Press the "Drilling" and "Centering" softkeys.
   The "Centering" input window opens.
2. Enter the following technology parameters:

T Centering_tool_10mm	**D1**	**F** 1000 mm/min	**S** 12000 rev/min

3. Enter the following parameters:

Diameter/tip	Diameter
⌀	5
DT	0.6 s

4. Press the "Accept" softkey.

9. Drilling/reaming

1. Press the "Drilling", "Drilling reaming" and "Drilling" softkeys.
   The "Drilling" input window opens.
2. Enter the following technology parameters:

T DRILL10	**D1**	**F** 500 mm/min	**S** 1600 rev/min

3. Enter the following parameters:

Diameter/tip	Tip
Z1	-25 abs
Predrilling	No
Through drilling	No

DT	0

4. Press the "Accept" softkey.

10. Positions

1. Press the "Drilling", "Positions" and "Drilling Positions" softkeys.
   The "Any Positions" input window opens.

2. Enter the following parameters:

	Right-angled
Z0	-10 abs
X0	15 abs
Y0	15 abs
X1	165 abs
Y1	15 abs

3. Press the "Accept" softkey.

11. Obstacle

1. Press the "Drilling", "Positions", and "Obstacle" softkeys.
   The "obstacle" input window opens.

2. Enter the following parameters:

Z	2 abs

3. Press the "Accept" softkey.

Note

If this obstacle cycle is not inserted, the drill will violate the right-hand corner of the island contour. Alternately, you could increase the safety clearance.

12. Positions

1. Press the "Drilling", "Positions" and "Drilling Positions" softkeys.
   The "Any Positions" input window opens.

2. Enter the following parameters:

	Right-angled
Z0	-10 abs
X2	165 abs
Y2	165 abs
X3	15 abs
Y3	165 abs

3. Press the "Accept" softkey.

13. Milling the circular pocket

1. Press the "Milling", "Pocket" and "Circular pocket" softkeys.
   The "Circular Pocket" input window opens.

2. Enter the following technology parameters:
   T End_mill_8mm **D1** **F** 0.018 mm/tooth **V** 230 m/min

3. Enter the following parameters:

Machining	Roughing (∇)
Machining type	Plane-by-plane
Machining position	Single position
X0	85 abs
Y0	135 abs
Z0	-10 abs
Diameter	30
Z1	12 inc
DXY	40 %
DZ	5
UXY	0 mm
UZ	0
Insertion	Helical

EP	1.0
ER	2.0
Solid machining	Complete machining

4. Press the "Accept" softkey.

You also program the four countersinks ∅16 and 4 deep using a circular pocket and repeating positions 2, 3 and 4.

9.18.3 Results/simulation test

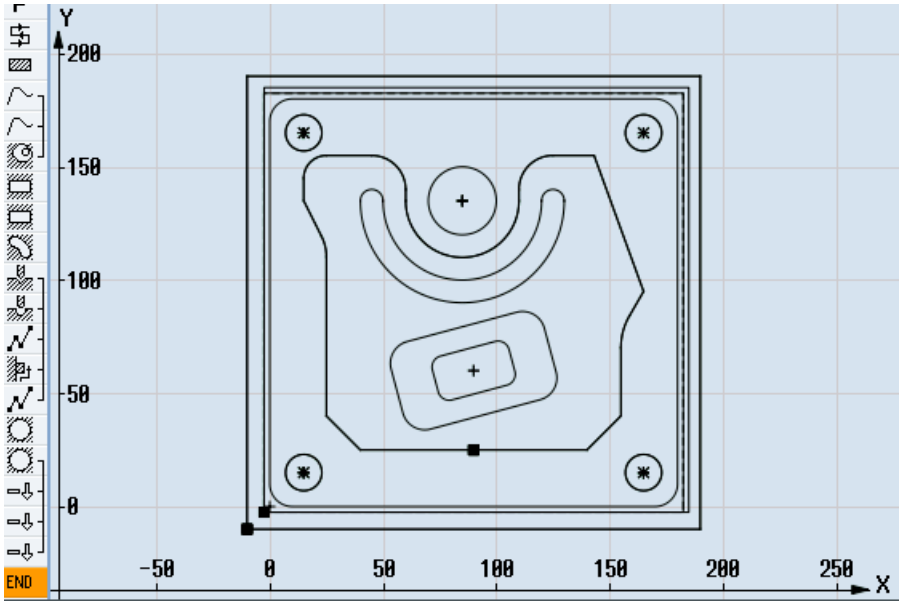


Figure 9-5 Programming graphics

P	Program header		G54 Block
	Face milling	▽	T=PLANFRAESER_80mm F0.1/t V=750m X0=-2.5
	Rectang.spigot	▽	T=Schaftfraeser_20mm F0.14/t V=240m X0=0 Y0=0
	Contour		TEIL_4_TASCHE
	Contour		TEIL_4_INSEL
	Mill pocket	▽	T=Schaftfraeser_20mm F0.1/t V=240m Z0=0
	Rectang.pocket	▽	T=Schaftfraeser_10mm F0.04/t V=260m X0=90
	Rectang.pocket	▽	T=Schaftfraeser_10mm F0.04/t V=260m X0=90
	Circumfer. slot	▽	T=Schaftfraeser_8mm F0.018/t V=230m X0=85
	Centering		T=Zentrierer_10mm F1000/min S=12000rev ∅5
	Drilling		T=Bohrer_10mm F500/min S=1600rev Z1=-25inc
	002: Positions		Z0=-10 X0=15 Y0=15 X1=165 Y1=15
	003: Obstacle		Z=2
	004: Positions		Z0=-10 X0=165 Y0=165 X1=15 Y1=165
	Circular pocket	▽	T=Schaftfraeser_8mm F0.018/t V=230m X0=85
	Circular pocket	▽	T=Schaftfraeser_8mm F0.018/t V=230m Z1=4inc ∅16
	Repeat position		002: Positionen
	Repeat position		003: Hindernis
	Repeat position		004: Positionen
	End of program		

Figure 9-6 Machining schedule

Program test by means of simulation

During simulation, the current program is calculated in its entirety and the result displayed in graphic form.

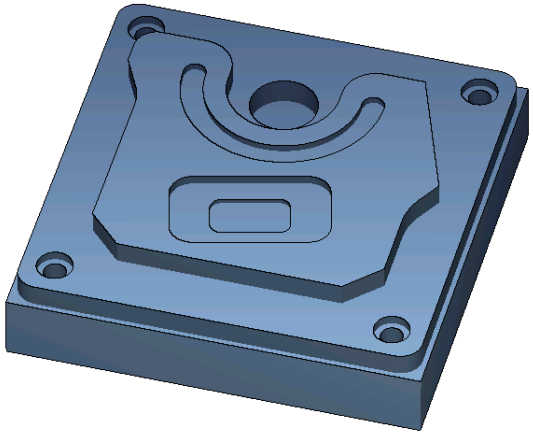


Figure 9-7 3D view

9.18.4 G code machining program

```
G17 G54 G71
WORKPIECE(,,"","BOX",112,1,-20,-100,-2.5,-2.5,182.5,182.5)
;****************Tool change****************
T="FACING TOOL" D1 M6
G95 FZ=0.1 S3000 M3 M8
CYCLE61(50,1,1,0,-2.5,-2.5,185,185,2,80,0,0.1,31,0,1,10)
G0 Z200 M9
;****************Tool change****************
T="MILLER20" D1 M6
G95 FZ=0.14 S3900 M3 M8
CYCLE76(50,0,1,,20,180,180,10,0,0,0,5,0,0,0.14,0.14,0,1,185,185,1,2,2100,1,101)
;CYCLE62(,2,"MA1","MA0")
CYCLE62(,2,"E_LAB_A_PART_4_POCKET","E_LAB_E_PART_4_POCKET")
CYCLE62(,2,"E_LAB_A_PART_4_ISLAND","E_LAB_E_PART_4_ISLAND")
CYCLE63("PART_4_GEN_01",11,50,0,1,10,0.1,0.3,40,3.5,0,0,0,0,0,2,1,15,1,2,,,,0,101,111)
G0 Z200 M9
;****************Tool change****************
T="MILLER10" D1 M6
G95 FZ=0.04 S8500 M3 M8
POCKET3(50,0,1,4,70,40,10,90,60,15,4,0,0,0.04,0.2,0,21,40,8,3,15,2,1,0,1,2,11100,11,111)
POCKET3(50,-4,1,2,35,20,6,90,60,15,2,0,0,0.04,0.2,0,31,40,8,3,15,10,2,0,1,2,11100,11,111)
G0 Z200 M9
;****************Tool change****************
T="MILLER8" D1 M6
G95 FZ=0.018 S9000 M3 M8
SLOT2(50,0,1,,3,1,180,10,85,135,40,180,90,0.01,0.018,3,0,0,2001,0,0,0,,0,1,2,100,1001,101)
G0 Z200 M9
;****************Tool change****************
T="CENTERING TOOL10" D1 M6
G94 F1000 S12000 M3 M8
MCALL CYCLE81(50,-10,1,5,,0,10,1,11)
POS_1: CYCLE802(111111111,111111111,15,15,165,15,165,165,15,165,,,,,,,,,,0,0,1)
MCALL
G0 Z200 M9
;****************Tool change****************
T="DRILL10" D1 M6
G94 F500 S1600 M3 M8
MCALL CYCLE82(50,-10,1,-25,,0,0,1,12)
REPEATB POS_1 ;#SM
MCALL
G0 Z200 M9
;****************Tool change****************
T="MILLER8" D1 M06
```

```
G95 FZ=0.018 S12000 M3 M8
POCKET4(50,-10,1,12,30,85,135,5,0,0,0.018,0.01,0,21,40,9,15,2,1,0,1,2,10100,111,111)
MCALL POCKET4(50,-10,1,4,16,0,0,5,0,0,0.018,0.018,0,11,40,9,15,0,2,0,1,2,10100,111,111)
REPEATB POS_1 ;#SM
MCALL
G0 Z200 M9
;****************Tool change***************
;Contour chamfering
T="CENTERING TOOL10" D1 M6
G94 F500 S8000 M3 M8
CYCLE62(,2,"E_LAB_A_PART_4_ISLAND","E_LAB_E_PART_4_ISLAND")
CYCLE72("",100,0,1,20,2,0.5,0.5,500,100,305,41,1,0,0.1,1,0,0,0.3,2,101,1011,101)
POCKET3(50,0,1,4,70,40,10,90,60,15,4,0,0,500,0.2,0,25,40,8,3,15,2,1,0,0.3,2,11100,11,111)
POCKET3(50,-4,1,2,35,20,6,90,60,15,2,0,0,500,0.2,0,35,40,8,3,15,10,2,0,0.3,2,11100,11,111)
SLOT2(50,0,1,,3,1,180,10,85,135,40,180,90,0.01,500,3,0,0,2005,0,0,0,,0,0.3,2,100,1001,101)
POCKET4(50,-10,1,12,30,85,135,5,0,0,500,0.01,0,15,40,9,15,0,2,0,0.3,2,10100,111,111)
MCALL POCKET4(50,-10,1,4,16,0,0,5,0,0,500,0.025,0,15,40,9,15,0,2,0,0.3,4,10100,111,111)
REPEATB POS_1 ;#SM
MCALL
G0 Z200 M9
M30
;******************Contour*******************
E_LAB_A_PART_4_POCKET: ;#SM Z:5
;#7__DlgK contour definition begin - Don't change!;*GP*;*RO*;*HD*
G17 G90 DIAMOF;*GP*
G0 X-10 Y-10 ;*GP*
G1 X190 ;*GP*
Y190 ;*GP*
X-10 ;*GP*
Y-10 ;*GP*
;CON,0,0.0000,4,4,MST:0,0,AX:X,Y,I,J;*GP*;*RO*;*HD*
;S,EX:-10,EY:-10;*GP*;*RO*;*HD*
;LR,EX:190;*GP*;*RO*;*HD*
;LU,EY:190;*GP*;*RO*;*HD*
;LL,EX:-10;*GP*;*RO*;*HD*
;LA,EX:-10,EY:-10;*GP*;*RO*;*HD*
;#End contour definition end - Don't change!;*GP*;*RO*;*HD*
E_LAB_E_PART_4_POCKET:
;
E_LAB_A_PART_4_ISLAND: ;#SM Z:2
;#7__DlgK contour definition begin - Don't change!;*GP*;*RO*;*HD*
G17 G90 DIAMOF;*GP*
G0 X90 Y25 ;*GP*
G1 X25 CHR=15 ;*GP*
Y115 RND=20 ;*GP*
```

```
X15 Y135 ;*GP*
Y155 RND=10 ;*GP*
X60 RND=15 ;*GP*
Y135 ;*GP*
G3 X110 I=AC(85) J=AC(135) ;*GP*
G1 Y155 RND=15 ;*GP*
X143.162 ;*GP*
X165 Y95 ;*GP*
X155 Y77.679 RND=28 ;*GP*
Y40 ;*GP*
X140 Y25 ;*GP*
X90 ;*GP*
;CON,0,0.0000,14,14,MST:0,0,AX:X,Y,I,J;*GP*;*RO*;*HD*
;S,EX:90,EY:25;*GP*;*RO*;*HD*
;LL,EX:25;*GP*;*RO*;*HD*
;F,LFASE:15;*GP*;*RO*;*HD*
;LU,EY:115;*GP*;*RO*;*HD*
;R,RROUND:20;*GP*;*RO*;*HD*
;LA,EX:15,EY:135;*GP*;*RO*;*HD*
;LU,EY:155;*GP*;*RO*;*HD*
;R,RROUND:10;*GP*;*RO*;*HD*
;LR,EX:60;*GP*;*RO*;*HD*
;R,RROUND:15;*GP*;*RO*;*HD*
;LD,EY:135;*GP*;*RO*;*HD*
;ACCW,EX:110,RAD:25;*GP*;*RO*;*HD*
;LU,EY:155,AT:0;*GP*;*RO*;*HD*
;R,RROUND:15;*GP*;*RO*;*HD*
;LR;*GP*;*RO*;*HD*
;LA,EX:165,EY:95,ASE:290;*GP*;*RO*;*HD*
;LA,EX:155,ASE:240;*GP*;*RO*;*HD*
;R,RROUND:28;*GP*;*RO*;*HD*
;LD;*GP*;*RO*;*HD*
;LA,EX:140,EY:25,ASE:225;*GP*;*RO*;*HD*
;LA,EX:90,EY:25;*GP*;*RO*;*HD*
;#End contour definition end - Don't change!;*GP*;*RO*;*HD*
E_LAB_E_PART_4_ISLAND:
```

Programming technological functions (cycles) 10

10.1 Know-how protection

To protect your technological know-how, you can protect your cycles with individual access rights and additional file encryption.

Implement this cycle protection by means of the following measures:

- Encrypt your cycle data with the additional SIEMENS application SINUCOM Protector. Further information about the SINUCOM Protector can be found here (https://support.industry.siemens.com/cs/ww/en/view/109474775).
- Assign individual access rights to your cycle data and adapt the authorization levels for the user.
  For further information on the individual assignment of access rights, refer to the SINUMERIK Operate Commissioning Manual.

10.2 Drilling

10.2.1 General

General geometry parameters

- Retraction plane RP and reference point Z0
  Normally, reference point Z0 and retraction plane RP have different values. The cycle assumes that the retraction plane is in front of the reference point.

  Note

  If the values for reference point and retraction planes are identical, a relative depth specification is not permitted. Error message "Reference plane defined incorrectly" is output and the cycle is not executed.

  This error message is also output if the retraction plane is located after the reference point, i.e. its distance to the final drilling depth is smaller.

- Safety clearance SC
  Acts in relation to the reference point. The direction in which the safety clearance is active is automatically determined by the cycle.
- Drilling depth
  Depending on the selection of the drill shank or drill tip or the centering diameter, the programmed drilling depth refers to the following for cycles with a selection field:
  - Tip (drilling depth in relation to the tip)
    The drill is inserted into the workpiece until the drill tip reaches the value programmed for Z1.
  - Shank (drilling depth in relation to the shank)
    The drill is inserted into the workpiece until the drill shank reaches the value programmed for Z1. The angle entered in the tool list is taken into account.
  - Diameter (centering in relation to the diameter, only for CYCLE81)
    The diameter of the centering hole is programmed at Z1. In this case, the tip angle of the tool must be specified in the tool list. The drill is inserted into the workpiece until the specified diameter is reached.

Drilling positions

The cycle assumes the tested hole coordinates of the plane.

The hole centers should therefore be programmed before or after the cycle call as follows (see also Section, Cycles on single position or position pattern (MCALL)):

- A single position should be programmed before the cycle call
- Position patterns (MCALL) should be programmed after the cycle call
  - as drilling pattern cycle (line, circle, etc.) or
  - as a sequence of positioning blocks for the hole centers

10.2.2 Centering (CYCLE81)

Function

With the "Centering" function, the tool drills with the programmed spindle speed and feedrate either:

- Down to the programmed final drilling depth or
- So deep until the programmed diameter of the centering is reached

The tool is retracted after a programmed dwell time has elapsed.

Approach/retraction

1. The tool moves with G0 to safety clearance of the reference point.
2. Inserted into the workpiece with G1 and the programmed feedrate F until the depth or the centering diameter is reached.
3. On expiry of a dwell time DT, the tool is retracted at rapid traverse G0 to the retraction plane.

Procedure

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

2. Press the "Drilling" softkey.

3. Press the "Center" softkey.
   The "Centering" input window opens.

G code program parameters				ShopMill program parameters		
PL	Machining plane			T	Tool name	
RP	Retraction plane	mm		D	Cutting edge number	
SC	Safety clearance	mm		F	Feedrate	mm/min mm/rev
				S / V	Spindle speed or constant cutting rate **Note:** Please observe the information provided by your machine manufacturer.	rpm m/min

Parameter	Description	Unit
Machining position (only for G code)	• Single position Drill hole at programmed position • Position pattern Position with MCALL	
Z0 (only for G code)	Reference point Z	mm
Centering	• Diameter (centered with reference to the diameter) The angle for the center drill entered in the tool list is applied. • Tip (centered with reference to the depth) The drill is inserted into the workpiece until the programmed insertion depth is reached.	
∅	It is inserted into the workpiece until the diameter is correct. - (for diameter centering only)	mm
Z1	Drilling depth (abs) or drilling depth in relation to Z0 (inc) It is inserted into the workpiece until it reaches Z1. - (for tip centering only)	mm
DT	• Dwell time (at final drilling depth) in seconds • Dwell time (at final drilling depth) in revolutions	s rev
Predrilling	• Yes • No	
ZA	Predrilling depth (abs) or predrilling depth referred to the reference point (inc) - ("yes", only for predrilling)	mm
FA	Predrilling feedrate - ("yes", only for predrilling)	% F7min F/U

10.2.3 Drilling (CYCLE82)

Function

With the "Drilling" function, the tool drills with the programmed spindle speed and feedrate down to the specified final drilling depth (shank or tip).

The tool is retracted after a programmed dwell time has elapsed.

Input simple

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

Machine manufacturer

Various defined values can be pre-assigned using setting data.

Please observe the information provided by the machine manufacturer.

If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

Approach/retraction

1. The tool moves with G0 to safety clearance of the reference point.
2. The tool is inserted into the workpiece with G1 and the programmed feedrate F until it reaches the programmed final depth Z1.
3. When a dwell time DT expires, the tool is retracted at rapid traverse G0 to the retraction plane.

Procedure

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

2. Press the "Drilling" softkey.

3. Press the "Drilling Reaming" softkey.

4. Press the "Drilling" softkey.
   The "Drilling" input window opens.

Parameters in the "Input complete" mode

G code program parameters				ShopMill program parameters		
Input		• complete				
PL	Machining plane			T	Tool name	
RP	Retraction plane	mm		D	Cutting edge number	
SC	Safety clearance	mm		F	Feedrate	mm/min mm/rev
				S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
Machining position (only for G code)	• Single position Drill hole at programmed position • Position pattern Position with MCALL	
Z0 (only for G code)	Reference point Z	mm
Drilling depth	• Shank (drilling depth in relation to the shank) The drill is inserted into the workpiece until the drill shank reaches the value programmed for Z1. The angle entered in the tool list is taken into account. • Tip (drilling depth in relation to the tip) The drill is inserted into the workpiece until the drill tip reaches the value programmed for Z1.	
Z1	Drilling depth (abs) or drilling depth in relation to Z0 (inc) It is inserted into the workpiece until it reaches Z1.	mm
Predrilling	• Yes • No	
ZA - (only for predrilling "yes")	Predrilling depth (abs) or predrilling depth in relation to the reference point (inc)	mm
FA - (only for predrilling "yes")	Reduced predrilling feedrate as a percentage of the drilling feedrate	mm/min %
	Predrilling feedrate (ShopMill)	mm/min or mm/rev.
	Predrilling feedrate (G code)	Distance/min or distance/rev
Through drilling	• Yes Through drilling with feedrate FD • No	
ZD - (only for through drilling "yes")	Depth for feedrate reduction (abs) or depth for feedrate reduction in relation to Z1 (inc)	mm
FD - (only for through drilling "yes")	Reduced feedrate for through drilling referred to drilling feedrate F	%
	Feedrate for through drilling (ShopTurn)	mm/min or mm/rev.
	Feedrate for through drilling (G code)	Distance/min or distance/rev
DT - (only for through drilling "no")	• Dwell time at final depth in seconds • Dwell time at final depth in revolutions	s rev

Parameters in the "Input simple" mode

G code program parameters				ShopMill program parameters		
Input		• simple				
RP	Retraction plane	mm		T	Tool name	
				D	Cutting edge number	
				F	Feedrate	mm/min mm/rev
				S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	
Machining position (only for G code)	• Single position Drill hole at programmed position • Position pattern Position with MCALL	
Z0 (only for G code)	Reference point Z	mm
Drilling depth	• Shank (drilling depth in relation to the shank) The drill is inserted into the workpiece until the drill shank reaches the value programmed for Z1. The angle entered in the tool list is taken into account. • Tip (drilling depth in relation to the tip) The drill is inserted into the workpiece until the drill tip reaches the value programmed for Z1.	
Z1	Drilling depth (abs) or drilling depth in relation to Z0 (inc) It is inserted into the workpiece until it reaches Z1.	mm
DT	• Dwell time (at final drilling depth) in seconds • Dwell time (at final drilling depth) in revolutions	s rev

Hidden parameters

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.

Parameter	Description	Value	Can be set in SD
PL (only for G code)	Machining plane	Defined in MD 52005	
SC (only for G code)	Safety clearance	1 mm	x
Predrilling			
ZA	Predrilling depth		
FA	Reduced predrilling feedrate		
Through drilling			
ZD	Depth for reduced feedrate		
FD	Reduced through drilling feedrate		

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

10.2.4 Reaming (CYCLE85)

Function

With the "Reaming" cycle, the tool is inserted in the workpiece with the programmed spindle speed and the feedrate programmed at F.

If Z1 has been reached and the dwell time expired, the reamer is retracted at the programmed retraction feedrate to the retraction plane.

Approach/retraction

1. The tool moves with G0 to safety clearance of the reference point.
2. The tool is inserted into the workpiece with the programmed feedrate F until it reaches the final depth Z1.
3. Dwell time DT at final drilling depth.
4. Retraction to retraction plane with programmed retraction feedrate FR.

Procedure

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

2. Press the "Drilling" softkey.

3. Press the "Drilling Reaming" softkey.

4. Press the "Reaming" softkey
   The "Reaming" input window opens.

Parameters, G code program				Parameters, ShopMill program		
PL	Machining plane			T	Tool name	
RP	Retraction plane	mm		D	Cutting edge number	
SC	Safety clearance	mm		F	Feedrate	mm/min mm/rev
F	Feedrate	*		S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
Machining position (only for G code)	• Single position Drill hole at programmed position • Position pattern Position with MCALL	
Z0 (only for G code)	Reference point Z	mm
FR (only for G code)	Feedrate during retraction	*
FR (only for ShopMill)	Feedrate during retraction	mm/min mm/rev
Z1	Drilling depth (abs) or drilling depth in relation to Z0 (inc) It is inserted into the workpiece until it reaches Z1. - (for tip centering only)	mm
DT	• Dwell time (at final drilling depth) in seconds • Dwell time (at final drilling depth) in revolutions	s rev

* Unit of feedrate as programmed before the cycle call

10.2.5 Deep-hole drilling 1 (CYCLE83)

Function

With the "Deep-hole drilling 1" cycle, the tool is inserted in the workpiece with the programmed spindle speed and feedrate in several infeed steps until the depth Z1 is reached. You have the option of entering the following infeed steps.

- Number of infeed steps constant or decreasing (via programmable degression factor)
- Chip breaking without lifting or swarf removal with tool retraction
- Feedrate factor for 1st infeed to reduce the feedrate or increase the feedrate (e.g. if a hole has already be predrilled)
- Dwell times
- Depth in relation to drill shank of drill tip

Input simple

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

Machine manufacturer
Various defined values can be pre-assigned using setting data.
Please refer to the machine manufacturer's specifications.

If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

Approach/retraction during chip breaking

1. The tool moves with G0 to safety clearance of the reference point.
2. The tool drills with the programmed spindle speed and feedrate F = F · FD1 [%] up to the first infeed depth.
3. Dwell time at drilling depth DTB.
4. The tool is retracted by retraction distance V2 for chip breaking and drills up to the next infeed depth with programmed feedrate F.
5. Step 4 is repeated until the final drilling depth Z1 is reached.
6. Dwell time at final drilling depth DT.
7. The tool retracts to the retraction plane at rapid traverse.

Approach/retraction during stock removal

1. The tool moves with G0 to safety clearance of the reference point.
2. The tool drills with the programmed spindle speed and feedrate F = F · FD1 [%] up to the first infeed depth.
3. Dwell time at drilling depth DTB.
4. The tool retracts from the workpiece for the stock removal with rapid traverse to the safety clearance.
5. Dwell time at starting point DTS.
6. Approach of the last drilling depth with G0, reduced by the clearance distance V3.
7. Drilling is then continued to the next drilling depth.
8. Steps 4 to 7 are repeated until the programmed final drilling depth Z1 is reached.
9. Dwell time at final drilling depth.
10. The tool retracts to the retraction plane at rapid traverse.

Procedure

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

2. Press the "Drilling" softkey.

3. Press the "Deep-hole drilling" and "Deep-hole drilling 1" softkeys.

The "Deep-hole Drilling 1" input window opens.

Parameters in the "Input complete" mode

G code program parameters				ShopMill program parameters		
Input		• Complete				
PL	Machining plane			T	Tool name	
RP	Retraction plane	mm		D	Cutting edge number	
SC	Safety clearance	mm		F	Linear feedrate Revolutional feedrate	mm/min mm/rev
				S / V	Spindle speed / Constant cutting rate	rpm m/min

Parameter	Description	Unit
Machining position (only G code)	• Single position Drill hole at programmed position. • Position pattern (MCALL) Position with MCALL	
Machining	• Swarf removal The drill is retracted from the workpiece for swarf removal. • Chip breaking The drill is retracted by the retraction distance V2 for chip breaking.	
Z0 (only G code)	Reference point Z	mm
Drilling depth	• Shank (drilling depth in relation to the shank) The drill is inserted into the workpiece until the drill shank reaches the value programmed for Z1. The angle entered in the tool list is taken into account. • Tip (drilling depth in relation to the tip) The drill is inserted into the workpiece until the drill tip reaches the value programmed for Z1.	
Z1	Final drilling depth (abs) or final drilling depth in relation to Z0 (inc) It is inserted into the workpiece until it reaches Z1.	mm
FD1	Percentage for the feedrate at the first infeed.	%
D - (only for G code)	First drilling depth (abs) or first drilling depth in relation to Z0 (inc)	mm
D - (only ShopMill)	Maximum depth infeed	mm
DF	Infeed: • Degression amount by which each additional infeed is reduced (inc) • Percentage for each additional infeed. DF = 100%: Infeed increment remains constant. DF < 100%: Infeed increment is reduced in direction of final drilling depth. **Example:** Last infeed was 4 mm; DF is 80% Next infeed = 4 x 80% = 3.2 mm Next infeed = 3.2 x 80% = 2.56 mm, etc.	mm %

Parameter	Description	Unit
V1	Minimum depth infeed - (only for DF in %) Parameter V1 is only provided if DF<100 has been programmed. If the infeed increment becomes very small, a minimum infeed can be programmed in parameter "V1". V1 < Infeed increment: The tool is inserted by the infeed increment. V1 > Infeed increment: The tool is inserted by the infeed value programmed under V1.	mm
V2	Retraction distance after each machining step – (for chip breaking only). Distance by which the drill is retracted for chip breaking. V2 = 0: The tool is not retracted but is left in place for one revolution.	mm
Clearance distance (for swarf removal only)	• Manual The clearance distance must be entered manually. • Automatic The clearance distance is calculated by the cycle.	
V3 (only for stock removal and "manual" clearance distance)	Clearance distance	mm
DTB - (only G code)	• Dwell time at drilling depth in seconds • Dwell time at drilling depth in revolutions	s rev
DT	• Dwell time at final drilling depth in seconds • Dwell time at final drilling depth in revolutions	s rev
DTS - (only for stock removal) (only G code)	• Dwell time for swarf removal in seconds • Dwell time for swarf removal in revolutions	s rev

Parameters in the "Input simple" mode

G code program parameters				ShopMill program parameters		
Input		• simple				
RP	Retraction plane	mm		T	Tool name	
				D	Cutting edge number	
				F	Feedrate	mm/min mm/rev
				S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	
Machining position 	• Single position Drill hole at programmed position. • Position pattern Position with MCALL	
Machining 	• Swarf removal The drill is retracted from the workpiece for swarf removal. • Chip breaking The drill is retracted by the retraction distance V2 for chip breaking.	
Z0 (only for G code)	Reference point Z	mm
Z1 	Drilling depth (abs) or drilling depth in relation to Z0 (inc) It is inserted into the workpiece until it reaches Z1.	
D - (only for G code)	First drilling depth (abs) or first drilling depth in relation to Z0 (inc)	mm
D - (only for Shop-Mill)	Maximum depth infeed	mm

Hidden parameters

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.

Parameter	Description	Value	Can be set in SD
PL (only for G code)	Machining plane	Defined in MD 52005	
SC (only for G code)	Safety clearance	1 mm	x
Drilling depth	Drilling depth in relation to the tip	Tip	
FD1	Percentage for the feedrate for the first infeed	90 %	x
DF	Percentage for each additional infeed	90 %	x
V1	Minimum infeed	1.2 mm	x
V2	Retraction distance after each machining step	1.4 mm	x
Clearance distance	The clearance distance is calculated by the cycle	Automatic	
DBT	Dwell time at drilling depth	0.6 s	x
DT	Dwell time at final drilling depth	0.6 s	x
DTS (only for G code)	Dwell time for swarf removal (for swarf removal only)	0.6 s	x

Machine manufacturer

Please refer to the machine manufacturer's specifications.

10.2.6 Deep-hole drilling 2 (CYCLE830)

Function

The cycle "Deep-hole drilling 2" covers the complete functionality of "Deep-hole drilling 1".

in addition, the cycle provides the following functions:

- Predrilling with reduced feedrate
- Taking into account a pilot hole
- Soft first cut when entering the material
- Drilling to the final depth in one cut
- Through drilling with reduced feedrate
- Control for switching-in and switching-out the coolant

Input simple

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

Machine manufacturer

Various defined values can be pre-assigned using setting data.

Please refer to the machine manufacturer's specifications.

If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

Approach/retraction during chipbreaking

1. The tool moves with G0 to safety clearance of the reference point.
2. The tool drills with the programmed spindle speed and feedrate F = F · FD1 [%] up to the 1st infeed depth.
3. Dwell time at drilling depth DTB.
4. The tool is retracted by retraction distance V2 for chipbreaking and drills up to the next infeed depth with programmed feedrate F.
5. Step 4 is repeated until the final drilling depth Z1 is reached.
6. Dwell time at final drilling depth DT.
7. The tool retracts to the retraction plane at rapid traverse.

Approach/retraction during stock removal

1. The tool moves with G0 to safety clearance of the reference point.
2. The tool drills with the programmed spindle speed and feedrate F = F · FD1 [%] up to the 1st infeed depth.
3. Dwell time at drilling depth DTB.
4. The tool retracts from the workpiece for the stock removal with rapid traverse to the safety clearance.
5. Dwell time at starting point DTS.
6. Approach of the last drilling depth with G0, reduced by the clearance distance V3.
7. Drilling is then continued to the next drilling depth.
8. Steps 4 to 7 are repeated until the programmed final drilling depth Z1 is reached.
9. The tool retracts to the retraction plane at rapid traverse.

Deep-hole drilling at the entrance to the hole

The following versions are available for deep-hole drilling 2:

- Deep-hole drilling with/without predrilling
- Deep-hole drilling with pilot hole

Note

Predrilling or pilot hole mutually exclude one another.

Predrilling

For predrilling, the reduced feedrate (FA) is used up to the predrilling depth (ZA) and then the drilling feedrate is used. For drilling with several infeed steps, the predrilling depth must be between the reference point and the 1st drilling depth.

Through drilling

For a through-hole, starting from the remaining drilling depth (ZD), a reduced feedrate (FD) is used.

Pilot hole

The cycle optionally takes into account the depth of a pilot hole. This can be programmed with abs/inc – or a multiple of the hole diameter (1.5 to 5*D is typical, for example) – and is assumed that it is available.

If a pilot hole exists, the 1st drilling depth must be between the pilot hole and the final drilling depth. The tool enters the pilot hole with reduced feedrate and reduced speed; these values can be programmed.

Direction of spindle rotation

The direction of spindle rotation with which the tool enters and withdraws from the pilot hole can be programmed as follows:

- with stationary spindle
- with clockwise rotating spindle
- with counterclockwise rotating spindle

This avoids long or thin drills from being broken.

Horizontal drilling

For horizontal drilling using spiral drills, entering the pilot hole is improved if the cutting edges of the drill are also in the horizontal position. To support this, the alignment of the drill in the spindle can be programmed for a specific position (SPOS).

The feedrate is stopped before reaching the pilot hole depth, the speed increased to the drilling speed and the coolant switched in.

Soft first cut into the material

The entry into the material can be influenced, depending on the tool and the material.

The soft first cut comprises two partial distances:

- The first cut feedrate is maintained to a programmable first feed distance ZS1.
- An additional programmable feed distance ZS2 immediately following ZS1 is used to continuously increase the first cut feedrate (with FLIN) to the drilling feedrate.

With chip breaking / swarf removal, this mechanism takes effect at each infeed.

The input parameters ZS1 and ZS2 are maximum values that are limited by the cycle to the infeed depth to be executed.

Deep-hole drilling at the exit from the hole

It makes sense to reduce the feedrate when for through drilling the exit is inclined with respect to the tool axis.

- Through drilling "no"
  The machining feedrate is used when drilling to the final drilling depth. You then have the option of programming a dwell time at the drilling depth.
- Through drilling "yes"
  Up to the remaining drilling depth, you program drilling with the drilling feedrate and, from that point onward, you program drilling with a special feedrate FD.

Retraction

Retraction can be realized at the pilot hole depth or the retraction plane.

- Retraction to the retraction plane is realized with G0 or feedrate, programmable speed as well as direction of rotation respectively stationary spindle.
- For retraction at the pilot hole depth, subsequent retraction and insertion are performed with the same data.

Note

Direction of spindle rotation

The direction of spindle rotation is not reversed; however, where necessary, can be stopped.

Coolant

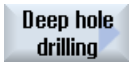
The technology and tools require that also in the G code, the control for the coolant is supported.

- Coolant on
  Switch on at Z0 + safety clearance or at the pilot hole depth (if a pilot hole is being used)
- Coolant off
  Always switch off at the final drilling depth
- Programming in the G code
  An executable block (M command or subprogram call), which can be programmed as string.

Procedure

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

2. Press the "Drilling" softkey.



3. Press the "Deep-hole drilling" and "Deep-hole drilling 2" softkeys.

The "Deep-hole Drilling 2" input window opens.

Parameters in the "Input complete" mode

G code program parameters				ShopMill program parameters		
Input		• Complete				
PL	Machining plane					

G code program parameters					ShopMill program parameters		
RP	Retraction plane		mm		T	Tool name	
SC	Safety clearance		mm		D	Cutting edge number	
F	Feedrate		distance/min distance/rev		F	Linear feedrate Revolutional feedrate	mm/min mm/rev
S / V		Direction of spindle rotation during drilling			S / V	Spindle speed / Constant cutting rate	rpm m/min
	Spindle speed / Constant cutting rate		rpm distance/min				

Parameter	Description			Unit
Machining position (only for G code)	• Single position Drill hole at programmed position • Position pattern with MCALL			
Z0 (only G code)	Reference point Z			mm
Drilling depth	• Shank (drilling depth in relation to the shank) The drill is inserted into the workpiece until the drill shank reaches the value programmed for Z1. The angle entered in the tool list is taken into account. • Tip (drilling depth in relation to the tip) The drill is inserted into the workpiece until the drill tip reaches the value programmed for Z1.			
Z1	Final drilling depth (abs) or final drilling depth in relation to Z0 (inc) It is inserted into the workpiece until it reaches Z1.			mm
Coolant on - (only G code)	M function to switch on the coolant.			
Technology at the entrance to the hole	Selecting the drilling feedrate • Without predrilling Drilling with feedrate F • With predrilling Drilling with feedrate FA • With pilot hole Insertion in the pilot hole with feedrate FP.			
ZP - (only for pilot hole)	Depth of the pilot hole as a factor of the bore diameter Depth of the pilot hole in relation to Z0 (inc) or depth of the pilot hole (abs)			* Ø mm
ZPV - (only for pilot hole)	Clearance distance of pilot hole			mm
FP - (only for pilot hole)	First cut feedrate as a percentage of the drilling feedrate			%
	First cut feedrate (ShopMill)			mm/rev or mm/min
	First cut feedrate (G code)			distance/min or distance/rev
SP - (only for pilot hole)		Approach with stationary spindle	Spindle position during approach	Degrees

Parameter	Description			Unit
SP / VP (only for pilot hole)		Direction of spindle rotation during approach	Spindle speed during approach as a percentage of the drilling speed	%
			Spindle speed during approach	rpm
			Constant cutting speed during approach (G code)	distance/min
			Constant cutting speed during approach (ShopMill)	m/min
ZA - (only for predrilling)	Predrilling depth (abs) or predrilling depth in relation to Z0 (inc)			mm
FA - (only for predrilling)	Predrilling feedrate as a percentage of the drilling feedrate			%
	Predrilling feedrate (ShopMill)			mm/min or mm/rev.
	Predrilling feedrate (G code)			distance/min or distance/rev
Soft first cut	• Yes Soft first cut with feedrate FS • No First cut with drilling feedrate			
ZS1 (only "Yes" for soft first cut)	Depth of each first cut with constant first cut feedrate FS (inc)			mm
FS (only "Yes" for soft first cut)	First cut feedrate as a percentage of the drilling feedrate			%
	First cut feedrate (ShopMill)			mm/min or mm/rev.
	First cut feedrate (G code)			distance/min or distance/rev
ZS2 (only "Yes" for soft first cut)	Depth of each first cut for feedrate increase (inc)			mm
Machining	• One cut • Chip breaking • Swarf removal • Chip breaking and swarf removal			
FD1	Percentage for the feedrate for the first infeed			%
D	First drilling depth (abs) or first drilling depth in relation to Z0 (inc)			mm
DF	Infeed: • Degression amount by which each additional infeed is reduced (inc) • Percentage for each additional infeed. DF = 100%: Infeed increment remains constant. DF < 100%: Infeed increment is reduced in direction of final drilling depth. **Example:** Last infeed was 4 mm; DF is 80% Next infeed = 4 x 80% = 3.2 mm Next infeed = 3.2 x 80% = 2.56 mm, etc.			mm %

Parameter	Description	Unit
V1	Minimum depth infeed - (only for DF in %) Parameter V1 is only provided if DF<100 has been programmed. If the infeed increment becomes very small, a minimum infeed can be programmed in parameter "V1". V1 < Infeed increment: The tool is inserted by the infeed increment V1 > Infeed increment: The tool is inserted by the infeed value programmed under V1.	mm
V2 (for chip breaking only)	Retraction distance after each machining step Distance by which the drill is retracted for chip breaking. V2 = 0: The tool is not retracted but is left in place for one revolution.	mm
DTB	• Dwell time at drilling depth in seconds • Dwell time at drilling depth in revolutions	s rev
Clearance distance - (for stock removal only)	• Manual The clearance distance must be entered manually. • Automatic The clearance distance is calculated by the cycle.	
V3 – (for "manual" clearance distance only)	Clearance distance (inc)	mm
N - (only for "chip breaking and swarf removal")	Number of chip breaking strokes before each chip removal	
Retraction for swarf removal	• Swarf removal at the pilot hole depth • Swarf removal at the safety clearance	
DTS	• Dwell time for swarf removal in seconds • Dwell time for swarf removal in revolutions	s rev
Through drilling	• Yes Through drilling with feedrate FD • No Drilling with constant feedrate	
ZD - (only for through drilling "yes")	Depth for feedrate reduction (abs) or depth for feedrate reduction in relation to Z1 (inc)	mm
FD - (only for through drilling "yes")	Feedrate for through drilling referred to drilling feedrate F.	%
	Feedrate for through drilling (ShopMill)	mm/min or mm/rev.
	Feedrate for through drilling (G code).	distance/min or distance/rev
DT - (only for through drilling "no")	• Dwell time at final depth in seconds • Dwell time at final depth in revolutions	s U
Retraction	• Retraction to pilot hole depth • Retraction to retraction plane	

Parameter	Description			Unit
FR	Retraction (G code) • with rapid traverse • Feedrate for retraction			distance/min
	Retraction (ShopMill) • with rapid traverse • Feedrate for retraction			mm/min
SR / VR		Retraction with stationary spindle		
	Direction of spindle rotation during retraction		Spindle speed for retraction to the drilling speed	%
			Spindle speed for retraction	rpm
			Constant cutting rate for retraction (G code)	distance/min
			Constant cutting rate for retraction (ShopMill)	m/min
Coolant off - (only G code)	M9 - M function to switch off the coolant			

Parameters in the "Input simple" mode

G code program parameters				ShopMill program parameters		
Input		• simple				
RP	Retraction plane	mm		T	Tool name	
				D	Cutting edge number	
F	Feedrate	distance/min distance/rev		F	Linear feedrate Revolutional feedrate	mm/min mm/rev
S / V				S / V	Spindle speed Constant cutting rate	rpm m/min
	Spindle speed Constant cutting rate	rpm distance/min				

Parameter	Description	Unit
Machining position (only for G code)	• Single position Drill hole at programmed position. • Position pattern with MCALL	
Z0 (only for G code)	Reference point Z	
Z1	Final drilling depth (abs) or final drilling depth in relation to Z0 (inc) It is inserted into the workpiece until it reaches Z1.	mm
Coolant on - (only G code)	M8 M function to switch on the coolant	

Parameter	Description			Unit
Technology at the entrance to the hole	Selecting the drilling feedrate • With pilot hole Insertion in the pilot hole with feedrate FP.			
ZP	Depth of the pilot hole as a factor of the bore diameter			* Ø
	Depth of the pilot hole in relation to Z0 (inc) or depth of the pilot hole (abs)			mm
ZPV	Clearance distance of pilot hole			mm
FP	First cut feedrate as a percentage of the drilling feedrate			%
	First cut feedrate (ShopMill)			mm/rev or mm/min
	First cut feedrate (G code)			distance/min or distance/rev
SP		Approach with stationary spindle	Spindle position during approach	Degrees
SP / VP		Direction of spindle rotation during approach	Spindle speed during approach as a percentage of the drilling speed	%
			Spindle speed during approach	rpm
			Constant cutting speed during approach	distance/min
SP / VP				
Soft first cut	• Yes Soft first cut with feedrate FS • No First cut with drilling feedrate			
ZS1 - (only "Yes" for soft first cut)	First cut depth (abs) or depth of the first cut with constant first cut feedrate FS (inc)			mm
FS	First cut feedrate as a percentage of the drilling feedrate			%
	First cut feedrate (G code)			distance/min or distance/rev
	First cut feedrate (ShopMill)			mm/min or mm/rev.
ZS2 - (only "Yes" for soft first cut)	First cut depth (abs) or depth of each cut for the feedrate increase (ink)			mm
Through drilling	• Yes Through drilling with feedrate FD • No			
ZD - (only for through drilling "yes")	Depth for feedrate reduction (abs) or depth for feedrate reduction in relation to Z1 (inc)			mm

Parameter	Description	Unit
FD - (only for through drilling "yes")	Feedrate for through drilling as a percentage of the drilling feedrate	%
	Feedrate for through drilling (G code).	distance/min or distance/rev
	Feedrate for through drilling (ShopMill)	mm/min or mm/rev.
Coolant off - (only G code)	M9 M function to switch off the coolant	

Hidden parameters

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.

Parameter	Description	Value	Can be set in SD
PL (only for G code)	Machining plane	Defined in MD 52005	
SC (only for G code)	Safety clearance	1 mm	x
Drilling depth	Drilling depth referred to the shaft or tip	Tip	
Entrance to the hole	Technology at the entrance to the hole	With pilot hole	
ZA	Predrilling depth (inc)	1mm	
FA	Predrilling feed	50 %	
Drilling interruption	• One cut • Chip breaking • Swarf removal • Chip breaking and swarf removal		
D	1. Drilling depth referred to Z0 (inc.)	10 mm	
FD1	Percentage for the feedrate for the first infeed		
DF	Percentage for the feedrate for each additional infeed Infeed increment is continually reduced in the direction of final drilling depth	90 %	
V1	Minimum infeed V1 < Infeed increment: The tool is inserted by the infeed increment V1 > Infeed increment: The tool is inserted by the infeed value programmed under V1.	2 mm	
V2	Retraction distance after each machining step	1 mm	
Clearance distance	The clearance distance is calculated by the cycle.	Automatic	
DTB	Dwell time at each drilling depth	0.6 s	
N - (only for "chip breaking and swarf removal")	Number of chipbreaking strokes before each swarf removal operation	1	
Retraction for swarf removal	Swarf removal at the pilot hole depth or safety clearance	Safety clearance	
DTS	Dwell time for swarf removal in seconds	0.6 s	

Parameter	Description	Value	Can be set in SD
DT - (only for through drilling "no")	Dwell time at final depth in seconds	0.6 s	
Retraction	Retraction to pilot hole depth or retraction plane	Pilot hole depth	
FR	Retraction in rapid traverse		
Direction of spindle rotation during retraction		M5	
SR (only for selected spindle direction of rotation)	Spindle speed for retraction referred to the drilling speed	10 %	

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

10.2.7 Boring (CYCLE86)

Function

With the "Boring" cycle, the tool approaches the programmed position in rapid traverse, allowing for the retraction plane and safety clearance. It is then inserted into the workpiece at the feedrate programmed under F until it reaches the programmed depth (Z1). There is an oriented spindle stop with the SPOS command. After the dwell time has elapsed, the tool is retracted either with or without lift of the tool.

Note

If, for example, swiveling of mirroring has been performed with CYCLE800 before machining, the SPOS command must be adapted so that the spindle position acts synchronously with DX and DY.

Lift

When lifting, define the amount of lift D and the tool orientation angle α.

Note

The "Boring" cycle can be used if the spindle to be used for the boring operation is technically able to go into position-controlled spindle operation.

Approach/retraction

1. The tool moves with G0 to safety clearance of the reference point.
2. Travel to the final drilling depth with G1 and the speed and feedrate programmed before the cycle call.
3. Dwell time at final drilling depth.
4. Oriented spindle hold at the spindle position programmed under SPOS.
5. With the "Lift" selection, the cutting edge retracts from the hole edge with G0 in up to three axes.
6. Retraction with G0 to the safety clearance of the reference point.
7. Retraction to retraction plane with G0 to drilling position in the two axes of the plane (coordinates of the hole center point).

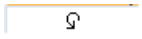
Procedure

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

2. Press the "Drilling" softkey.

3. Press the "Boring" softkey.
   The "Boring" input window opens.

Parameters, G code program				Parameters, ShopMill program		
PL	Machining plane			T	Tool name	
RP	Retraction plane	mm		D	Cutting edge number	
SC	Safety clearance	mm		F	Feedrate	mm/min mm/rev
				S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
Machining position (only for G code)	• Single position Drill hole at programmed position • Position pattern Position with MCALL	
Z0 (only for G code)	Reference point Z	mm
DIR (only for G code)	Direction of rotation •  •	
Z1	Drilling depth (abs) or drilling depth in relation to Z0 (inc)	mm
DT	• Dwell time at final drilling depth in seconds • Dwell time at final drilling depth in revolutions	s rev

Parameter	Description	Unit
SPOS	Spindle stop position	Degrees
Lift mode 	• Do not lift off contour The cutting edge is not fully retracted, but traverses back to the retraction plane. • Lift The cutting edge retracts from the edge of the hole and then retracts to the safety clearance from the reference point and then positions at the retraction plane and hole center point.	
DX (only G Code)	Retraction distance in the X direction (incremental) - (for lift only)	mm
DY (only G code)	Retraction distance in the Y direction (incremental) - (for lift only)	mm
DZ (only G code)	Retraction distance in the Z direction (incremental) - (for lift only)	mm
D (only ShopMill)	Retraction distance (incremental) - (for lift only)	mm

10.2.8 Tapping (CYCLE84, 840)

Function

You can machine an internal thread with the "tapping" cycle.

The tool moves to the safety clearance with the active speed and rapid traverse. The spindle stops, spindle and feedrate are synchronized. The tool is then inserted in the workpiece with the programmed speed (dependent on %S).

You can choose between drilling in one cut, chip breaking or retraction from the workpiece for swarf removal.

Depending on the selection in the "Compensating chuck mode" field, alternatively the following cycle calls are generated:

- With compensating chuck: CYCLE840
- Without compensating chuck: CYCLE84

When tapping with compensating chuck, the thread is produced in one cut. CYCLE84 enables tapping to be performed in several cuts, when the spindle is equipped with a measuring system.

Input simple (only for G code programs)

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

Machine manufacturer

Various defined values can be pre-assigned using setting data.

Please refer to the machine manufacturer's specifications.

If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

Approach/retraction - CYCLE840 - with compensating chuck

1. The tool moves with G0 to safety clearance of the reference point.
2. The tool drills with G1 and the programmed spindle speed and direction of rotation to depth Z1. The feedrate F is calculated internally in the cycle from the speed and pitch.
3. The direction of rotation is reversed.
4. Dwell time at final drilling depth.
5. Retraction to safety clearance with G1.
6. Reversal of direction of rotation or spindle stop.
7. Retraction to retraction plane with G0.

Approach/retraction CYCLE84 - without compensating chuck in the "1 cut" mode

1. Travel with G0 to the safety clearance of the reference point.
2. Spindle is synchronized and started with the programmed speed (dependent on %S).
3. Tapping with spindle-feedrate synchronization to Z1.
4. Spindle stop and dwell time at drilling depth.
5. Spindle reverse after dwell time has elapsed.
6. Retraction with active spindle retraction speed (dependent on %S) to safety clearance
7. Spindle stop.
8. Retraction to retraction plane with G0.

Approach/retraction CYCLE84 - without compensating chuck in the "swarf removal" mode

1. The tool drills at the programmed spindle speed S (dependent on %S) as far as the 1st infeed depth (maximum infeed depth D).
2. Spindle stop and dwell time DT.
3. The tool retracts from the workpiece for the stock removal with spindle speed SR to the safety clearance.
4. Spindle stop and dwell time DT.
5. The tool then drills with spindle speed S as far as the next infeed depth.
6. Steps 2 to 5 are repeated until the programmed final drilling depth Z1 is reached.
7. On expiry of dwell time DT, the tool is retracted with spindle speed SR to the safety clearance. The spindle stops and retracts to the retraction plane.

Approach/retraction CYCLE84 - without compensating chuck in the "chip breaking" mode

1. The tool drills at the programmed spindle speed S (dependent on %S) as far as the 1st infeed depth (maximum infeed depth D).
2. Spindle stop and dwell time DT.
3. The tool retracts by the retraction distance V2 for chip breaking.

4. The tool then drills to the next infeed depth at spindle speed S (dependent on %S).
5. Steps 2 to 4 are repeated until the programmed final drilling depth Z1 is reached.
6. On expiry of dwell time DT, the tool is retracted with spindle speed SR to the safety clearance. The spindle stops and retracts to the retraction plane.

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

Procedure

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

2. Press the "Drilling" softkey.

3. Press the "Thread" and "Tap" softkeys.
   The "tapping" input window opens.

Parameters in the "Input complete" mode

G code program parameters				ShopMill program parameters		
Input (only for G code)		• complete				
PL ↻	Machining plane			T	Tool name	
RP	Retraction plane	mm		D	Cutting edge number	
SC	Safety clearance	mm		S / V ↻	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
Compensating chuck mode ↻	• With compensating chuck • Without compensating chuck	
Machining position ↻ (only for G code)	• Single position Drill hole at programmed position • Position pattern Position with MCALL	
Z0 (only for G code)	Reference point Z	mm
Z1 ↻	End point of the thread (abs) or thread length (inc) It is inserted into the workpiece until it reaches Z1.	mm

Parameter	Description	Unit
Machining - (with compensating chuck) ↻	You can select the following technologies for tapping: • With encoder Tapping with spindle encoder • Without encoder Tapping without spindle encoder - the following fields are displayed: – Select the "pitch" parameter (only G code) – Enter parameter "DT" (only ShopMill) **Note:** For ShopMill, the selection box is only displayed if tapping without encoder is enabled. Please observe the information provided by your machine manufacturer.	
SR (only for ShopMill)	Spindle speed for retraction - (only for S)	rpm
VR (only for ShopMill)	Constant cutting rate for retraction (only for V)	m/min
Pitch - (only machining without encoder) (only for G code) ↻	• User input Pitch is obtained from the input • Active feedrate Pitch is obtained from the feedrate	
Thread ↻ (only for G code)	Direction of rotation of the thread • Right-hand thread • Left-hand thread (only in mode "without compensating chuck")	
Table ↻	Thread table selection: • Without • ISO metric • Whitworth BSW • Whitworth BSP • UNC	
Selection ↻	Selection of table value: e.g. • M3; M10; etc. (ISO metric) • W3/4"; etc. (Whitworth BSW) • G3/4"; etc. (Whitworth BSP) • 1" - 8 UNC; etc. (UNC)	
P ↻ - (selection only possible for table selection "without")	Pitch ... • in MODULUS: MODULUS = Pitch/π • in turns per inch: Used with pipe threads, for example. When entered per inch, enter the integer number in front of the decimal point in the first parameter field and the figures after the decimal point as a fraction in the second and third field. • in mm/rev • in inch/rev The pitch is determined by the tool used.	MODULUS Turns/" mm/rev in/rev

Parameter	Description	Unit
αS (only for G code)	Starting angle offset - (for rigid tapping only)	Degrees
S (only for G code)	Spindle speed - (for rigid tapping only)	rpm
Machining (not in the "with compensating chuck" mode)	The following machining operations can be selected: • 1 cut The thread is drilled in one cut without interruption. • Chip breaking The drill is retracted by the retraction amount V2 for chip breaking. • Swarf removal The drill is retracted from the workpiece for swarf removal.	
D	Maximum depth infeed - (only when used without compensating chuck, swarf removal or chip breaking)	mm
Retraction	Retraction distance - (for chip breaking only) • Manual Retraction distance after each machining step (V2) • Automatic The tool is retracted by one revolution.	
V2	Retraction distance after each machining step – (only without compensating chuck, chip breaking and manual retraction) Distance by which the drill is retracted for chip breaking.	mm
DT (for ShopMill, only in the "with compensating chuck without encoder" mode)	Dwell time in seconds: • without compensating chuck – 1 cut: Dwell time at final drilling depth – Chip breaking: Dwell time at drilling depth – Swarf removal: Dwell time at the drilling depth and after retraction • with compensating chuck – with encoder: Dwell time after drilling – without encoder: Dwell time at final drilling depth	s
SR (only for G code)	Spindle speed for retraction - (only for when a compensating chuck is not used)	rpm
SDE (only for G code)	Direction of rotation after end of cycle: • • •	

Parameter	Description	Unit
Technology	Adapting the technology: • Yes – Exact stop – Precontrol – Acceleration – Spindle • No **Note:** The technology fields are only displayed if their display has been enabled. Please observe the information provided by your machine manufacturer.	
Exact stop (only for technology, yes)	• Empty: Behavior the same as it was before the cycle was called • G601: Block advance for exact stop fine • G602: Block advance for exact stop coarse • G603: Block advance if the setpoint has been reached	
Precontrol (only for technology, yes)	• Empty: Behavior the same as it was before the cycle was called • FFWON: with precontrol • FFWOF: without precontrol	
Acceleration (only for technology, yes)	(only in mode "without compensating chuck") • Empty: Behavior the same as it was before the cycle was called • SOFT: Jerk-limited (soft) acceleration of the axes • BRISK: Abrupt acceleration of the axes • DRIVE: Reduced axis acceleration	
Spindle (only for technology, yes)	(only in mode "without compensating chuck") • Speed controlled: Spindle for MCALL: Speed-controlled operation • Position controlled: Spindle for MCALL: Position-controlled operation	

Parameters in the mode "Input simple" (only for G code program)

G code program parameters						
Input (only for G code)		• Simple				
RP	Retraction plane	mm				

Parameter	Description	
Compensating chuck mode	• With compensating chuck • Without compensating chuck	
Machining position	• Single position Drill hole at programmed position. • Position pattern Position with MCALL	
Z0	Reference point Z	mm
Z1	End point of the thread (abs) or thread length (inc). It is inserted into the workpiece until it reaches Z1.	mm
Machining - (with compensating chuck)	• With encoder Tapping with spindle encoder • Without encoder Tapping without spindle encoder; selection: - Define "Pitch" parameter	
SR	Spindle speed for retraction - (only for S)	rpm
VR	Constant cutting rate for retraction (only for V)	m/min
Pitch - (only machining without encoder)	• User input Pitch is obtained from the input • Active feedrate Pitch is obtained from the feedrate	
Thread	Direction of rotation of the thread • Right-hand thread • Left-hand thread (only in mode "without compensating chuck")	
Selection	Selection of table value: e.g. • M3; M10; etc. (ISO metric) • W3/4; etc. (Whitworth BSW) • G3/4; etc. (Whitworth BSP) • 1" - 8 UNC; etc. (UNC)	
P	Pitch ... • In MODULUS: MODULUS = Pitch/π • In turns per inch: Used with pipe threads, for example. When entered per inch, enter the integer number in front of the decimal point in the first parameter field and the figures after the decimal point as a fraction in the second and third field. • In mm/rev • In inch/rev The pitch is determined by the tool being used.	MODULUS Turns/" mm/rev in/rev
S	Spindle speed - (only for tapping without compensating chuck).	

Parameter	Description	
Machining (not for "with compensating chuck")	The following machining operations can be selected: • One cut The thread is drilled in one cut without interruption. • Chip breaking The drill is retracted by the retraction amount V2 for chip breaking. • Swarf removal The drill is retracted from the workpiece for swarf removal.	
D	1st drilling depth (abs) or 1st drilling depth referred to Z0 (inc)	mm
SR	Spindle speed for retraction - (only for "without compensating chuck").	rpm

Hidden parameters

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.

Parameter	Description	Value	Can be set in SD
PL	Machining plane	Defined in MD 52005	
SC	Safety clearance	1 mm	x
Table	Thread table selection	Without	
αS	Starting angle offset	0°	
Retraction	Without retraction distance after each machining step - (for chip breaking only)	Automatic	
DT	Dwell time at final drilling depth	0.6 s	x
SDE	Direction of rotation after end of cycle		

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

10.2.9 Drill and thread milling (CYCLE78)

Function

You can use a drill and thread milling cutter to manufacture an internal thread with a specific depth and pitch in one operation. This means that you can use the same tool for drilling and thread milling, a change of tool is superfluous.

The thread can be machined as a right- or left-hand thread.

Approach/retraction

1. The tool traverses with rapid traverse to the safety clearance.
2. If pre-drilling is required, the tool traverses at a reduced drilling feedrate to the predrilling depth defined in a setting data (ShopMill/ShopTurn). When programming in G code, the predrilling depth can be programmed using an input parameter.

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

1. The tool bores at drilling feedrate F1 to the first drilling depth D. If the final drilling depth Z1 is not reached, the tool will travel back to the workpiece surface in rapid traverse for stock removal. Then the tool will traverse with rapid traverse to a position 1 mm above the drilling depth previously achieved - allowing it to continue drilling at drill feedrate F1 at the next infeed. Parameter "DF" is taken into account from the 2nd infeed and higher (refer to the table "Parameters").
2. If another feedrate FR is required for through-boring, the residual drilling depth ZR is drilled with this feedrate.
3. If required, the tool traverses back to the workpiece surface for stock removal before thread milling with rapid traverse.
4. The tool traverses to the starting position for thread milling.
5. The thread milling is carried out (climbing, conventional or conventional + climbing) with milling feedrate F2. The thread milling acceleration path and deceleration path is traversed in a semicircle with concurrent infeed in the tool axis.

Procedure

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

2. Press the "Drilling" softkey.

3. Press the "Thread" and "Drill and thread mill" softkeys.
   The "Drilling and thread milling" input window opens.

Parameters, G code program				Parameters, ShopMill program		
PL	Machining plane			T	Tool name	
RP	Retraction plane	mm		D	Cutting edge number	
SC	Safety clearance	mm		F	Feedrate	mm/min mm/rev
				S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
Machining position (only for G code)	• Single position Drill hole at programmed position • Position pattern Position with MCALL	
F1 (only for G-code)	Drilling feedrate	mm/min mm/rev
Z0 (only for G code)	Reference point Z	mm
Z1	Thread length (inc) or end point of the thread (abs)	mm
D	Maximum depth infeed • D ≥ Z1: Infeed to the final drilling depth • D < Z1: Several infeeds with stock removal	mm
DF	• Percentage for each additional infeed DF=100: Infeed increment remains constant DF<100: Amount of infeed is reduced in direction of final drilling depth Z1 **Example:** last infeed 4 mm; DF 80% next infeed = 4 x 80% = 3.2 mm next but one infeed = 3.2 x 80% = 2.56 mm etc. • Amount for each additional infeed	% mm
V1	Minimum infeed - (only for DF, percentage for each additional infeed) Parameter V1 is only provided if DF<100 has been programmed. If the infeed increment becomes very small, a minimum infeed can be programmed in parameter "V1". • V1 < Infeed increment: The tool is inserted by the infeed increment • V1 > Infeed increment: The tool is inserted by the infeed value programmed under V1.	mm
Predrilling	Predrilling with reduced feedrate • Yes • No The reduced drilling feedrate is obtained as follows: Drilling feedrate F1 < 0.15 mm/rev: Predrilling feedrate = 30% of F1 Drilling feedrate F1 ≥ 0.15 mm/U: Predrilling feedrate = 0.1 mm/rev	
ZA	Predrill to the depth with reduced drilling feedrate (inc) - ("yes", only for predrilling)	mm
Through drilling	Remaining drilling depth with drilling feedrate • Yes • No	
ZD	Depth for feedrate reduction - ("yes", only for through drilling)	mm
FD	Feedrate for through drilling - ("yes", only for through drilling)	mm/min mm/rev
Swarf removal	Swarf removal before thread milling • Yes • No Return to workpiece surface for swarf removal before thread milling.	

Parameter	Description	Unit
Thread	Direction of rotation of the thread • Right-hand thread • Left-hand thread	
F2	Feedrate for thread milling	mm/min mm/tooth
Table	Thread table selection: • without • ISO metric • Whitworth BSW • Whitworth BSP • UNC	
Selection - (not for table "Without")	Selection of table value: e.g. • M3; M10; etc. (ISO metric) • W3/4"; etc. (Whitworth BSW) • G3/4"; etc. (Whitworth BSP) • N1" - 8 UNC; etc. (UNC)	
P - (selection option only for table selection "without")	Pitch ... • in MODULUS: MODULUS = Pitch/π • in turns per inch: Used with pipe threads, for example. When entered per inch, enter the integer number in front of the decimal point in the first parameter field and the figures after the decimal point as a fraction in the second and third field. • in mm/rev • in inch/rev The pitch is determined by the tool used.	MODULUS Turns/" mm/rev in/rev
Z2	Retraction amount before thread milling The thread depth in the direction of the tool axis is defined using Z2. Z2 is relative to the tool tip.	mm
∅	Nominal diameter	mm
Milling direction	• Climb milling: Mill thread in one cycle. • Up-cut operation milling: Mill thread in one cycle. • Synchronous operation - Up-cut operation: Mill thread in two cycles: rough cutting is performed by Up-cut operation milling with defined allowances, then finish cutting is performed by climb milling with milling feedrate FS.	
FS	Finishing feedrate - (only for Synchronous operation - Up-cut operation milling)	mm/min mm/tooth

10.2.10 Positioning and position patterns

Function

The positions are programmed after the technology (cycle call). Several position patterns are available:

- Arbitrary positions
- Position on a row, on a grid or a frame
- Position on a full or pitch circle

Several position patterns can be programmed in succession. They are executed in the order in which you program them.

Note

The number of positions that can be programmed in the one "Positions" step is limited to a maximum of 600!

Programming a position pattern in ShopMill

Several position patterns can be programmed in succession (up to 20 technologies and position patterns in total). They are executed in the order in which you program them.

The programmed technologies and subsequently programmed positions are automatically linked by the control.

Displaying and hiding positions

You can display or hide any positions (Section "Displaying and hiding positions (Page 431)").

Approach/retraction

1. Within a position pattern, or while approaching the next position pattern, the tool is retracted to the retraction plane and the new position or position pattern is then approached at rapid traverse.
2. With subsequent technological operations (e.g. centering - drilling - tapping), the respective drilling cycle must be programmed after calling the next tool (e.g. drill) and immediately afterwards the call of the position pattern to be machined.

Tool traverse path

- ShopMill
  The programmed positions are machined with the previously programmed tool (e.g. center drill). Machining of the positions always starts at the reference point. In the case of a grid, machining is performed first in the direction of the 1st axis and then meandering back and forth. The frame and circle or pitch circle are machined counter-clockwise.
- G code
  For G code, for rows/frames/grids, a start is always made at the next corner of the frame or grid or the end of the row. The frame and circle or pitch circle are machined counter-clockwise.

Working with rotary axes

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

If a rotary axis in the table is set up on your machine that permits peripheral surface machining, this axis is supported during drilling (any position pattern, full circle, and pitch circle).

If peripheral surface machining is not possible in the basic position of the machine because the tool is perpendicular to the end face of the cylinder, the table or head must be swiveled before machining (swiveling plane).

Example: Compensator with A axis and C rotary table

For peripheral surface machining, the table is swiveled through 90°. This means the C table rotates about the geometry axis Y and acts as a B axis in G17.

If several rotary axes are set up in the table, you can choose between these rotary axes. The following description assumes an A axis (rotates about the geometry axis X).

You define a work offset:

X = end face of the cylinder

Y = center point of the cylinder in the Y direction

Z = center point of the cylinder in the Z direction

The "cylinder" in this case refers to any part that is clamped in the A axis.

Note

A work offset in the rotary axis is effective even while cylinder surface transformation is active.

10.2.11 Arbitrary positions (CYCLE802)

Function

The "Arbitrary positions" function allows you to program any positions, i.e. in rectangular or polar coordinates. Individual positions are approached in the order in which you program them.

Press softkey "Delete all" to delete all positions programmed in X/Y.

Rotary axis

XA plane

You program in XA to prevent the Y axis moving during machining.

To ensure that the holes point to the center of the "Cylinder", you must first position the Y axis centrally above the "Cylinder".

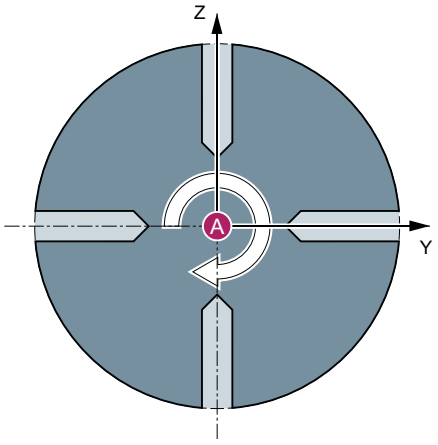


Figure 10-1 Y axis is centered above the cylinder

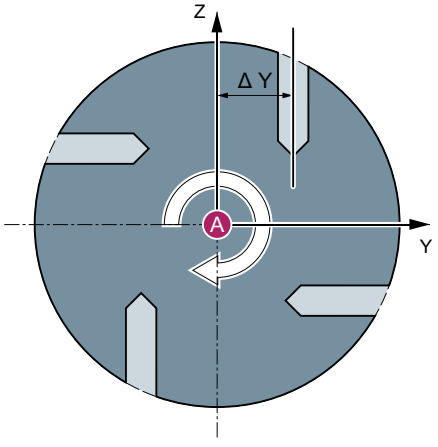


Figure 10-2 Y axis is not centered above the cylinder

XYA plane

You program in XYA if the Y axis should also move during machining. A value can be specified for each position.
In addition to the possibilities of XA, the following is also possible, for example.

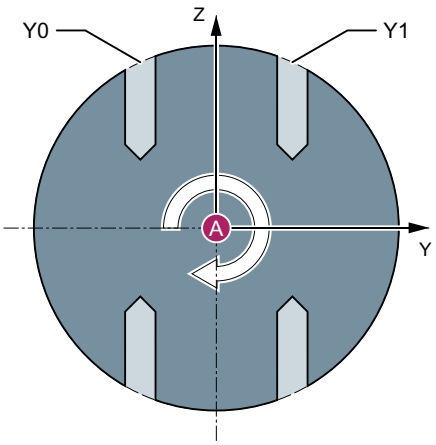


Figure 10-3 Y axis is traversed (Y0, Y1)

See also

Positioning and position patterns (Page 421)

Procedure

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

2. Press the "Drilling" softkey.

3. Press the "Positions" softkey.
   The "Positions" input window opens.

Parameter	Description	Unit
LAB - (only for G code)	Repeat jump label for position	
PL - (only for G code)	Machining plane	

Parameter	Description	Unit
Axes	Selection of the participating axes • XY (1st and 2nd axis of the plane) • XA (1st rotary axis and assigned linear axis) • XYA (1st rotary axis and both axes of the plane) **Note**: Rotary axes are only displayed in the selection field if they have been released for use in the position pattern. Please observe the information provided by your machine manufacturer.	
Selection	Coordinate system (only available for axis selection XY) • Right-angled • Polar	mm
Z0	Z coordinate of the reference point (abs)	mm
	Axes: XY / coordinate system – polar	
XP	X coordinate of the reference point – pole (abs)	mm
YP	Y coordinate of the reference point – pole (abs)	mm
L0	Length (abs) – distance of the 1st Position from the pole	mm
C0	Angle (abs) – Angle of the 1st position referred to the reference axis	Degrees
L1 ... L7	Lengths of additional positions (abs or inc)	mm
C1 ... C7	Angles of additional positions (abs or inc)	Degrees
(only for ShopMill)	**Axes: XY / coordinate system – right angled**	
X0	X coordinate of the 1st position (abs)	mm
Y0	Y coordinate of the 1st position (abs)	mm
X1 ... X8	X coordinate for further positions (abs. or inc.)	mm
Y1 ... Y8	Y coordinate for further positions (abs. or inc.)	mm
(only for G code)	(only "right angled")	
X0	X coordinate of the 1st position (abs)	mm
Y0	Y coordinate of the 1st position (abs)	mm
X1 ... X8	X coordinate of additional positions (abs or inc)	mm
Y1 ... Y8	Y coordinate of additional positions (abs or inc)	mm
	Axes: XA	
X0	X coordinate of the 1st position (abs)	mm
A0	A coordinate (angle) of the 1st position (abs)	Degrees
X1 ... X8	X coordinates of additional positions (abs or inc)	mm
A1 ... A8	A coordinates (angle) of additional positions (abs or inc)	mm
	Axes: XYA	
X0	X coordinate of the 1st position (abs)	mm
Y0	Y coordinate of the 1st position (abs)	mm
A0	A coordinate (angle) of the 1st position (abs)	Degrees
X1 ... X5	X coordinates of additional positions (abs or inc)	mm
Y1 ... Y5	Y coordinates of additional positions (abs or inc)	mm
A1 ... A5	A coordinates (angle) of additional positions (abs or inc)	Degrees

10.2.12 Row position pattern (HOLES1)

Function

You can program any number of positions at equal distances along a line using the "Row position pattern" function.

Procedure

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

2. Press the "Drilling" softkey.

3. Press the "Positions" and "Row" softkeys.

The "Position row" input window opens.

Parameter	Description	Unit
LAB (only for G code)	Repeat jump label for position	
PL (only for G code)	Machining plane	
Z0 (only for ShopMill)	Z coordinate of reference point Z (abs)	mm
X0	X coordinate of the reference point X (abs) This position must be programmed absolutely in the 1st call.	mm
Y0	Y coordinate of the reference point Y (abs) This position must be programmed absolutely in the 1st call.	mm
α0	Angle of rotation of the line referred to the X axis Positive angle: Line is rotated counter-clockwise. Negative angle: Line is rotated clockwise.	Degrees
L0	Distance of the 1st position to reference point	mm
L	Distance between the positions	mm
N	Number of positions	

10.2.13 Grid or frame position pattern (CYCLE801)

Function

- You can use the "Grid position pattern" function (CYCLE801) to program any number of positions that are spaced at an equal distance along one or several parallel lines.
  If you want to program a rhombus-shaped grid, enter angle αX or αY.
- Frame
  You can use the "Frame position pattern" function (CYCLE801) to program any number of positions that are spaced at an equal distance on a frame. The spacing may be different on both axes.
  If you want to program a rhombus-shaped frame, enter angle αX or αY.

Procedure

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

2. Press the "Drilling" softkey.

3. Press the "Positions" softkey.

4. Press the "Grid" softkey.

   - OR -

   Press the "Frame" softkey.

   The "Position grid" or "Position frame" input window opens.

Parameters - "Grid" position pattern

Parameter	Description	Unit
LAB (only for G code)	Repeat jump label for position	
PL (only for G code)	Machining plane	
Z0 (only for Shop-Mill)	Z coordinate of reference point Z (abs)	mm
X0	X coordinate of the reference point X (abs) This position must be programmed absolutely in the 1st call.	mm
Y0	Y coordinate of the reference point Y (abs) This position must be programmed absolutely in the 1st call.	mm

Parameter	Description	Unit
α0	Angle of rotation of the line referred to the X axis Positive angle: Line is rotated counter-clockwise. Negative angle: Line is rotated clockwise.	Degrees
αX	Shear angle X	Degrees
αY	Shear angle Y	Degrees
L1	Distance between columns	mm
L2	Distance between rows	mm
N1	Number of columns	
N2	Number of rows	

Parameters - "Frame" position pattern

Parameter	Description	Unit
LAB (only for G code)	Repeat jump label for position	
PL (only for G code)	Machining plane	
Z0 (only for Shop-Mill)	Z coordinate of reference point Z (abs)	mm
X0	X coordinate of the reference point X (abs) This position must be programmed absolutely in the 1st call.	mm
Y0	Y coordinate of the reference point Y (abs) This position must be programmed absolutely in the 1st call.	mm
α0	Angle of rotation of the line referred to the X axis Positive angle: Line is rotated counter-clockwise. Negative angle: Line is rotated clockwise.	Degrees
	Frame	
αX	Shear angle X	Degrees
αY	Shear angle Y	Degrees
L1	Distance between columns	mm
L2	Distance between rows	mm
N1	Number of columns	
N2	Number of rows	

10.2.14 Circle or pitch circle position pattern (HOLES2)

Function

You can program holes on a full circle or a pitch circle of a defined radius with the "Circle position pattern" and "Pitch circle position pattern" functions. The basic angle of rotation (α0) for the 1st position is relative to the X axis. The control calculates the angle of the next hole position as a function of the total number of holes. The angle it calculates is identical for all positions.

The tool can approach the next position along a linear or circular path.

Rotary axes

If rotary axes are set up on your machine, you can select these axes for the "circle" or "pitch circle" position patterns.

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

See also

Positioning and position patterns (Page 421)

Procedure

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

Press the "Drilling" softkey.
3.

Press the "Positions" softkey.
4.
Press the "Circle" softkey.
- OR -

Press the "Partial circle" softkey.

The "Circle position" or "Pitch circle position" input window opens.

Parameters - "Circle" position pattern

Parameter	Description	Unit
LAB (only for G code)	Repeat jump label for position	
PL (only for G code)	Machining plane	

Parameter	Description	Unit
Axes	Selection of the participating axes • XY (1st and 2nd axis of the plane) • XA (1st rotary axis and assigned linear axis) **Note**: Rotary axes are only displayed in the selection field if they have been released for use in the position pattern. Please observe the information provided by your machine manufacturer.	
Z0 (only for ShopMill)	Z coordinate of the reference point	mm
	Axes XY (at right angles)	
X0	X coordinate of the reference point X (abs)	mm
Y0	Y coordinate of the reference point Y (abs)	mm
α0	Starting angle for first position. Positive angle: Full circle is rotated counter-clockwise. Negative angle: Full circle is rotated in clockwise direction.	Degrees
R	Radius	
N	Number of positions	mm
Positioning	Positioning motion between the positions • Straight line Next position is approached linearly in rapid traverse. • Circle Next position is approached along a circular path at the feedrate defined in the machine data.	
	Axes: XA	
X0	X coordinate of the reference point (abs)	mm
A0	Start angle of the A axis (abs)	Degrees
N	Number of positions	

Parameters - "Pitch circle" position pattern

Parameter	Description	Unit
LAB (only for G code)	Repeat jump label for position	
PL (only for G code)	Machining plane	
Axes	Selection of the participating axes • XY (1st and 2nd axis of the plane) • XA (1st rotary axis and assigned linear axis) **Note**: Rotary axes are only displayed in the selection field if they have been released for use in the position pattern. Please observe the information provided by your machine manufacturer.	
Z0	Z coordinate of the reference point	mm

Parameter	Description	Unit
	Axes XY (at right angles)	
X0	X coordinate of the reference point X (abs)	mm
Y0	Y coordinate of the reference point Y (abs)	mm
α0	Starting angle for first position. Positive angle: Full circle is rotated counter-clockwise. Negative angle: Full circle is rotated in clockwise direction.	Degrees
α1	Advance angle After the first hole has been drilled, all further positions are advanced by this angle. Positive angle: Further positions are rotated counter-clockwise. Negative angle: Further positions are rotated clockwise.	Degrees
R	Radius	mm
N	Number of positions	
Positioning 	Positioning motion between the positions • Straight line Next position is approached linearly in rapid traverse. • Circle Next position is approached along a circular path at the feedrate defined in the machine data.	
	Axes: XA	
X0	X coordinate of the reference point (abs)	mm
A0	Start angle of the A axis (abs)	Degrees
A1	Advance angle of the A axis (inc)	Degrees
N	Number of positions	

10.2.15 Displaying and hiding positions

Function

You can hide any positions in the following position patterns:

- Position pattern line
- Position pattern grid
- Position pattern frame
- Full circle position pattern
- Pitch circle position pattern

The hidden positions are skipped when machining.

Display

The programmed positions of the position pattern are shown as follows in the programming graphic:

x Position is activated = displayed (position is shown as a cross)
o Position deactivated = hidden (position shown as a circle)

Selecting positions

You have the option of either displaying or hiding positions - by activating the checkbox in the displayed position table either using the keyboard or mouse.

Procedure

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

2. Press the "Drilling" and "Positions" softkeys.

3. Press the "Line/Grid/Frame" or "Full/Pitch Circle" softkeys.

4. Press the "Hide position" softkey.

   The "Hide position" window opens on top of the input form of the position pattern. The positions are displayed in a table.

   The numbers of the positions, their angle(α) as well as a checkbox with the state (activated = check mark set / deactivated = no check mark set) are displayed.

   The selected position in the graphic is highlighted in color.

5. Using the mouse, select the required position and deactivate or activate the checkbox in order to hide the position or display it again.

   In the diagram, skipped positions are shown in the form of a circle and displayed (active) positions are shown in the form of a cross.

   Note: You have the option of selecting individual positions using the <Cursor up> or <Cursor down> keys – and hiding and displaying using the <SELECT> key.

Display or hide all positions at once

1. Press the "Hide all" softkey to hide all positions.

2. Press the "Show all" softkey to display all positions again.

10.2.16 Repeating positions

Function

If you want to approach positions that you have already programmed again, you can do this quickly with the function "Repeat position".

You must specify the number of the position pattern. The cycle automatically assigns this number (for ShopMill). You will find this position pattern number in the work plan (program view) or G-code program after the block number.

Procedure

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

2. Press the "Drilling", and "Repeat position" softkeys.
   The "Repeat positions" input window opens.

3. After you have entered the label or the position pattern number, e.g. 1, press the "Accept" softkey. The position pattern you have selected is then approached again.

Parameter	Description	Unit
LAB (only for G code)	Repeat jump label for position	
Position (only for ShopMill)	Enter the number of the position pattern	

10.3 Milling

10.3.1 Face milling (CYCLE61)

Function

You can face mill any workpiece with the "Face milling" cycle.

A rectangular surface is always machined.

Workpieces with and without limits can be face-milled.

Approach/retraction

1. For vertical machining, the starting point is always at the top or bottom. For horizontal machining, it is at the left or right.
   The starting point is marked in the help display.
2. Machining is performed from the outside to the inside.

Machining type

The cycle makes a distinction between roughing and finishing:

- Roughing:
  Milling the surface
  Tool turns above the workpiece edge
- Finishing:
  Milling the surface once
  Tool turns at safety distance in the X/Y plane
  Retraction of milling cutter

Depth infeed always takes place outside the workpiece.

For a workpiece with edge breaking, select the rectangular spigot cycle.

In face milling, the effective tool diameter for a tool of type "Milling cutter" is stored in a machine data item.

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

Selecting the machining direction

Toggle the machining direction in the "Direction" field until the symbol for the required machining direction appears.

- Same direction of machining
- Alternating direction of machining

Selecting limits

Press the respective softkey for the required limit.

Left

Top

Bottom

Right

The selected limits are shown in the help screen and in the broken-line graphics.

Procedure

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

2. Press the "Milling" softkey.

3. Press the "Face milling" softkey.
   The "Face Milling" input window opens.

Parameters, G code program				**Parameters, ShopMill program**		
PL	Machining plane			T	Tool name	
RP	Retraction plane	mm		D	Cutting edge number	
SC	Safety clearance	mm		F	Feedrate	mm/min mm/tooth
F	Feedrate	*		S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
Machining ○	The following machining operations can be selected: • ∇ (roughing) • ∇∇∇ (finishing)	
Direction ○	Same direction of machining • • Alternating direction of machining • •	
	The positions refer to the reference point:	
X0	Corner point 1 in X	mm
Y0	Corner point 1 in Y	mm
Z0	Height of blank	mm
X1 ○	Corner point 2X (abs) or corner point 2X in relation to X0 (inc)	mm
Y1 ○	Corner point 2Y (abs) or corner point 2Y in relation to Y0 (inc)	mm
Z1 ○	Height of blank (abs) or height of blank in relation to Z0 (inc)	mm
DXY ○	Maximum plane infeed Alternately, you can specify the plane infeed in %, as a ratio → plane infeed (mm) to the milling cutter diameter (mm).	mm %
DZ	Maximum depth infeed – (for roughing only)	mm
UZ	Finishing allowance, depth	mm

* Unit of feedrate as programmed before the cycle call

Note

The same finishing allowance must be entered for both roughing and finishing. The finishing allowance is used to position the tool for retraction.

10.3.2 Rectangular pocket (POCKET3)

Function

You can mill any rectangular pocket with the "rectangular pocket milling" function.

The following machining variants are available:

- Mill rectangular pocket from solid material.
- Predrill rectangular pocket in the center first if, for example, the milling cutter does not cut in the center (program the drilling, rectangular pocket and position program blocks in succession).
- Machine pre-machined rectangular pocket (see "Solid machining" parameter):
  - Complete machining
  - Post machining

Depending on the dimensions of the rectangular pocket in the workpiece drawing, you can select a corresponding reference point for the rectangular pocket.

Note

Predrilling

If the programmed input parameters, deviating from Pocket3, result in a longitudinal slot or a longitudinal hole, then in the cycle, from Pocket3, the corresponding cycle to machine slots (Slot1 or Longhole) is called. In these cases, the insertion points can deviate from the pocket center point.

Note this special feature if you wish to predrill.

Input simple

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

Machine manufacturer

Various defined values can be pre-assigned using setting data.

Please refer to the machine manufacturer's specifications

If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

Approach/retraction

1. The tool approaches the center point of the rectangular pocket in rapid traverse at the height of the retraction plane and adjusts to the safety clearance.
2. The tool is inserted into the material according to the chosen strategy.
3. The rectangular pocket is always machined with the chosen machining type from inside out.
4. The tool moves back to the safety clearance at rapid traverse.

Machining type

- Roughing
  During roughing, the individual planes of the rectangular pocket are machined one after the other from the center point until depth Z1 is reached.
- Finishing
  During finishing, the edge is always machined first. The rectangular pocket edge is approached on the quadrant that joins the corner radius. In the last infeed, the base is finished from the center out.
- Edge finishing
  Edge finishing is performed in the same way as finishing, except that the last infeed (finish base) is omitted.
- Chamfering
  Chamfering involves edge breaking at the upper edge of the rectangular pocket.

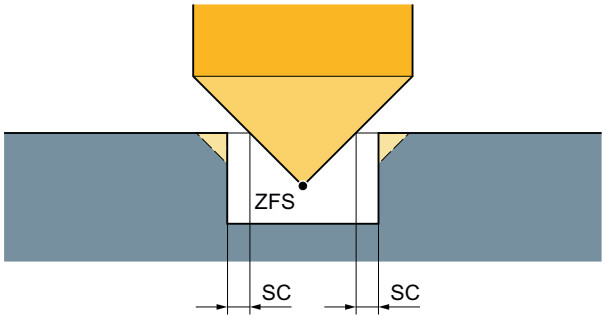


Figure 10-4 Geometries when chamfering inside contours

Note

The tip angle is evaluated when chamfering. During chamfering, the end mill behaves like a centering tool with a 90° tip angle.

Note

The following error messages can occur when chamfering inside contours:

- **Safety clearance in the program header too large**
  This error message appears when chamfering would, in principle, be possible with the parameters entered for FS and ZFS, but the safety clearance then could not be maintained.
- **Immersion depth too large**
  This error message appears when chamfering would be possible through the reduction of the immersion depth ZFS.
- **Tool diameter too large**
  This error message appears when the tool would already damage the edges during insertion. In this case, the chamfer FS must be reduced.

Procedure

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

2. Press the "Milling" softkey.

3. Press the "Pocket" and "Rectangular pocket" softkeys.
   The "Rectangular pocket" input window opens.

Parameters in the "Input complete" mode

Parameters, G code program				**Parameters, ShopMill program**		
Input		• complete				
PL	Machining plane			T	Tool name	
	Milling direction			D	Cutting edge number	
RP	Retraction plane	mm		F	Feedrate	mm/min mm/tooth
SC	Safety clearance	mm		S / V	Spindle speed or constant cutting rate	rpm m/min
F	Feedrate	*				

Parameter	**Description**	**Unit**
Reference point	The following different reference point positions can be selected: • (center) • (bottom left) • (bottom right) • (top left) • (top right) The reference point (highlighted in blue) is displayed in the Help screen.	
Machining	The following machining operations can be selected: • ∇ (roughing) • ∇∇∇ (finishing) • ∇∇∇ edge (edge finishing) • Chamfering	
Machining position	• Single position Mill rectangular pocket at the programmed position (X0, Y0, Z0). • Position pattern Position with MCALL	

Parameter	Description	Unit
	The positions refer to the reference point:	
X0	Reference point X – (single position only)	mm
Y0	Reference point Y – (single position only)	mm
Z0	Reference point Z – (single position only and G Code position pattern)	mm
W	Pocket width	mm
L	Pocket length	mm
R	Corner radius	mm
α0	Angle of rotation	Degrees
Z1	Depth referred to Z0 (inc) or pocket depth (abs) - (only for ∇, ∇∇∇ or ∇∇∇ edge)	mm
DXY	• Maximum plane infeed • Maximum plane infeed as a percentage of the milling cutter diameter - (only for ∇ and ∇∇∇)	mm %
DZ	Maximum depth infeed – (only for ∇, ∇∇∇ or ∇∇∇ edge)	mm
UXY	Plane finishing allowance – (only for ∇, ∇∇∇ or ∇∇∇ edge)	mm
UZ	Depth finishing allowance – (only for ∇ or ∇∇∇)	mm
Insertion	The following insertion modes can be selected – (only for ∇, ∇∇∇ or ∇∇∇ edge): • **Predrilled:** (only for G code) With G0, the pocket center point is approached at the retraction plane level, and then, from this position, also with G0, the axis travels to the reference point brought forward by the safety clearance. The machining of the rectangular pocket is then performed according to the selected insertion strategy, taking into account the programmed blank dimensions. • **Perpendicular: Insert vertically at the center of pocket** The tool executes the calculated actual depth infeed at the pocket center in a single block. This setting can be used only if the cutter can cut across center or if the pocket has been predrilled. • **Helical: Insert along helical path** The cutter center point traverses along the helical path determined by the radius and depth per revolution (helical path). If the depth for one infeed has been reached, a full circle motion is executed to eliminate the inclined insertion path. • **Oscillating: Insert with oscillation along center axis of rectangular pocket** The cutter center point oscillates back and forth along a linear path until it reaches the depth infeed. When the depth has been reached, the path is traversed again without depth infeed in order to eliminate the inclined insertion path.	
FZ (only for G code)	Depth infeed rate – (for vertical insertion only)	*
FZ (only for ShopMill)	Depth infeed rate – (for vertical insertion only)	mm/min mm/tooth
EP	Maximum pitch of helix – (for helical insertion only)	mm/rev
ER	Radius of helix – (for helical insertion only) The radius cannot be any larger than the cutter radius; otherwise, material will remain.	mm
EW	Maximum insertion angle – (for insertion with oscillation only)	Degrees

Parameter	Description	Unit
Solid machining (for roughing only) U	• Complete machining The rectangular pocket is milled from the solid material. • Post machining A rectangular pocket or hole has already been machined in the workpiece. This needs to be enlarged in one or several axes. You must program parameters AZ, W1 and L1 for this purpose.	
AZ	Depth of premachining – (for remachining only)	mm
W1	Width of premachining – (for remachining only)	mm
L1	Length of premachining – (for remachining only)	mm
FS	Chamfer width for chamfering - (for chamfering only)	mm
ZFS U	Insertion depth of tool tip (abs or inc) – (for chamfer only)	mm

* Unit of feedrate as programmed before the cycle call

Parameters in the "Input simple" mode

G code program parameters				ShopMill program parameters		
Input U		• simple				
U	Milling direction			T	Tool name	
RP	Retraction plane	mm		D	Cutting edge number	
F	Feedrate	*		F U	Feedrate	mm/min mm/rev
				S / V U	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	
Machining U	The following machining operations can be selected: • ∇ (roughing) • ∇∇∇ (finishing) • ∇∇∇ edge (edge finishing) • Chamfering	
	The positions refer to the reference point:	
X0	Reference point X – (single position only)	mm
Y0	Reference point Y – (single position only)	mm
Z0	Reference point Z – (only single position and G Code position pattern)	mm
W	Pocket width	mm
L	Pocket length	mm
R	Corner radius	mm
Z1 U	Depth referred to Z0 (inc) or pocket depth (abs) - (only for ∇, ∇∇∇ or ∇∇∇ edge)	mm

Parameter	Description	
DXY	• Maximum plane infeed • Maximum plane infeed as a percentage of the milling cutter diameter - (only for ∇ and ∇∇∇)	mm %
DZ	Maximum depth infeed – (only for ∇, ∇∇∇ or ∇∇∇ edge)	mm
UXY	Plane finishing allowance – (only for ∇, ∇∇∇ or ∇∇∇ edge)	mm
UZ	Depth finishing allowance – (only for ∇ or ∇∇∇)	mm
Insertion	The following insertion modes can be selected – (only for ∇, ∇∇∇ or ∇∇∇ edge): • **Predrilled:** (only for G code) With G0, the pocket center point is approached at the retraction plane level, and then, from this position, also with G0, the axis travels to the reference point brought forward by the safety clearance. The machining of the rectangular pocket is then performed according to the selected insertion strategy, taking into account the programmed blank dimensions. • **Perpendicular: Insert vertically at the center of pocket** The tool executes the calculated actual depth infeed at the pocket center in a single block. This setting can be used only if the cutter can cut across center or if the pocket has been predrilled. • **Helical: Insert along helical path** The cutter center point traverses along the helical path determined by the radius and depth per revolution (helical path). If the depth for one infeed has been reached, a full circle motion is executed to eliminate the inclined insertion path. • **Oscillating: Insert with oscillation along center axis of rectangular pocket** The cutter center point oscillates back and forth along a linear path until it reaches the depth infeed. When the depth has been reached, the path is traversed again without depth infeed in order to eliminate the inclined insertion path.	
FZ (only for G code)	Depth infeed rate – (only for vertical insertion)	*
FZ (only for ShopMill)	Depth infeed rate – (only for vertical insertion)	mm/min mm/tooth
EP	Maximum pitch of helix – (for helical insertion only)	mm/rev
ER	Radius of helix – (for helical insertion only) The radius cannot be any larger than the cutter radius; otherwise, material will remain.	mm
EW	Maximum insertion angle – (for insertion with oscillation only)	Degrees
FS	Chamfer width for chamfering - (only for chamfering)	mm
ZFS	Insertion depth of tool tip (abs or inc) – (only for chamfering)	mm

* Unit of feedrate as programmed before the cycle call

Hidden parameters

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.

Parameter	Description	Value	Can be set in SD
PL (only for G code)	Machining plane	Defined in MD 52005	
SC (only for G code)	Safety clearance	1 mm	x
Reference point	Position of the reference point: Center		
Machining position	Mill rectangular pocket at the programmed position (X0, Y0, Z0).	Single position	
α0	Angle of rotation	0°	
Solid machining	The rectangular pocket is milled from the solid material - (only for roughing)	Complete machining	

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

10.3.3 Circular pocket (POCKET4)

Function

You can mill any circular pocket with the "Circular pocket" cycle.

The following machining methods are available:

- Mill circular pocket from solid material.
- Predrill circular pocket in the center first if, for example, the milling cutter does not cut in the center (program the drilling, circular pocket and position program blocks in succession).
- Machine pre-machined circular pocket (see "Solid machining" parameter).
  - Complete machining
  - Post machining

The following machining types are available for milling using the "Circular pocket" function:

- Plane-by-plane
- Helical

Input simple

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

Machine manufacturer

Various defined values can be pre-assigned using setting data.

Please refer to the machine manufacturer's specifications.

If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

Approach/retraction for plane-by-plane solid machining

In plane-by-plane machining of the circular pocket, the material is removed horizontally, one layer at a time.

1. The tool approaches the center point of the pocket at rapid traverse at the height of the retraction plane and adjusts to the safety clearance.
2. The tool is inserted into the material according to the chosen strategy.
3. The circular pocket is always machined from inside out using the selected machining method.
4. The tool moves back to the safety clearance at rapid traverse.

Approach/retraction for helical solid machining

In helical reaming, the material is removed down to pocket depth in a helical movement.

1. The tool approaches the center point of the pocket at rapid traverse at the height of the retraction plane and adjusts to the safety clearance.
2. Infeed to the first machining diameter.
3. The circular pocket is machined with the chosen machining type up to pocket depth or up to pocket depth with finishing allowance.
4. The tool moves back to the safety clearance at rapid traverse.
5. Lateral infeed to the next machining diameter.

Machining type: Plane-by-plane

When milling circular pockets, you can select the following machining types:

- Roughing
  During roughing, the individual planes of the circular pocket are machined one after the other from center point until depth Z1 is reached.
- Finishing
  During finishing, the edge is always machined first. The pocket edge is approached on the quadrant that joins the pocket radius. In the last infeed, the base is finished from the center out.

- Edge finishing
  Edge finishing is performed in the same way as finishing, except that the last infeed (finish base) is omitted.
- Chamfering
  Chamfering involves edge breaking at the upper edge of the circular pocket.

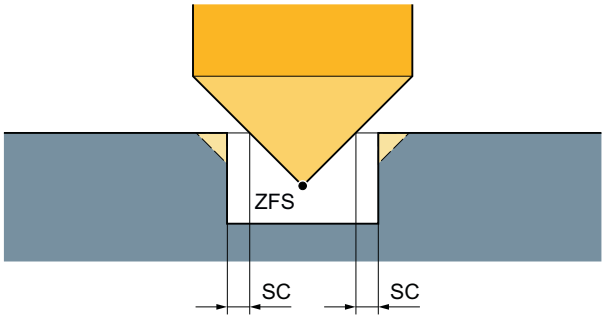


Figure 10-5 Geometries when chamfering inside contours

Note

During chamfering, the end mill behaves like a centering tool with a 90° tip angle.

Note

The following error messages can occur when chamfering inside contours:

- **Safety clearance in the program header too large**
  This error message appears when chamfering would, in principle, be possible with the parameters entered for FS and ZFS, but the safety clearance then could not be maintained.
- **Immersion depth too large**
  This error message appears when chamfering would be possible through the reduction of the immersion depth ZFS.
- **Tool diameter too large**
  This error message appears when the tool would already damage the edges during insertion. In this case, the chamfer FS must be reduced.

Machining type: Helical

When milling circular pockets, you can select the following machining types:

- Roughing
  During roughing, the circular pocket is machined downward with helical movements.
  A full circle is effected down to pocket depth to remove the residual material.
  The tool is retracted from the edge and base of the pocket in a quadrant and retracted with rapid traverse to a safety clearance.
  This process is repeated layer-by-layer, from inside out, until the circular pocket has been completely machined.
- Finishing
  In finishing mode, the edge is machined first with a helical movement down to the bottom.
  A full circle is effected down to pocket depth to remove the residual material.
  The base is milled from outside in a spiral movement.
  The tool is retracted from the center of the pocket to a safety clearance.
- Edge finishing
  In edge finishing, the edge is machined first with a helical movement down to the bottom.
  A full circle is effected down to pocket depth to remove the residual material.
  The tool is retracted from the edge and base of the pocket in a quadrant and retracted with rapid traverse to a safety clearance.

Procedure

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

2. Press the "Milling" softkey.

3. Press the "Pocket" and "Circular pocket" softkeys.
   The "Circular Pocket" input window opens.

Parameters in the "Input complete" mode

Parameters, G code program				Parameters, ShopMill program		
Input		• complete				
PL	Machining plane			T	Tool name	
↻	Milling direction			D	Cutting edge number	
RP	Retraction plane	mm		F ↻	Feedrate	mm/min mm/tooth
SC	Safety clearance	mm		S / V ↻	Spindle speed or constant cutting rate	rpm m/min
F	Feedrate	*				

Parameter	Description	Unit
Machining ↻	• ∇ (roughing, plane-by-plane or helical) • ∇∇∇ (finishing, plane-by-plane or helical) • ∇∇∇ edge (edge finishing, plane-by-plane or helical) • Chamfering	
Machining type ↻	• Plane-by-plane Machine circular pocket plane-by-plane • Helical Machine circular pocket using helical type	
Machining position ↻	• Single position A circular pocket is machined at the programmed position (X0, Y0, Z0). • Position pattern Several circular pockets are machined in a position pattern (e.g. full circle, pitch circle, grid, etc.).	
 X0 Y0 Z0	The reference points refer to the center point of the circular pocket: Reference point X – (for single position only) Reference point Y – (for single position only) Reference point Z – (single position only and G Code position pattern)	 mm mm mm
∅	Diameter of pocket	mm
Z1 ↻	Pocket depth (abs) or depth relative to Z0 (inc) – (only for ∇, ∇∇∇ and ∇∇∇ edge)	mm
DXY ↻	• Maximum plane infeed • Maximum plane infeed as a percentage of the milling cutter diameter - (only for ∇ and ∇∇∇)	In %
DZ	Maximum depth infeed - (only for ∇, ∇∇∇ and ∇∇∇ Rand)	mm
UXY	Plane finishing allowance - (only for ∇, ∇∇∇ and ∇∇∇ edge)	mm
UZ	Depth finishing allowance – (only for ∇ and ∇∇∇)	mm
Insertion ↻	Various insertion modes can be selected – (only for plane-by-plane machining method and for ∇, ∇∇∇ or ∇∇∇ edge): • **Predrilled** (only for G code) • **Perpendicular: Insert vertically at the center of pocket** The tool executes the calculated depth infeed vertically in the center of the pocket. Feedrate: Infeed rate as programmed under FZ • **Helical: Insert along helical path** The cutter center point traverses along the helical path determined by the radius and depth per revolution. If the depth for one infeed has been reached, a full circle motion is executed to eliminate the inclined insertion path. Feedrate: Machining feedrate Note: The vertical insertion into pocket center method can be used only if the tool can cut across center or if the workpiece has been predrilled.	
FZ (only for G code)	Depth infeed rate – (only for insertion and vertical insertion)	*

Parameter	Description	Unit
FZ (only for ShopMill)	Depth infeed rate – (only for insertion and vertical insertion)	mm/min mm/tooth
EP	Maximum pitch of helix - (for helical insertion only) The helix pitch may be lower due to the geometrical situation.	mm/rev
ER	Radius of helix - (only for helical insertion) The radius must not be larger than the milling cutter radius, otherwise material will remain. Also make sure the circular pocket is not violated	mm
Solid machining	• **Complete machining** The circular pocket must be milled from a solid workpiece (e.g. casting). • **Post machining** A small pocket or hole has already been machined in the workpiece. This needs to be enlarged. Parameters AZ, and ∅1 must be programmed.	
FS	Chamfer width for chamfering - (for chamfering only)	mm
ZFS	Insertion depth of tool tip (abs or inc) - (for chamfering only)	mm
AZ	Depth of premachining - (for remachining only)	mm
∅1	Diameter of premachining - (for remachining only)	mm

* Unit of feedrate as programmed before the cycle call

Parameters in the "Input simple" mode

G code program parameters				ShopMill program parameters		
Input		• simple				
	Milling direction			T	Tool name	
RP	Retraction plane	mm		D	Cutting edge number	
F	Feedrate	*		F	Feedrate	mm/min mm/rev
				S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	
Machining	The following machining operations can be selected: • ∇ (roughing) • ∇∇∇ (finishing) • ∇∇∇ edge (edge finishing) • Chamfering	
Machining type	• Plane-by-plane Machine circular pocket plane-by-plane • Helical Machine circular pocket using helical type	
	The positions refer to the reference point:	
X0	Reference point X – (single position only)	mm
Y0	Reference point Y – (single position only)	mm
Z0	Reference point Z – (only single position and G Code position pattern)	mm
∅	Diameter of pocket	mm
Z1	Depth referred to Z0 (inc) or pocket depth (abs) - (only for ∇, ∇∇∇ or ∇∇∇ edge)	mm
DXY	• Maximum plane infeed • Maximum plane infeed as a percentage of the milling cutter diameter - (only for ∇ and ∇∇∇)	mm %
DZ	Maximum depth infeed – (only for ∇, ∇∇∇ or ∇∇∇ edge)	mm
UXY	Plane finishing allowance – (only for ∇, ∇∇∇ or ∇∇∇ edge)	mm
UZ	Depth finishing allowance – (only for ∇ or ∇∇∇)	mm
Insertion	Various insertion modes can be selected – (only for plane-by-plane machining method and for ∇, ∇∇∇ and ∇∇∇ edge): • **Predrilled** (only for G code) • **Perpendicular: Insert vertically at the center of pocket** The tool executes the calculated depth infeed vertically at the center of the pocket. Feedrate: Infeed rate as programmed under FZ • **Helical: Insert along helical path** The cutter center point traverses along the helical path determined by the radius and depth per revolution. If the depth for one infeed has been reached, a full circle motion is executed to eliminate the inclined insertion path. Feedrate: Machining feedrate Note: The vertical insertion into pocket center method can be used only if the tool can cut across center or if the workpiece has been predrilled.	
FZ (only for G code)	Depth infeed rate – (only for vertical insertion)	*
FZ (only for ShopMill)	Depth infeed rate – (only for vertical insertion)	mm/min mm/tooth
EP	Maximum pitch of helix - (for helical insertion only) The helix pitch may be lower due to the geometrical situation.	mm/rev

Parameter	Description	
ER	Radius of helix - (for helical insertion only) The radius must not be larger than the milling cutter radius, otherwise material will remain. Also make sure the circular pocket is not violated.	mm
FS	Chamfer width for chamfering - (for chamfering only)	mm
ZFS	Insertion depth of tool tip (abs or inc) - (for chamfering only)	mm

* Unit of feedrate as programmed before the cycle call

Hidden parameters

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.

Parameter	Description	Value	Can be set in SD
PL (only for G code)	Machining plane	Defined in MD 52005	
SC (only for G code)	Safety clearance	1 mm	x
Machining position	Mill circular pocket at the programmed position (X0, Y0, Z0).	Single position	
Solid machining	The pocket is milled from the solid material.	Complete machining	

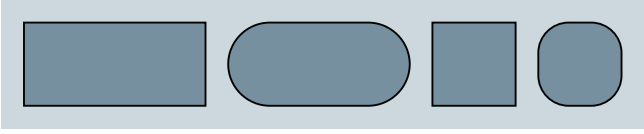
Machine manufacturer
Please refer to the machine manufacturer's specifications.

10.3.4 Rectangular spigot (CYCLE76)

Function

You can mill various rectangular spigots with the "Rectangular spigot" cycle.

You can select from the following shapes with or without a corner radius:



Depending on the dimensions of the rectangular spigot in the workpiece drawing, you can select a corresponding reference point for the rectangular spigot.

In addition to the required rectangular spigot, you must also define a blank spigot, i.e. the outer limits of the material. The tool moves at rapid traverse outside this area. The blank spigot must not overlap adjacent blank spigots and is automatically placed by the cycle in a central position on the finished spigot.

The rectangular spigot is machined using only one infeed. If you want to machine the spigot using multiple infeeds, you must program the "Rectangular spigot" cycle several times with a reducing finishing allowance.

Input simple

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

Machine manufacturer

Various defined values can be pre-assigned using setting data.

Please refer to the machine manufacturer's specifications.

If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

Sequence

1. The tool approaches the starting point at rapid traverse at the height of the retraction plane and is fed in to the safety clearance. The starting point is on the positive X axis rotated through α0.
2. The tool approaches the spigot contour sideways in a semicircle at machining feedrate. The tool first executes infeed at machining depth and then moves in the plane. The rectangular spigot is machined depending on the programmed machining direction (up-cut/down-cut) in a clockwise or counterclockwise direction.
3. When the rectangular spigot has been traversed once, the tool retracts from the contour in a semicircle and then infeed to the next machining depth is performed.
4. The rectangular spigot is approached again in a semicircle and traversed once. This process is repeated until the programmed spigot depth is reached.
5. The tool moves back to the safety clearance at rapid traverse.

Machining type

- Roughing
  Roughing involves moving around the rectangular spigot until the programmed finishing allowance has been reached.
- Finishing
  If you have programmed a finishing allowance, the rectangular spigot is moved around until depth Z1 is reached.
- Chamfering
  Chamfering involves edge breaking at the upper edge of the rectangular spigot.

Note

During chamfering, the end mill behaves like a centering tool with a 90° tip angle.

Procedure

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

2. Press the "Milling" softkey.

3. Press the "Multi-edge spigot" and "Rectangular spigot" softkeys.
   The "Rectangular Spigot" input window opens.

Parameters in the "Input complete" mode

Parameters, G code program				Parameters, ShopMill program		
Input		• complete				
PL	Machining plane			T	Tool name	
	Milling direction			D	Cutting edge number	
RP	Retraction plane	mm		F	Feedrate	mm/min mm/tooth
SC	Safety clearance	mm		S / V	Spindle speed or constant cutting rate	rpm m/min
F	Feedrate	*				

Parameter	Description	Unit
FZ (only for G code)	Depth infeed rate	*
Reference point	The following different reference point positions can be selected: • (center) • (bottom left) • (bottom right) • (top left) • (top right)	
Machining	• ∇ (roughing) • ∇∇∇ (finishing) • Chamfering	
Machining position	• Single position A rectangular spigot is machined at the programmed position (X0, Y0, Z0). • Position pattern Several rectangular spigots are machined in a position pattern (e.g. full circle, pitch circle, grid, etc.).	

Parameter	Description	Unit
	The positions refer to the reference point:	
X0	Reference point X – (for single position only)	mm
Y0	Reference point Y – (for single position only)	mm
Z0	Reference point Z – (single position only and G Code position pattern)	mm
W	Width of spigot	mm
L	Length of spigot	mm
R	Corner radius	mm
α0	Angle of rotation	Degrees
Z1	Spigot depth (abs) or depth relative to Z0 (inc) - (only for ∇ and ∇∇∇)	mm
DZ	Maximum depth infeed - (only for ∇ and ∇∇∇)	mm
UXY	Plane finishing allowance for the length (L) and width (W) of the rectangular spigot. Smaller rectangular spigot dimensions are obtained by calling the cycle again and programming it with a lower finishing allowance. - (only for ∇ and ∇∇∇)	mm
UZ	Depth finishing allowance (tool axis) - (only for ∇ and ∇∇∇)	mm
W1	Width of blank spigot (important for determining approach position) - (only for ∇ and ∇∇∇)	mm
L1	Length of blank spigot (important for determining approach position) - (only for ∇ and ∇∇∇)	mm
FS	Chamfer width for chamfering - (for chamfering only)	mm
ZFS	Insertion depth of tool tip (abs or inc) - (for chamfering only)	mm

* Unit of feedrate as programmed before the cycle call

Parameters in the "Input simple" mode

G code program parameters				ShopMill program parameters		
Input		• simple				
	Milling direction			T	Tool name	
RP	Retraction plane	mm		D	Cutting edge number	
F	Feedrate	*		F	Feedrate	mm/min mm/rev
				S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	
FZ (only for G code)	Depth infeed rate	*
Machining	The following machining operations can be selected: • ∇ (roughing) • ∇∇∇ (finishing) • Chamfering	

Parameter	Description	
	The positions refer to the reference point:	
X0	Reference point X – (single position only)	mm
Y0	Reference point Y – (single position only)	mm
Z0	Reference point Z – (only single position and G Code position pattern)	mm
W	Width of spigot	mm
L	Length of spigot	mm
R	Corner radius	mm
Z1	Depth referred to Z0 (inc) or spigot depth (abs) - (only for ∇ and ∇∇∇ edge)	mm
DZ	Maximum depth infeed – (only for ∇ and ∇∇∇)	mm
UXY	Plane finishing allowance for the length (L) and width (W) of the rectangular spigot. Smaller rectangular spigot dimensions are obtained by calling the cycle again and programming it with a lower finishing allowance. - (only for ∇ and ∇∇∇).	mm
UZ	Depth finishing allowance (tool axis) – (only for ∇ or ∇∇∇)	mm
W1	Width of blank spigot (important for determining approach position) - (only for ∇ and ∇∇∇)	mm
L1	Length of blank spigot (important for determining approach position) - (only for ∇ and ∇∇∇)	mm
FS	Chamfer width for chamfering - (for chamfering only)	mm
ZFS	Insertion depth of tool tip (abs or inc) - (for chamfering only)	mm

* Unit of feedrate as programmed before the cycle call

Hidden parameters

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.

Parameter	Description	Value	Can be set in SD
PL (only for G code)	Machining plane	Defined in MD 52005	
SC (only for G code)	Safety clearance	1 mm	x
Reference point	Position of the reference point: Center		
Machining position	Mill rectangular spigot at the programmed position (X0, Y0, Z0).	Single position	
α0	Angle of rotation	0°	

Machine manufacturer

Please refer to the machine manufacturer's specifications.

10.3.5 Circular spigot (CYCLE77)

Function

You can mill various circular spigots with the "Circular spigot" function.

In addition to the required circular spigot, you must also define a blank spigot, i.e. the outer limits of the material. The tool moves at rapid traverse outside this area. The blank spigot must not overlap adjacent blank spigots and is automatically placed on the finished spigot in a centered position.

The circular spigot is machined using only one infeed. If you want to machine the spigot using multiple infeeds, you must program the "Circular spigot" function several times with a reducing finishing allowance.

Input simple

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

Machine manufacturer

Various defined values can be pre-assigned using setting data.

Please refer to the machine manufacturer's specifications.

If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

Approach/retraction

1. The tool approaches the starting point at rapid traverse at the height of the retraction plane and is fed in to the safety clearance. The starting point is always on the positive X axis.
2. The tool approaches the spigot contour sideways in a semicircle at machining feedrate. The tool first executes infeed at machining depth and then moves in the plane. The circular spigot is machined depending on the programmed machining direction (up-cut/down-cut) in a clockwise or counterclockwise direction.
3. When the circular spigot has been traversed once, the tool retracts from the contour in a semicircle and then infeed to the next machining depth is performed.
4. The circular spigot is approached again in a semicircle and traversed once. This process is repeated until the programmed spigot depth is reached.
5. The tool moves back to the safety clearance at rapid traverse.

Machining type

You can select the machining mode for milling the circular spigot as follows:

- Roughing

  Roughing involves moving round the circular spigot until the programmed finishing allowance has been reached.

- Finishing
  If you have programmed a finishing allowance, the circular spigot is moved around until depth Z1 is reached.

- Chamfering

  Chamfering involves edge breaking at the upper edge of the circular spigot.

Note

During chamfering, the end mill behaves like a centering tool with a 90° tip angle.

Procedure

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

Press the "Milling" softkey.
3.

Press the "Multi-edge spigot" and "Circular spigot" softkeys.
The "Circular Spigot" input window opens.

Parameters in the "Input complete" mode

Parameters, G code program				Parameters, ShopMill program		
Input		• complete				
PL	Machining plane			T	Tool name	
↻	Milling direction			D	Cutting edge number	
RP	Retraction plane	mm		F ↻	Feedrate	mm/min mm/tooth
SC	Safety clearance	mm		S / V ↻	Spindle speed or constant cutting rate	rpm m/min
F	Feedrate	*				

Parameter	Description	Unit
FZ (only for G code)	Depth infeed rate	*
Machining ↻	• ∇ (roughing) • ∇∇∇ (finishing) • Chamfering	
Machining position ↻	• Single position A circular spigot is machined at the programmed position (X0, Y0, Z0). • Position pattern Several circular spigots are machined in a position pattern (e.g. full circle, pitch circle, grid, etc.).	
	The positions refer to the reference point:	
X0	Reference point X – (for single position only)	mm
Y0	Reference point Y – (for single position only)	mm
Z0	Reference point Z – (single position only and G Code position pattern)	mm
∅	Diameter of spigot	mm
Z1 ↻	Spigot depth (abs) or depth relative to Z0 (inc) - (only for ∇ and ∇∇∇)	mm
DZ	Maximum depth infeed - (only for ∇ and ∇∇∇)	mm
UXY	Plane finishing allowance for the length (L) and width (W) of the circular spigot. Smaller circular spigot dimensions are obtained by calling the cycle again and programming it with a lower finishing allowance - (only for ∇ and ∇∇∇).	mm
UZ	Depth finishing allowance (tool axis) - (only for ∇ and ∇∇∇)	mm
∅1	Diameter of blank spigot (important for determining approach position) - (only for ∇ and ∇∇∇)	mm
FS	Chamfer width for chamfering - (for chamfering only)	mm
ZFS ↻	Insertion depth of tool tip (abs or inc) - (for chamfering only)	mm

* Unit of feedrate as programmed before the cycle call

Parameters in the "Input simple" mode

G code program parameters				ShopMill program parameters		
Input ↻		• simple				
↻	Milling direction			T	Tool name	
RP	Retraction plane	mm		D	Cutting edge number	
F	Feedrate	*		F ↻	Feedrate	mm/min mm/rev
				S / V ↻	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	
FZ (only for G code)	Depth infeed rate	*
Machining	The following machining operations can be selected: • ∇ (roughing) • ∇∇∇ (finishing) • Chamfering	
	The positions refer to the reference point:	
X0	Reference point X	mm
Y0	Reference point Y	mm
Z0	Reference point Z	mm
⌀	Diameter of spigot	mm
⌀ 1	Diameter of blank spigot (important for determining approach position) - (only for ∇ and ∇∇∇)	mm
Z1	Depth referred to Z0 (inc) or spigot depth (abs) - (only for ∇ and ∇∇∇)	mm
DZ	Maximum depth infeed – (only for ∇ and ∇∇∇)	mm
UXY	Finishing allowance, plane Smaller circular spigot dimensions are obtained by calling the cycle again and programming it with a lower finishing allowance. - (only for ∇ and ∇∇∇).	mm
UZ	Depth finishing allowance (tool axis) – (only for ∇ or ∇∇∇)	mm
FS	Chamfer width for chamfering - (for chamfering only)	mm
ZFS	Insertion depth of tool tip (abs or inc) - (for chamfering only)	mm

* Unit of feedrate as programmed before the cycle call

Hidden parameters

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.

Parameter	Description	Value	Can be set in SD
PL (only for G code)	Machining plane	Defined in MD 52005	
SC (only for G code)	Safety clearance	1 mm	x
Machining position	Mill circular spigot at the programmed position (X0, Y0, Z0).	Single position	

Machine manufacturer

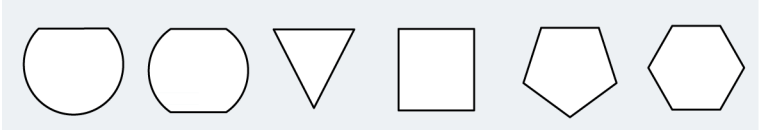
Please refer to the machine manufacturer's specifications.

10.3.6 Multi-edge (CYCLE79)

Function

You can mill a multi-edge with any number of edges with the "Multi-edge" cycle.

You can select from the following shapes with or without a corner radius or chamfer:



Note

Using side mills and saws

When using a side mill (type 150) or a saw (type 151), the first infeed is selected so that the top edge of the tool touches reference point Z0 exactly. At the end of the machining, the tool is extracted completely from the blank spigot. In this way, an inner multi-edge can be produced.

Input simple

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

Machine manufacturer

Various defined values can be pre-assigned using setting data.

Please refer to the machine manufacturer's specifications.

If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

Approach/retraction

1. The tool approaches the starting point at rapid traverse at the height of the retraction plane and is fed in to the safety clearance.
2. The tool traverses the multi-edge in a quadrant at machining feedrate. The tool first executes infeed at machining depth and then moves in the plane. The multi-edge is machined depending on the programmed machining direction (up-cut/down-cut) in a clockwise or counterclockwise direction.
3. When the first plane has been machined, the tool retracts from the contour in a quadrant and then infeed to the next machining depth is performed.

4. The multi-edge is traversed again in a quadrant. This process is repeated until the depth of the multi-edge has been reached.
5. The tool retracts to the safety clearance at rapid traverse.

Note

A multi-edge with more than two edges is traversed helically; with a single or double edge, each edge is machined separately.

Procedure

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

2. Press the "Milling" softkey.

3. Press the "Multi-edge spigot" and "Multi-edge" softkeys.
   The "Multi-edge" input window opens.

Parameters in the "Input complete" mode

Parameters, G code program				Parameters, ShopMill program		
Input		• complete				
PL	Machining plane			T	Tool name	
	Milling direction			D	Cutting edge number	
RP	Retraction plane	mm		F	Feedrate	mm/min mm/tooth
SC	Safety clearance	mm		S / V	Spindle speed or constant cutting rate	rpm m/min
F	Feedrate	*				

Parameter	Description	Unit
FZ (only for G code)	Depth infeed rate	*
Machining	• ∇ (roughing) • ∇∇∇ (finishing) • ∇∇∇ edge (edge finishing) • Chamfering	

Parameter	Description	Unit
Machining position ↻	• Single position A polygon is milled at the programmed position (X0, Y0, Z0). • Position pattern Several polygons are milled at the programmed position pattern (e.g. pitch circle, grid, line).	
	The positions refer to the reference point:	
X0	Reference point X – (for single position only)	mm
Y0	Reference point Y – (for single position only)	mm
Z0	Reference point Z – (single position only and G Code position pattern)	mm
∅	Diameter of blank spigot	mm
N	Number of edges	
SW or L ↻	Width across flats or edge length	mm
α0	Angle of rotation	Degrees
R1 or FS1 ↻	Rounding radius or chamfer width	mm
Z1 ↻	Multi-edge depth (abs) or depth in relation to Z0 (inc) - (only for ∇, ∇∇∇ and ∇∇∇ edge)	mm
DXY ↻	• Maximum plane infeed • Maximum plane infeed as a percentage of the milling cutter diameter - (only for ∇ and ∇∇∇)	mm %
DZ	Maximum depth infeed - (only for ∇ and ∇∇∇)	mm
UXY	Plane finishing allowance - (only for ∇, ∇∇∇ and ∇∇∇ edge)	mm
UZ	Depth finishing allowance – (only for ∇ and ∇∇∇)	mm
FS	Chamfer width for chamfering - (for chamfering only)	mm
ZFS ↻	Insertion depth of tool tip (abs or inc) - (for chamfering only)	mm %

* Unit of feedrate as programmed before the cycle call

Parameters in the "Input simple" mode

G code program parameters				ShopMill program parameters		
Input ↻		• simple				
↻	Milling direction			T	Tool name	
RP	Retraction plane	mm		D	Cutting edge number	
F	Feedrate	*		F ↻	Feedrate	mm/min mm/rev
				S / V ↻	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	
FZ (only for G code)	Depth infeed rate	*
Machining	The following machining operations can be selected: • ∇ (roughing) • ∇∇∇ (finishing) • ∇∇∇ edge (edge finishing) • Chamfering	
	The positions refer to the reference point:	
X0	Reference point X	mm
Y0	Reference point Y	mm
Z0	Reference point Z	mm
∅	Diameter of blank spigot	mm
N	Number of edges	
SW or L	Width across flats or edge length	mm
R1 and FS1	Rounding radius or chamfer width	
Z1	Multi-edge depth (abs) or depth in relation to Z0 (inc) - (only for ∇, ∇∇∇ and ∇∇∇ edge)	mm
DXY	• Maximum plane infeed • Maximum plane infeed as a percentage of the milling cutter diameter - (only for ∇ and ∇∇∇)	mm %
DZ	Maximum depth infeed – (only for ∇ and ∇∇∇)	mm
UXY	Plane finishing allowance - (only for ∇, ∇∇∇ and ∇∇∇ edge)	mm
UZ	Depth finishing allowance (only for ∇ and ∇∇∇)	mm
FS	Chamfer width for chamfering - (for chamfering only)	mm
ZFS	Insertion depth of tool tip (abs or inc) - (for chamfering only)	mm

* Unit of feedrate as programmed before the cycle call

Hidden parameters

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.

Parameter	Description	Value	Can be set in SD
PL (only for G code)	Machining plane	Defined in MD 52005	
SC (only for G code)	Safety clearance	1 mm	x
Machining position	Mill multi-edge at the programmed position (X0, Y0, Z0).	Single position	
α0	Angle of rotation	0°	

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

10.3.7 Longitudinal groove (SLOT1)

Function

You can mill any longitudinal slot with the "longitudinal slot" milling function.

The following machining methods are available:

- Mill longitudinal slot from solid material.
  Depending on the dimensions of the longitudinal slot in the workpiece drawing, you can select a corresponding reference point for the longitudinal slot.
- Predrill longitudinal slot first if, for example, the milling cutter does not cut in the center (e.g. for ShopMill, program the drilling, rectangular pocket and position program blocks in succession).
  In this case, select the predrilling position corresponding to the "Insertion", "Vertical" parameter (see "Procedure").

Input simple

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

Machine manufacturer
Various defined values can be pre-assigned using setting data.
Please refer to the machine manufacturer's specifications.

If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

Approach/retraction

1. The tool moves at rapid traverse to the retraction plane and infeeds at safety clearance.
2. The tool is inserted into the material according to the chosen strategy.
3. The longitudinal slot is always machined with the chosen machining type from inside out.
4. The tool moves back to the safety clearance at rapid traverse.

Machining type

You can select the machining mode for milling the longitudinal groove as follows:

- Roughing
  During roughing, the individual planes of the slot are machined one after the other until depth Z1 is reached.
- Finishing
  During finishing, the edge is always machined first. The slot edge is approached on the quadrant that joins the corner radius. In the last infeed, the base is finished from the center out.

- Edge finishing
  Edge finishing is performed in the same way as finishing, except that the last infeed (finish base) is omitted.
- Chamfering
  Chamfering involves edge breaking at the upper edge of the longitudinal groove.

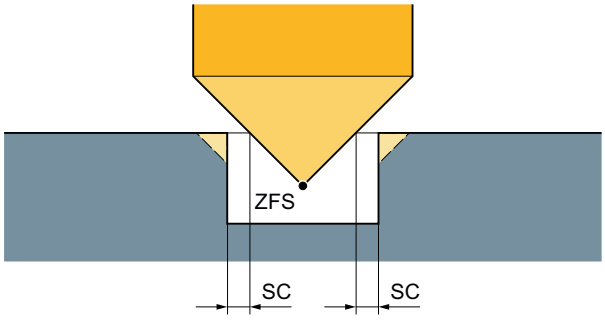


Figure 10-6 Geometries when chamfering inside contours

Note

During chamfering, the end mill behaves like a centering tool with a 90° tip angle.

Note

The following error messages can occur when chamfering inside contours:

- **Safety clearance in the program header too large**
  This error message appears when chamfering would, in principle, be possible with the parameters entered for FS and ZFS, but the safety clearance then could not be maintained.
- **Immersion depth too large**
  This error message appears when chamfering would be possible through the reduction of the immersion depth ZFS.
- **Tool diameter too large**
  This error message appears when the tool would already damage the edges during insertion. In this case, the chamfer FS must be reduced.

Procedure

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

2. Press the "Milling" softkey.

3. Press the "Groove" and "Longitudinal groove" softkeys.
   The "Longitudinal Groove (SLOT1)" input window opens.

Parameters in the "Input complete" mode

G code program parameters				ShopMill program parameters		
Input		• complete				
PL	Machining plane			T	Tool name	
	Milling direction			D	Cutting edge number	
RP	Retraction plane	mm		F	Feedrate	mm/min mm/tooth
SC	Safety clearance	mm		S / V	Spindle speed or constant cutting rate	rpm m/min
F	Feedrate	*				

Parameter	Description	Unit
Reference point	Position of the reference point: • (lefthand edge) • (inside left) • (center) • (inside right) • (righthand edge) 	
Machining	• ∇ (roughing) • ∇∇∇ (finishing) • ∇∇∇ edge (edge finishing) • Chamfering	
Machining position	• Single position A slot is milled at the programmed position (X0, Y0, Z0). • Position pattern Several slots are milled at the programmed position pattern (e.g. pitch circle, grid, line).	
	The positions refer to the reference point:	
X0	Reference point X – (only for single position)	mm
Y0	Reference point Y – (only for single position)	mm
Z0	Reference point Z – (only for single position and G code position pattern)	mm
W	Slot width	mm
L	Slot length	mm

Parameter	Description	Unit
α0	Angle of rotation	Degrees
Z1	Slot depth (abs) or depth relative to Z0 (inc) – (only for ∇, ∇∇∇ and ∇∇∇ edge)	mm
DXY (only for ShopMill)	• Maximum plane infeed • Maximum plane infeed as a percentage of the milling cutter diameter - (only for ∇ and ∇∇∇)	mm %
DZ	Maximum depth infeed - (only for ∇, ∇∇∇ and ∇∇∇ edge)	mm
UXY	Plane finishing allowance - (only for ∇, ∇∇∇ and ∇∇∇ edge)	mm
UZ	Depth finishing allowance (slot base) - (only for ∇ and ∇∇∇)	mm
Insertion	The following insertion modes can be selected: • **Predrilled:** (only for G code) Approach reference point shifted by the amount of the safety clearance with G0. • **Perpendicular:** ShopMill: Depending on the effective milling tool width (milling tool diameter x DXY[%]) or DXY [mm]) the tool is moved to the infeed depth at the pocket center or at the pocket edge. – At the edge of the longitudinal slot ("inside left"): Effective milling tool width >= half the slot width. – At the longitudinal slot center: Effective milling tool width < half the slot width. G code: The tool is inserted to the infeed depth at the reference point "inside left" Note: This setting can be used only if the cutter can cut across center. • **Helical:** Insertion on helical path (only for G code) The cutter center point traverses along the helical path determined by the radius and depth per revolution (helical path). If the depth for one infeed has been reached, a full longitudinal slot is machined to eliminate the inclined insertion path. • **Oscillating:** Insert with oscillation along center axis of longitudinal slot The cutter center point oscillates along a linear path until it reaches the depth infeed. When the depth has been reached, the path is traversed again without depth infeed in order to remove the slope caused by insertion.	mm
FZ (only for G code)	Depth infeed rate – (only for vertical insertion)	*
FZ (only for ShopMill)	Depth infeed rate – (only for vertical insertion)	mm/min mm/tooth
EP (only for G code)	Maximum pitch of helix – (only for helical insertion)	mm/rev
ER (only for G code)	Radius of helix – (only for helical insertion) The radius cannot be any larger than the cutter radius; otherwise, material will remain.	mm
EW	Maximum insertion angle - (only for insertion with oscillation)	Degrees
FS	Chamfer width for chamfering (inc) - (only for chamfering)	mm
ZFS	Insertion depth of tool tip (abs or inc) - (only for chamfering)	mm

Note

Predrilling position

The position at which insertion is performed if "predrilled" is selected, is the same position that you select when specifying the reference point with "left inside". In the case of a slot without an angle of rotation, the predrilled position is the center point of the left rounding radius of the slot. When the cycle is called on a position circle, the predrilled position is always the center point of the rounding radius that is closer to the center point.

* Unit of feedrate as programmed before the cycle call

Parameters in the "Input simple" mode

G code program parameters				ShopMill program parameters		
Input		• simple				
	Milling direction			T	Tool name	
RP	Retraction plane	mm		D	Cutting edge number	
F	Feedrate	*		F	Feedrate	mm/min mm/rev
				S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	
Machining	The following machining operations can be selected: • ∇ (roughing) • ∇∇∇ (finishing) • ∇∇∇ edge (edge finishing) • Chamfering	
X0 Y0 Z0	The positions refer to the reference point: Reference point X Reference point Y Reference point Z	 mm mm mm
W	Slot width	mm
L	Slot length	mm
α0	Angle of rotation	Degrees
Z1	Slot depth (abs) or depth relative to Z0 (inc) – (only for ∇, ∇∇∇ and ∇∇∇ edge)	mm
DXY (only for ShopMill)	• Maximum plane infeed • Maximum plane infeed as a percentage of the milling cutter diameter - (only for ∇ and ∇∇∇)	mm %
DZ	Maximum depth infeed – (only for ∇, ∇∇∇ and ∇∇∇ edge)	mm
UXY	Plane finishing allowance - (only for ∇, ∇∇∇ and ∇∇∇ edge)	mm

Parameter	Description	
UZ	Depth finishing allowance (slot base) - (only for ∇ and ∇∇∇)	mm
Insertion	The following insertion modes can be selected: • Predrilled: (only for G code) Approach reference point shifted by the amount of the safety clearance with G0. • Perpendicular: ShopMill: Depending on the effective milling tool width (milling tool diameter x DXY[%]) or DXY [mm]) the tool is moved to the infeed depth at the pocket center or at the pocket edge. – At the edge of the longitudinal slot ("inside left"): Effective milling tool width >= half the slot width. – At the longitudinal slot center: Effective milling tool width < half the slot width. G code: The tool is inserted to the infeed depth at the reference point "inside left" Note: This setting can be used only if the cutter can cut across center. • Helical: Insertion on helical path (only for G code) The cutter center point traverses along the helical path determined by the radius and depth per revolution (helical path). If the depth for one infeed has been reached, a full longitudinal slot is machined to eliminate the inclined insertion path. • Oscillating: Insert with oscillation along center axis of longitudinal slot The cutter center point oscillates along a linear path until it reaches the depth infeed. When the depth has been reached, the path is traversed again without depth infeed in order to remove the slope caused by insertion.	
FZ (only for G code)	Depth infeed rate – (only for vertical insertion)	*
FZ (only for ShopMill)	Depth infeed rate – (only for vertical insertion)	mm/min mm/tooth
EP (only for G code)	Maximum pitch of helix – (for helical insertion only)	mm/rev
ER (only for G code)	Radius of helix – (for helical insertion only) The radius cannot be any larger than the cutter radius; otherwise, material will remain.	mm
EW	Maximum insertion angle - (only for insertion with oscillation)	Degrees
FS	Chamfer width for chamfering (inc) - (for chamfering only)	mm
ZFS	Insertion depth of tool tip (abs or inc) - (for chamfering only)	mm

Note

Predrilling position

The position at which insertion is performed if "predrilled" is selected, is the same position that you select when specifying the reference point with "left inside". In the case of a slot without an angle of rotation, the predrilled position is the center point of the left rounding radius of the slot. When the cycle is called on a position circle, the predrilled position is always the center point of the rounding radius that is closer to the center point.

* Unit of feedrate as programmed before the cycle call

Hidden parameters

The following parameters are hidden. They are preassigned fixed values or values that can be adjusted using setting data.

Parameter	Description	Value	Can be set in SD
PL (only for G code)	Machining plane	Defined in MD 52005	
SC (only for G code)	Safety clearance	1 mm	x
Reference point	Position of the reference point: Center		
Machining position	Mill slot at the programmed position (X0, Y0, Z0).	Single position	
α0	Angle of rotation	0°	

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

10.3.8 Circumferential groove (SLOT2)

Function

You can mill one or several circumferential slots of equal size on a full or pitch circle with the "circumferential slot" cycle.

Tool size

Please note that there is a minimum size for the milling cutter used to machine the circumferential slot:

- Roughing:
  1/2 slot width W – finishing allowance UXY ≤ milling cutter diameter
- Finishing:
  1/2 slot width W ≤ milling cutter diameter
- Finishing edge:
  Finishing allowance UXY ≤ milling cutter diameter

Annular groove

To create an annular groove, you must enter the following values for the "Number N" and "Aperture angle α1" parameters:

N = 1

α1 = 360°

Clamping the spindle

For ShopTurn, the "Clamp spindle" function can be set up by the machine manufacturer.

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

Activating the damping brake

For ShopTurn, the "Damping brake" function can be set up by the machine manufacturer.

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

Input simple

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

Machine manufacturer
Various defined values can be pre-assigned using setting data.
Please refer to the machine manufacturer's specifications.

If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

Approach/retraction

1. The tool approaches the center point of the semicircle at the end of the slot at rapid traverse at the height of the retraction plane and adjusts to the safety clearance.
2. Then, the tool enters the workpiece at machining infeed (taking into consideration the maximum infeed in the Z direction and the finishing allowance). The circumferential slot is machined in the programmed machining direction (up-cut or down-cut) in a clockwise or counterclockwise direction.
3. When the first circumferential slot is finished, the tool moves to the retraction plane at rapid traverse.
4. The next circumferential slot is approached along a straight line or circular path and then machined.
5. The tool moves back to the safety clearance at rapid traverse.

Machining type

You can select the machining mode for milling the circumferential groove as follows:

- Roughing
  During roughing, the individual planes of the groove are machined one after the other from the center point of the semicircle at the end of the groove until depth Z1 is reached.
- Finishing
  In "Finishing" mode, the edge is always machined first until depth Z1 is reached. The groove edge is approached on the quadrant that joins the radius. In the last infeed, the base is finished from the center point of the semicircle to the end of the groove.
- Edge finishing
  Edge finishing is performed in the same way as finishing, except that the last infeed (finish base) is omitted.
- Chamfering
  Chamfering involves edge breaking at the upper edge of the circumferential groove.

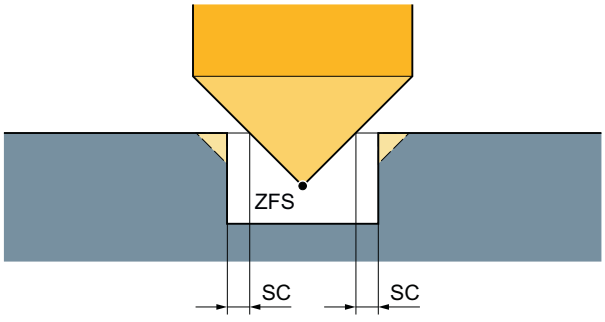


Figure 10-7 Geometries when chamfering inside contours

Note

During chamfering, the end mill behaves like a centering tool with a 90° tip angle.

Note

The following error messages can occur when chamfering inside contours:

- **Safety clearance in the program header too large**
  This error message appears when chamfering would, in principle, be possible with the parameters entered for FS and ZFS, but the safety clearance then could not be maintained.
- **Immersion depth too large**
  This error message appears when chamfering would be possible through the reduction of the immersion depth ZFS.
- **Tool diameter too large**
  This error message appears when the tool would already damage the edges during insertion. In this case, the chamfer FS must be reduced.

Procedure

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

2. Press the "Milling" softkey.

3. Press the "Groove" and "Circumferential groove" softkeys.
   The "Circumferential Groove" input window opens.

Parameters in the "Input complete" mode

Parameters, G code program				Parameters, ShopMill program		
Input		• complete				
PL	Machining plane			T	Tool name	
	Milling direction			D	Cutting edge number	
RP	Retraction plane	mm		F	Feedrate	mm/min mm/tooth
SC	Safety clearance	mm		S / V	Spindle speed or constant cutting rate	rpm m/min
F	Feedrate	*				

Parameter	Description	Unit
FZ (only for ShopMill)	Depth infeed rate (only for ∇ and ∇∇∇)	mm/min in/tooth
FZ (only for G code)	Depth infeed rate (only for ∇ and ∇∇∇)	*
Machining	• ∇ (roughing) • ∇∇∇ (finishing) • ∇∇∇ edge (edge finishing) • Chamfering	
Circular pattern	• Full circle The circumferential slots are positioned around a full circle. The distance from one circumferential slot to the next circumferential slot is always the same and is calculated by the control. • Pitch circle The circumferential slots are positioned around a pitch circle. The distance from one circumferential slot to the next circumferential slot can be defined using angle α2.	

Parameter	Description	Unit
	The positions refer to the center point:	
X0	Reference point X	mm
Y0	Reference point Y	mm
Z0	Reference point Z	mm
N	Number of grooves	
R	Radius of circumferential slot	mm
α0	Starting angle	Degrees
α1	Opening angle of the slot	Degrees
α2	Advance angle - (for pitch circle only)	Degrees
W	Slot width	mm
Z1	Slot depth (abs) or depth relative to Z0 (inc) – (only for ∇, ∇∇∇ and ∇∇∇ edge)	mm
DZ	Maximum depth infeed - (only for ∇, ∇∇∇ and ∇∇∇ Rand)	mm
FS	Chamfer width for chamfering (inc) - (for chamfering only),	mm
ZFS	Insertion depth of tool tip (abs or inc) - (for chamfering only),	mm
UXY	Plane finishing allowance - (only for ∇, ∇∇∇ and ∇∇∇ edge)	mm
Positioning	Positioning motion between the slots: • Straight line: Next position is approached linearly in rapid traverse. • Circular: Next position is approached along a circular path at the feedrate defined in the machine data.	

* Unit of feedrate as programmed before the cycle call

Parameters in the "Input simple" mode

G code program parameters				ShopMill program parameters		
Input		• simple				
	Milling direction			T	Tool name	
RP	Retraction plane	mm		D	Cutting edge number	
F	Feedrate	*		F	Feedrate	mm/min mm/rev
				S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	
FZ (only for G code)	Depth infeed rate (only for ∇ and ∇∇∇)	*
FZ (only for ShopMill)	Depth infeed rate (only for ∇ and ∇∇∇)	mm/min in/tooth

Parameter	Description	
Machining	The following machining operations can be selected: • ∇ (roughing) • ∇∇∇ (finishing) • ∇∇∇ edge (edge finishing) • Chamfering	
Circular pattern	• Full circle The circumferential slots are positioned around a full circle. The distance from one circumferential slot to the next circumferential slot is always the same and is calculated by the control. • Pitch circle The circumferential slots are positioned around a pitch circle. The distance from one circumferential slot to the next circumferential slot can be defined using angle α2.	
	The positions refer to the center point:	
X0	Reference point X	mm
Y0	Reference point Y	mm
Z0	Reference point Z	mm
N	Number of edges	
R	Radius of circumferential slot	mm
α1	Opening angle of the slot	Degrees
α2	Advance angle - (for pitch circle only)	Degrees
W	Slot width	mm
Z1	Slot depth (abs) or depth in relation to Z0 (inc) - (only for ∇, ∇∇∇ and ∇∇∇ edge)	mm
DZ	Maximum depth infeed – (only for ∇ and ∇∇∇ edge)	mm
FS	Chamfer width for chamfering - (for chamfering only)	mm
ZFS	Insertion depth of tool tip (abs or inc) - (for chamfering only)	mm
UXY	Plane finishing allowance - (only for ∇, ∇∇∇ and ∇∇∇ edge)	mm
Positioning	Positioning motion between the slots: • Straight line: Next position is approached linearly in rapid traverse. • Circular: Next position is approached along a circular path at the feedrate defined in the machine data.	

* Unit of feedrate as programmed before the cycle call

Hidden parameters

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.

Parameter	Description	Value	Can be set in SD
PL (only for G code)	Machining plane	Defined in MD 52005	
SC (only for G code)	Safety clearance	1 mm	x
α0	Angle of rotation / starting angle	0°	

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

10.3.9 Open groove (CYCLE899)

Function

Use the "Open slot" function if you want to machine open slots.

For roughing, you can choose between the following machining strategies, depending on your workpiece and machine properties:

- Vortex milling
- Plunge milling

The following machining types are available to completely machine the slot:

- Roughing
- Rough-finishing
- Finishing
- Finishing the base
- Finishing wall
- Chamfering

Vortex milling

Particularly where hardened materials are concerned, this process is used for roughing and contour machining using coated VHM milling cutters.

Vortex milling is the preferred technique for HSC roughing, as it ensures that the tool is never completely inserted. This means that the set overlap is precisely maintained.

Plunge milling

Plunge cutting is the preferred method of machining slots for "unstable" machines and workpiece geometries. This method generally only exerts forces along the tool axis, i.e. perpendicular to the surface of the pocket/slot to be machined (with the XY plane in Z direction). Therefore, the tool is subject to virtually no deformation. As a result of the axial loading of the tool, there is hardly any danger of vibration occurring for unstable workpieces.

The cutting depth can be considerably increased. The so-called plunge cutter ensures a longer service life due to less vibration for long overhangs.

Input simple

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

Machine manufacturer

Various defined values can be pre-assigned using setting data.

Please refer to the machine manufacturer's specifications.

If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

Clamping the spindle

For ShopTurn, the "Clamp spindle" function can be set up by the machine manufacturer.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Activating the damping brake

For ShopTurn, the "Damping brake" function can be set up by the machine manufacturer.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Approach/retraction for vortex milling

1. The tool approaches the starting point in front of the slot in rapid traverse and maintains the safety clearance.
2. The tool goes to the cutting depth.
3. The open slot is always machined along its entire length using the selected machining method.
4. The tool retracts to the safety clearance in rapid traverse.

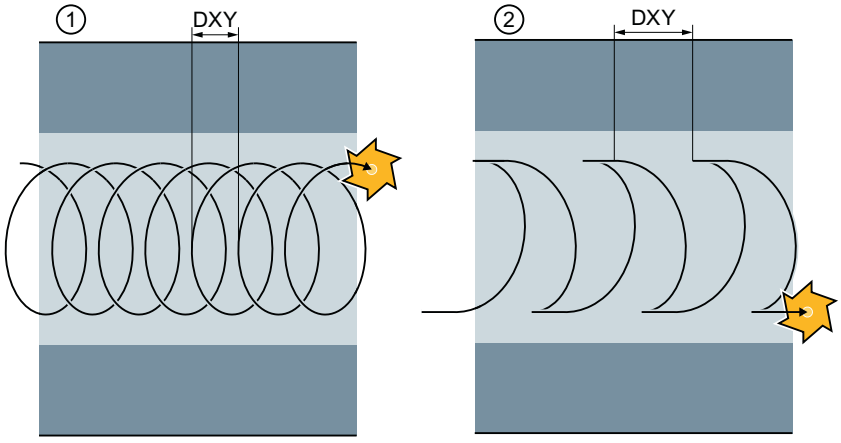
Approach/retraction for plunge cutting

1. The tool moves in rapid traverse to the starting point in front of the slot at the safety clearance.
2. The open slot is always machined along its entire length using the selected machining method.
3. The tool retracts to the safety clearance in rapid traverse.

Machining type, roughing vortex milling

Roughing is performed by moving the milling cutter along a circular path.

While performing this motion, the milling cutter is continuously fed into the plane. Once the milling cutter has traveled along the entire slot, it returns to its starting point, while continuing to move in a circular fashion. By doing this, it removes the next layer (infeed depth) in the Z direction. This process is repeated until the set slot depth plus the finishing allowance has been reached.



① Vortex milling: Down-cut or up-cut
② Vortex milling: Down-cut-up-cut

Supplementary conditions for vortex milling

- Roughing
  1/2 slot width W – finishing allowance UXY ≤ milling cutter diameter
- Slot width
  minimum 1.15 x milling cutter diameter + finishing allowance
  maximum, 2 x milling cutter diameter + 2 x finishing allowance
- Radial infeed
  minimum, 0.02 x milling cutter diameter
  maximum, 0.25 x milling cutter diameter
- Maximum infeed depth ≤ cutting height of milling cutter

Please note that the cutting height of the milling cutter cannot be checked.

The maximum radial infeed depends on the milling cutter.

For hard materials, use a lower infeed.

Machining type, roughing plunge cutting

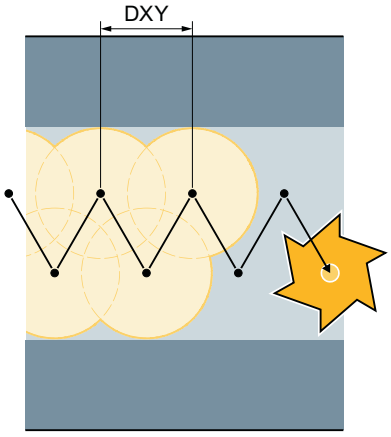
Roughing of the slot takes place sequentially along the length of the groove, with the milling cutter performing vertical insertions at the machining feedrate. The milling cutter is then retracted and repositioned at the next insertion point.

The milling cutter moves along the length of the slot, at half the infeed rate, and inserts alternately at the left-hand and right-hand walls.

The first insertion motion takes place at the slot edge, with the milling cutter inserted at half the infeed, less the safety clearance (if the safety clearance is greater than the infeed, this will be on the outside). For this cycle, the maximum width of the slot must be less than double the width of the milling cutter + the finishing allowance.

Following each insertion, the milling cutter is lifted by the height of the safety clearance at the machining feedrate. As far as possible, this occurs during what is known as the retraction process, i.e. if the milling cutter's wrap angle is less than 180°, it is lifted at an angle below 45° in the opposite direction to the bisector of the wrap area.

The milling cutter then traverses over the material in rapid traverse.



Supplementary conditions for plunge cutting

- Roughing
  1/2 slot width W - finishing allowance UXY ≤ milling cutter diameter
- Maximum radial infeed
  The maximum infeed depends on the cutting edge width of the milling cutter.
- Increment
  The lateral increment is calculated on the basis of the required slot width, milling cutter diameter and finishing allowance.
- Retraction
  Retraction involves the milling cutter being retracted at a 45° angle if the wrap angle is less than 180°. Otherwise, retraction is perpendicular, as is the case with drilling.
- Retraction
  Retraction is performed perpendicular to the wrapped surface.
- Safety clearance
  Traverse through the safety clearance beyond the end of the workpiece to prevent rounding of the slot walls at the ends.

Please note that the milling cutter’s cutting edge cannot be checked for the maximum radial infeed.

Machining type, rough finishing

If there is too much residual material on the slot walls, unwanted corners are removed to the finishing dimension.

Machining type, finishing

When finishing walls, the milling cutter travels along the slot walls, whereby just like for roughing, it is again fed in the Z direction, increment by increment. During this process, the milling cutter travels through the safety clearance beyond the beginning and end of the slot, so that an even slot wall surface can be guaranteed across the entire length of the slot.

Machining type, edge finishing

Edge finishing is performed in the same way as finishing, except that the last infeed (finish base) is omitted.

Machining type, finishing base

When finishing the base, the milling cutter moves backwards and forwards once in the finished slot.

Machining type, chamfering

Chamfering involves breaking the edge at the upper slot edge.

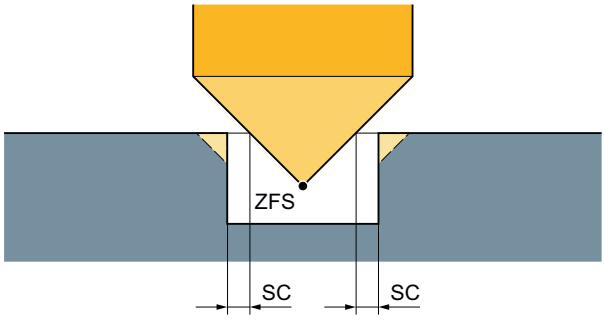


Figure 10-8 Geometries when chamfering inside contours

Note

During chamfering, the end mill behaves like a centering tool with a 90° tip angle.

Note

The following error messages can occur when chamfering inside contours:

- **Safety clearance in the program header too large**
  This error message appears when chamfering would, in principle, be possible with the parameters entered for FS and ZFS, but the safety clearance then could not be maintained.
- **Immersion depth too large**
  This error message appears when chamfering would be possible through the reduction of the immersion depth ZFS.
- **Tool diameter too large**
  This error message appears when the tool would already damage the edges during insertion. In this case, the chamfer FS must be reduced.

Additional supplementary conditions

- Finishing
  1/2 slot width W ≤ milling cutter diameter
- Edge finishing
  Finishing allowance UXY ≤ milling cutter diameter
- Chamfering
  The tip angle must be entered into the tool table.

Procedure

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

2. Press the "Milling" softkey.

3. Press the "Slot" and "Open slot" softkeys.
   The "Open slot" input window opens.

Parameters in the "Input complete" mode

Parameters, G code program				Parameters, ShopMill program		
Input		• complete				
PL	Machining plane			T	Tool name	
RP	Retraction plane	mm		D	Cutting edge number	
SC	Safety clearance	mm		F	Feedrate	mm/min mm/tooth
F	Feedrate	*		S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
Reference point	Position of the reference point: • (lefthand edge) • (center) • (righthand edge) 	
Machining	• ∇ (roughing) • ∇∇ (pre-finishing) • ∇∇∇ (finishing) • ∇∇∇ base (base finishing) • ∇∇∇ edge (edge finishing) • Chamfering	
Technology	• Vortex milling The milling cutter performs circular motions along the length of the slot and back again. • Plunge cutting Sequential drilling motion along the tool axis.	
	Milling direction - (except plunge cutting) • Climbing cutting • Conventional cutting • Climbing-conventional cutting	
Machining position	• Single position Mill a slot at the programmed position (X0, Y0, Z0). • Position pattern Mill slots at a programmed position pattern (e.g. full circle or grid).	

Parameter	Description	Unit
	The positions refer to the reference point:	
X0	Reference point X – (for single position only)	mm
Y0	Reference point Y – (for single position only)	mm
Z0	Reference point Z – (single position only and G Code position pattern)	mm
W	Slot width	mm
L	Slot length	mm
α0	Angle of rotation of slot	Degrees
Z1	Slot depth (abs) or depth relative to Z0 (abs) – (only for ∇, ∇∇∇, ∇∇∇ base and ∇∇∇ rough finishing)	mm
DXY 	• Maximum plane infeed • Maximum plane infeed as a percentage of the milling cutter diameter - (only for ∇)	mm %
DZ	Maximum depth infeed - (only for ∇, ∇∇∇ rough finishing, ∇∇∇ and ∇∇∇ edge) - (only for vortex milling)	mm
UXY	Plane finishing allowance (slot edge) - (only for ∇, ∇∇∇ rough finishing and ∇∇∇ base)	mm
UZ	Depth finishing allowance (slot base) - (only for ∇, ∇∇∇ rough finishing and ∇∇∇ edge)	mm
FS	Chamfer width for chamfering (inc) - (for chamfering only)	mm
ZFS	Insertion depth of tool tip (abs or inc) - (for chamfering only)	mm

* Unit of feedrate as programmed before the cycle call

Parameters in the "Input simple" mode

G code program parameters				ShopMill program parameters		
Input 		• simple				
	Milling direction			T	Tool name	
RP	Retraction plane	mm		D	Cutting edge number	
F	Feedrate	*		F	Feedrate	mm/min mm/rev
				S / V 	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	
Machining	The following machining operations can be selected: • ∇ (roughing) • ∇∇ (pre-finishing) • ∇∇∇ (finishing) • ∇∇∇ base (base finishing) • ∇∇∇ edge (edge finishing) • Chamfering	
Technology	• Vortex milling The milling cutter performs circular motions along the length of the slot and back again. • Plunge cutting Sequential drilling motion along the tool axis.	
	Milling direction: - (except plunge cutting). • Climbing cutting • Conventional cutting • Climbing-conventional cutting	
	The positions refer to the reference point:	
X0	Reference point X	mm
Y0	Reference point Y	mm
Z0	Reference point Z	mm
W	Slot width	
L	Slot length	mm
Z1	Slot depth (abs) or depth in relation to Z0 (inc)	mm
DXY	• Maximum plane infeed • Maximum plane infeed as a percentage of the milling cutter diameter- (only for ∇ and ∇∇∇)	mm %
DZ	Maximum depth infeed - (only for ∇, ∇∇ pre-finishing, ∇∇∇ and ∇∇∇ edge)- (only for vortex milling)	mm
UXY	Plane finishing allowance (slot base) - (only for ∇, ∇∇ pre-finishing and ∇∇∇ base)	mm
UZ	Plane finishing allowance (slot edge) - (only for ∇, ∇∇ pre-finishing and ∇∇∇ edge)	mm
FS	Chamfer width for chamfering (inc) - (for chamfering only)	mm
ZFS	Insertion depth of tool tip (abs or inc) - (for chamfering only)	mm

* Unit of feedrate as programmed before the cycle call

Hidden parameters

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.

Parameter	Description	Value	Can be set in SD
PL (only for G code)	Machining plane	Defined in MD 52005	
SC (only for G code)	Safety clearance	1 mm	x
Reference point	Position of the reference point: Center		
Machining position	Mill slot at the programmed position (X0, Y0, Z0).	Single position	
α0	Angle of rotation	0°	

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

10.3.10 Long hole (LONGHOLE) - only for G code programs

Function

In contrast to the groove, the width of the elongated hole is determined by the tool diameter.

Internally in the cycle, an optimum traversing path of the tool is determined, ruling out unnecessary idle passes. If several depth infeeds are required to machine an elongated hole, the infeed is carried out alternately at the end points. The path to be traversed in the plane along the longitudinal axis of the elongated hole changes its direction after each infeed. The cycle searches for the shortest path when changing to the next elongated hole.

Note

The cycle requires a milling cutter with a "face tooth cutting over center" (DIN 844).

Approach/retraction

1. Using G0, the starting position for the cycle is approached. In both axes of the current plane, the closest end point of the first elongated hole to be machined is approached at the level of the retraction plane in the tool axis and then lowered to the reference point shifted by the amount of the safety clearance.
2. Each elongated hole is milled in a reciprocating motion. The machining in the plane is performed using G1 and the programmed feedrate. At each reversal point, the infeed to the next machining depth calculated internally in the cycle is performed with G1 and the feedrate, until the final depth is reached.

3. Retraction to the retraction plane using G0 and approach to the next elongated hole on the shortest path.
4. After the last elongated hole has been machined, the tool at the position reached last in the machining plane is moved with G0 to the retraction plane, and the cycle terminated.

Procedure

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

2. Press the "Milling" softkey.

3. Press the "Groove" and "Elongated hole" softkeys.
   The "Elongated Hole" input window opens.

Parameter	Description	Unit
PL	Machining plane	
RP	Retraction plane (abs)	
SC	Safety clearance (inc)	
F	Feedrate	*
Machining type	• **Plane-by-plane** The tool is inserted to infeed depth in the pocket center. Note: This setting can be used only if the cutter can cut across center. • **Oscillating** Insert with oscillation along center axis of longitudinal slot**:** The cutter center point oscillates along a linear path until it reaches the depth infeed. When the depth has been reached, the path is traversed again without depth infeed in order to eliminate the inclined insertion path.	mm
Reference point	Position of the reference point: 	
Machining position	• Single position An elongated hole is machined at the programmed position (X0, Y0, Z0). • Position pattern Several elongated holes are machined in the programmed position pattern (e.g. pitch circle, grid, line).	

Parameter	Description	Unit
	The positions refer to the reference point:	
X0	Reference point X – (for single position only)	mm
Y0	Reference point Y – (for single position only)	mm
Z0	Reference point Z	mm
L	Elongated hole length	mm
α0	Angle of rotation	Degrees
Z1	Elongated hole depth (abs) or depth in relation to Z0 (inc)	mm
DZ	Maximum depth infeed	mm
FZ	Depth infeed rate	*

* Unit of feedrate as programmed before the cycle call

10.3.11 Thread milling (CYCLE70)

Function

Using a thread cutter, internal or external threads can be machined with the same pitch. Threads can be machined as right-hand or left-hand threads and from top to bottom or vice versa.

For metric threads (thread pitch P in mm/rev), the cycle assigns a value (calculated on the basis of the thread pitch) to the thread depth H1 parameter. You can change this value. The default selection must be activated via a machine data code.

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

The entered feedrate acts on the workpiece contour, i.e. it refers to the thread diameter. However the feedrate of the cutter center point is displayed. That is why a smaller value is displayed for internal threads and a larger value is displayed for external threads than was entered.

Clamping the spindle

For ShopTurn, the "Clamp spindle" function can be set up by the machine manufacturer.

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

Activating the damping brake

For ShopTurn, the "Damping brake" function can be set up by the machine manufacturer.

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

Approach/retraction when milling internal threads

1. Positioning on retraction plane with rapid traverse.
2. Approach of starting point of the approach circle in the current plane with rapid traverse.

   Note

   The safety clearance SC at the side of the thread is taken into account. If the safety margin cannot be complied with because the thread is only a little larger than the tool, the tool will be positioned at the thread center.

3. Infeed to a starting point in the tool axis calculated internally in the controller with rapid traverse.

   Note

   Depending on the machining direction (Z0 → Z1 or Z1 → Z0), the start position can be at Z0 or Z1.

4. Approach motion to thread diameter on an approach circle calculated internally in the controller with the programmed feedrate, taking into account the finishing allowance and maximum plane infeed.
5. Thread cutting along a spiral path in clockwise or counter-clockwise direction (depending on whether it is left-hand/right-hand thread, for number of cutting teeth of a milling plate (NT) ≥ 2 only one rotation, offset in the Z direction).
   To reach the programmed thread length, traversing is beyond the Z1 value for different distances depending on the thread parameters.
6. Exit motion along a circular path in the same rotational direction at programmed feedrate.
7. With a programmed number of threads per cutting edge NT > 2, the tool is fed in (offset) by the amount NT-1 in the Z direction. Points 4 to 7 are repeated until the programmed thread depth is reached.
8. If the plane infeed is less than the thread depth, points 3 to 7 are repeated until the thread depth + programmed allowance is reached.
9. Retraction on the retraction plane in the tool axis with rapid traverse.
10. Rapid traverse thread center point approach with position pattern (MCALL).

Please note that when milling an internal thread the tool must not exceed the following value:

Milling cutter diameter < (nominal diameter - 2 · thread depth H1)

Approach/retraction when milling external threads

1. Positioning on retraction plane with rapid traverse.
2. Approach of starting point of the approach circle in the current plane with rapid traverse.

   Note

   The safety clearance SC at the side of the thread is taken into account.

3. Infeed to a starting point in the tool axis calculated internally in the controller with rapid traverse.

   Note

   Depending on the machining direction (Z0 → Z1 or Z1 → Z0), the start position can be at Z0 or Z1.

4. Approach motion to thread core diameter on an approach circle calculated internally in the controller with the programmed feedrate, taking into account the finishing allowance and maximum plane infeed.
5. Cut thread along a spiral path in clockwise or counter-clockwise direction (depending on whether it is left-hand/right-hand thread, with NT ≥ 2 only one rotation, offset in Z direction). To reach the programmed thread length, traversing is beyond the Z1 value for different distances depending on the thread parameters.
6. Exit motion along a circular path in opposite rotational direction at programmed feedrate.
7. With a programmed number of threads per cutting edge NT > 2, the tool is fed in (offset) by the amount NT-1 in the Z direction. Points 4 to 7 are repeated until the programmed thread depth is reached.
8. If the plane infeed is less than the thread depth, points 3 to 7 are repeated until the thread depth + programmed allowance is reached.
9. Retraction on the retraction plane in the tool axis with rapid traverse.

Procedure

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

2. Press the "Milling" softkey.

3. Press the "Thread milling" softkey.
   The "thread milling" input window opens.

Parameters, G code program				Parameters, ShopMill program		
PL	Machining plane			T	Tool name	
RP	Retraction plane	mm		D	Cutting edge number	
SC	Safety distance acts laterally to the thread and not in the tool axis	mm		F	Feedrate	mm/min mm/tooth
F	Feedrate	*		S / V	Spindle speed or constant cutting speed	rpm m/min

Parameter	Description	Unit
Machining	• ∇ (roughing) • ∇∇∇ (finishing)	
	Machining direction: • Z0 → Z1 Machining from top to bottom • Z1 → Z0 Machining from bottom to top	
	Direction of rotation of the thread: • Right-hand thread A right-hand thread is cut. • Left-hand thread A left-hand thread is cut.	
	Position of the thread: • Internal thread An internal thread is cut. • External thread An external thread is cut.	
NT	Number of teeth per cutting edge Single or multiple toothed milling inserts can be used. The motions required are executed by the cycle internally, so that the tip of the bottom tooth on the milling insert corresponds to the programmed end position when the thread end position is reached. Depending on the cutting edge geometry of the milling insert, the retraction path must be taken into account at the base of the workpiece.	
(only for G code)	Machining position: • Single position • Position pattern (MCALL)	
X0 Y0 Z0 (only for G code)	The positions refer to the center point: Reference point X – (for single position only) Reference point Y – (for single position only) Reference point Z	 mm mm mm
Z1	End point of the thread (abs) or thread length (inc)	mm
Table	Thread table selection: • Without • ISO metric • Whitworth BSW • Whitworth BSP • UNC	
Selection – (not for table "without")	Selection of table value: e.g. • M3; M10; etc. (ISO metric) • W3/4"; etc. (Whitworth BSW) • G3/4"; etc. (Whitworth BSP) • N1" - 8 UNC; etc. (UNC)	

Parameter	Description	Unit
P	Display of the thread pitch for the parameter input in the input field "Table" and "Selection".	MODULUS Turns/" mm/rev inch/rev
P - (selection option only for table selection "without")	Pitch ... • in MODULUS: For instance, used for worms that mesh with a gearwheel. • per inch: Used with pipe threads, for example. When entered per inch, enter the integer number in front of the decimal point in the first parameter field and the figures after the decimal point as a fraction in the second and third field. • in mm/rev • in inch/rev The tool used depends on the thread pitch.	MODUL Turns/" mm/rev in/rev
∅	Nominal diameter Example: Nominal diameter of M12 = 12 mm	mm
H1	Thread depth	mm
αS	Starting angle	Degrees
rev	Finishing allowance in X and Y - (only for ∇)	mm

* Unit of feedrate as programmed before the cycle call

10.3.12 Engraving (CYCLE60)

Function

The "Engraving" function is used to engrave a text on a workpiece along a line or arc.

You can enter the text directly in the text field as "fixed text" or assign it via a variable as "variable text".

Engraving uses a proportional font, i.e. the individual characters are of different widths.

Clamping the spindle

For ShopTurn, the "Clamp spindle" function can be set up by the machine manufacturer.

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

Activating the damping brake

For ShopTurn, the "Damping brake" function can be set up by the machine manufacturer.

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

Approach/retraction

1. The tool approaches the starting point at rapid traverse at the height of the retraction plane and adjusts to the safety clearance.
2. The tool moves to the machining depth FZ at the infeed feedrate Z1 and mills the characters.
3. The tool retracts to the safety clearance at rapid traverse and moves along a straight line to the next character.
4. Steps 2 and 3 are repeated until the entire text has been milled.
5. The tool moves to the retraction plane in rapid traverse.

Procedure

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

2. Press the "Milling" softkey.

3. Press the "Engraving" softkey.
   The "Engraving" input window opens.

Entering the engraving text

4. Press the "Special characters" softkey if you need a character that does not appear on the input keys.
   The "Special characters" window appears.
   - Position the cursor on the desired character.
   - Press the "OK" softkey.

   The selected character is inserted into the text at the cursor position.

5. If you wish to delete the complete text, press the "Delete text" and "Delete" softkeys one after the other.

6. Press the "Lowercase" softkey to enter lowercase letters. Press it again to enter uppercase letters.

7. Press the "Variable" and "Date" softkeys if you want to engrave the current date.

Date

The data is inserted in the European date format (<DD>.<MM>.<YYYY>).

To obtain a different date format, you must adapt the format specified in the text field. For example, to engrave the date in the American date format (month/day/year => 8/16/04), change the format to <M>/<D>/<YY> .

Variable

7. Press the "Variable" and "Time" softkeys if you want to engrave the current time.

Time

The time is inserted in the European format (<TIME24>).

To have the time in the American format, change the format to <TIME12>.

Example:

Text entry: Time: <TIME24> Execute: Time: 16.35

Time: <TIME12> Execute: Time: 04.35 PM

Variable

7.
- Press the "Variable" and "Workpiece count 000123" softkeys to engrave a workpiece count with a fixed number of digits and leading zeroes.
  The format text <######,_$AC_ACTUAL_PARTS> is inserted and you return to the engraving field with the softkey bar.

Quantity 000123

- Define the number of digits by adjusting the number of place holders (#) in the engraving field.
  If the specified number of positions (e.g. ##) is not sufficient to represent the unit quantity, then the cycle automatically increases the number of positions.

- OR

Variable

7.
- Press the "Variable" and "Workpiece count 123" softkeys if you want to engrave a workpiece count without lead zeroes.
  The format text <#,_$AC_ACTUAL_PARTS> is inserted and you return to the engraving field with the softkey bar.

Quantity 123

- Define the number of digits by adjusting the number of place holders in the engraving field.
  If the specified number of digits is not enough to display the workpiece count (e.g. 123), the cycle will automatically increase the number digits.

Variable

7.
- Press the "Variable" and "Number 123.456" softkeys if you want to engrave a any number in a certain format.
  The format text <#.###,_VAR_NUM> is inserted and you return to the engraving field with the softkey bar.

Number 123.456

- The place holders #.### define the digit format in which the number defined in _VAR_NUM will be engraved.
  For example, if you have stored 12.35 in _VAR_NUM, you can format the variable as follows.

Entry	Release	Meaning
<#,_VAR_NUM>	12	Places before decimal point unformatted, no places after the decimal point
<####,_VAR_NUM>	0012	4 places before decimal point, leading zeros, no places after the decimal point

< #,_VAR_NUM>	12	4 places before decimal point, leading blanks, no places after the decimal point
<#.,_VAR_NUM>	12.35	Integer and fractional digits not formatted.
<#.#,_VAR_NUM>	12.4	Places before decimal point unformatted, 1 place after the decimal point (rounded)
<#.##,_VAR_NUM>	12.35	Places before decimal point unformatted, 2 places after the decimal point (rounded)
<#.####,_VAR_NUM>	12.3500	Places before decimal point unformatted, 4 places after the decimal point (rounded)

If there is insufficient space in front of the decimal point to display the number entered, it is automatically extended. If the specified number of digits is larger than the number to be engraved, the output format is automatically filled with the appropriate number of leading and trailing zeroes.

Instead of the decimal point you can also use a blank.

Instead of _VAR_NUM you can use any other numerical variable (e.g. R0).

7. Press the "Variable" and "Variable text" softkeys if you want to take the text to be engraved (up to 200 characters) from a variable.

   The format text <Text, _VAR_TEXT> is inserted and you return to the engraving field with the softkey bar.

   You can use any other text variable instead of _VAR_TEXT.

Note

Entering the engraving text

Only single-line entries without line break are permissible!

Variable texts

There are various ways of defining variable text:

- Date and time
  For example, you can engrave the time and date of manufacture on a workpiece. The values for date and time are read from the NCK.
- Quantity

  Using the workpiece variables you can assign a consecutive number to the workpieces. You can define the format (number of digits, leading zeroes).
  The place holder (#) is used to format the number of digits at which the workpiece counts output will begin.
  If you do not want to output a count of 1 for the first workpiece, you can specify an additive value (e.g., <#,$AC_ACTUAL_PARTS + 100>). The workpiece count output is then incremented by this value (e. g. 101, 102, 103,...).
- Numbers

  When outputting number (e. g. measurement results), you can select the output format (digits either side of the point) of the number to be engraved.
- Text

  Instead of entering a fixed text in the engraving text field, you can specify the text to be engraved via a text variable (e. g., _VAR_TEXT="ABC123").

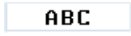
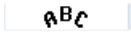
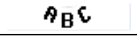
Mirror writing

You can engrave the text mirrored on the workpiece.

Full circle

If you want to distribute the characters evenly around a full circle, enter the arc angle α2=360°. The cycle then distributes the characters evenly around the full circle.

Parameters, G code program				**Parameters, ShopMill program**		
PL	Machining plane			T	Tool name	
RP	Retraction plane	mm		D	Cutting edge number	
SC	Safety clearance	mm		F	Feedrate	mm/min mm/tooth
F	Feedrate	*		S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
FZ (only for G code)	Depth infeed rate	*
FZ (only for ShopMill)	Depth infeed rate	mm/min mm/tooth
Alignment	•  (linear alignment) •  (curved alignment) •  (curved alignment)	
Reference point	Position of the reference point • bottom left • bottom center • bottom right • top left • top center • top right • left-hand edge • center • right-hand edge	
Mirror writing	• Yes The mirrored text is engraved on the workpiece. • No The text is engraved on the workpiece without mirroring.	
Engraving text	maximum 100 characters	
X0 or R	Reference point X (abs) or reference point length polar – (in ShopMill for curved alignment only)	mm
Y0 or α0	Reference point Y (abs) or reference point angle polar – (in ShopMill for curved alignment only)	mm or degrees
X0	Reference point Y (abs) (for linear alignment only)	mm
Y0	Reference point Y (abs) (for linear alignment only)	mm
Z0	Reference point Z (abs)	mm
Z1	Engraving depth (abs) or depth referred to Z0 (inc)	mm
W	Character height	mm
DX1 or α2	Distance between characters or angle of opening – (for curved alignment only)	mm or degrees
DX1 or DX2	Distance between characters or total width – (for linear alignment only)	mm
α1	Text direction (for linear alignment only)	Degrees
XM or LM (only for G code)	Center point X (abs) or center point length polar – (for curved alignment only)	mm
YM or αM (only for G code)	Center point Y (abs) or center point angle polar – (for curved alignment only)	mm or degrees

Parameter	Description	Unit
XM (ShopMill only)	Center point X (abs) – (only for curved alignment)	mm
YM (ShopMill only)	Center point Y (abs) – (only for curved alignment)	mm

* Unit of feedrate as programmed before the cycle call

10.4 Contour milling

10.4.1 General

Function

You can mill simple or complex contours with the "Contour milling" cycle. You can define open contours or closed contours (pockets, islands, spigots).

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.

With contour milling, you must always program the geometry of the contour before you program the technology.

10.4.2 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.

ShopMill program

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 symbols adjacent to the graphics window. They appear in the order in which they were entered.

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

Contour element	Symbol	Meaning
Straight line left Straight line right		Straight line in 90° grid Straight line in 90° grid
Straight line in any direction		Straight line with any gradient
Arc right Arc left		Circle Circle
Pole		Straight diagonal or circle in polar coordinates
Finish contour	END	End of contour definition

The different colors of the symbols 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.

10.4.3 Creating a new contour

Function

For each contour that you want to mill, you must create a new contour.

The contours are stored at the end of the program.

Note

When programming in the G code, it must be ensured that the contours are located after the end of program identifier!

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 tool axis, the cycle will automatically adjust the associated starting point axes. You can enter any additional commands (up to 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 or ShopMill program to be processed has been created and you are in the editor.

2. Press the "Contour milling" 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. Enter the starting point for the contour.
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. 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. **Note:** The additional commands are only effective for path milling!	

10.4.4 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.

Cylinder surface transformation

For contours (e.g. slots) on cylinders, lengths are frequently specified in the form of angles. If the "Cylinder surface transformation" function is activated, you can also define on a cylinder the length of contours (in the circumferential direction of the cylinder surface) using angles. This means instead of X, Y and I, J, you enter Xα, Yα and Iα, Jα.

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

Parameter input

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.

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 thus far, one of the options must be selected.
- Close contour
  From the actual position, you can close the contour with a straight line to the starting point.

Procedure for entering contour elements

1. The part program or the machining schedule is opened. Position the cursor at the desired entry position.
2. Contour input using contour support:

2.1 Press the "Contour milling", "Contour" and "New contour" softkeys.

2.2 In the opened input window, enter a name for the contour, e.g. contour_1.

Press the "Accept" softkey.

2.3 The input screen to enter the contour opens, in which you initially enter a starting point for the contour. This is marked in the lefthand navigation bar using the "+" symbol.

Press the "Accept" softkey.

3. Enter the individual contour elements of the machining direction.

Select a contour element via softkey.

The "Straight (e.g. X)" input window opens.

- OR

The "Straight (e.g. Y)" input window opens.

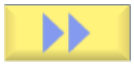
- OR

The "Straight (e.g. XY)" input window opens.

- OR

The "Circle" input window opens.

- OR



The "Pole Input" input window opens.

4. Enter all the data available from the workpiece drawing in the input screen (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 into the process plan (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.

Contour element "Straight line, e.g. X"

Parameter	Description		Unit
X	End point X (abs or inc)		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 "straight line, e.g. Y"

Parameter	Description		Unit
Y	End point Y (abs or inc)		mm
α1	Starting angle to X axis		Degrees
Transition to next element	Type of transition • Radius • Chamfer		
Radius	R	Transition to following element - radius	mm

Parameter	Description		Unit
Chamfer	FS	Transition to following element - chamfer	mm
Additional commands	Additional G code commands		

Contour element "Straight line, e.g. XY"

Parameter	Description		Unit
X	End point X (abs or inc)		mm
Y	End point Y (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 "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. Y	End point Y (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
α1	Starting angle to X axis		Degrees
α2	Angle to the preceding element		Degrees
β1	End angle to Z axis		Degrees
β2	Angle of opening		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
Y	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.

10.4.5 Changing the contour

Function

You can change a previously created contour later.

If you want to create a contour that is similar to an existing contour, you can copy the existing one, rename it and just alter selected contour elements.

Individual contour elements can be

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

Procedure for changing a contour element

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

   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 or ShopMill program to be executed.
2. Position the cursor on the contour element that you want to delete.
3. Press the "Delete element" softkey.

4. Press the "Delete" softkey.

10.4.6 Contour call (CYCLE62) - only for G code program

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.

Procedure

1. The subprogram to be edited has been created and you are in the editor.
2. Press the "Contour milling" softkey.

3. Press the "Contour" and "Contour call" softkeys.
   The "Contour Call" input window opens.

4. Assign parameters to the contour selection.

Parameter	Description	Unit
Contour selection 	• Contour name • Labels • Subprogram • Labels in the subprogram	
Contour name	CON: Contour name	

Parameter	Description	Unit
Labels	• LAB1: Label 1 • LAB2: Label 2	
Subprogram	PRG: Subprogram	
Labels in the subprogram	• PRG: Subprogram • LAB1: Label 1 • LAB2: Label 2	

Note

EXTCALL / EES

When calling a part program via EXTCALL without EES, the contour can only be called via “Contour name” and/or “Labels”. This is monitored in the cycle, which means that contour calls via "subprogram" or "labels in subprogram" are only possible if EES is active.

10.4.7 Path milling (CYCLE72)

Function

You can mill along any programmed contour with the "Path milling" cycle. The function operates with cutter radius compensation. You can machine in any direction, i.e. in the direction of the programmed contour or in the opposite direction.

It is not imperative that the contour is closed. The following machining operations are possible:

- Inside or outside machining (on left or right of the contour).
- Machining along the center-point path

For machining in the opposite direction, contours must not consist of more than 170 contour elements (incl. chamfers/radii). Special aspects (except for feed values) of free G code input are ignored during path milling in the opposite direction to the contour.

Note

Activating G40

We recommend that you activate G40 before the cycle call.

Programming of arbitrary contours

The machining of arbitrary open or closed contours is generally programmed as follows:

1. Enter contour
   You build up the contour gradually from a series of different contour elements.
   Define the contour in a subprogram or in the machining program, e.g. after the end of program (M02 or M30).
2. Contour call (CYCLE62)
   You select the contour to be machined.
3. Path milling (roughing)
   The contour is machined taking into account various approach and retract strategies.
4. Path milling (finishing)
   If you programmed a finishing allowance for roughing, the contour is machined again.
5. Path milling (chamfering)
   If you have planned edge breaking, chamfer the workpiece with a special tool.

Path milling on right or left of the contour

A programmed contour can be machined with the cutter radius compensation to the right or left. You can also select various modes and strategies of approach and retraction from the contour.

Note

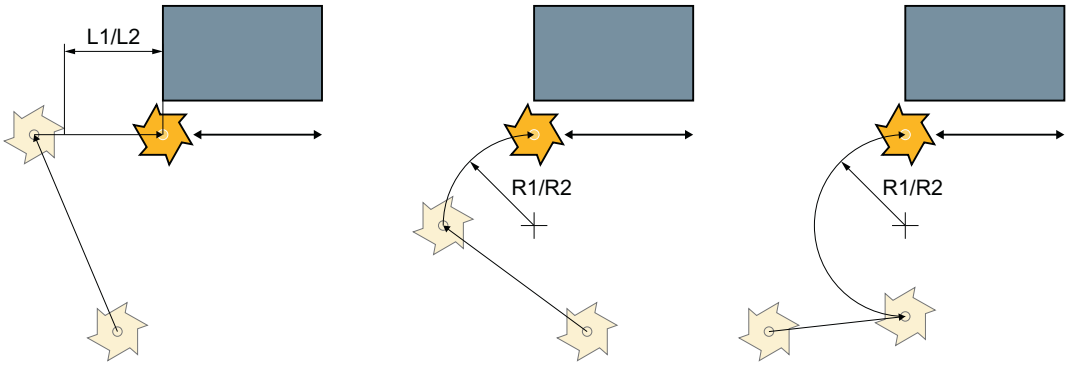
During chamfering, the end mill behaves like a centering tool with a 90° tip angle.

Approach/retraction mode

The tool can approach or retract from the contour along a quadrant, semi-circle or straight line.

- With a quadrant or semi-circle, you must specify the radius of the cutter center point path.
- With a straight line, you must specify the distance between the cutter outer edge and the contour starting or end point.

You can also program a mixture of modes, e.g. approach along quadrant, retract along semi-circle.



L1 Approach length
L2 Retraction length
R1 Approach radius
R2 Retraction radius

Figure 10-9 Approach and retraction along straight line, quadrant and semi-circle

Approach/retraction strategy

You can choose between planar approach/retraction and spatial approach/retraction:

- Planar approach:
  Approach is first at depth and then in the machining plane.
- Spatial approach:
  Approach is at depth and in machining plane simultaneously.
- Retraction is performed in reverse order.
  Mixed programming is possible, for example, approach in the machining plane, retract spatially.

Path milling along center point path.

A programmed contour can also be machined along the center point path if the radius correction was switched out. In this case, approaching and retraction is only possible along a straight line or vertical. Vertical approach/retraction can be used for closed contours, for example.

Procedure

1. The part program or ShopMill program to be processed has been created and you are in the editor.
2. Press the "Contour milling" and "Path milling" softkeys.
   The "Path Milling" input window opens.

Parameters, G code program				Parameters, ShopMill program		
PL	Machining plane			T	Tool name	
RP	Retraction plane	mm		D	Cutting edge number	
SC	Safety clearance	mm		F	Feedrate	mm/min mm/tooth
F	Feedrate	*		S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
Machining	• ∇ (roughing) • ∇∇∇ (finishing) • Chamfering	
Machining direction	Machining in the programmed contour direction • Forward: Machining is performed in the programmed contour direction • Backward: Machining is performed in the opposite direction to the programmed contour	
Radius compensation	• Left (machining to the left of the contour) • Right (machining to the right of the contour) • off A programmed contour can also be machined on the center-point path. In this case, approaching and retraction is only possible along a straight line or vertical. Vertical approach/ retraction can be used for closed contours, for example.	
Z0	Reference point Z (abs or inc)	
Z1	Final depth (abs) or final depth referred to Z0 (inc) - (only for ∇ and ∇∇∇)	mm
DZ	Maximum depth infeed - (only for ∇ and ∇∇∇)	mm
UZ	Depth finishing allowance - (only for ∇)	mm
FS	Chamfer width for chamfering (inc) - (for chamfering only)	mm
ZFS	Insertion depth of tool tip (abs or inc) - (for chamfering only)	mm
UXY	Finishing allowance, plane - (only for ∇ and G code, not for radius compensation off)	mm

Parameter	Description	Unit
Approach	Planar approach mode: • Straight line: Slope in space • Quadrant: Part of a spiral (only with path milling left and right of the contour) • Semi-circle: Part of a spiral (only with path milling left and right of the contour) • Perpendicular: Perpendicular to the path (only with path milling on the center-point path)	
Approach strategy	• Axis by axis • Spatial (only for "quadrant, semi-circle or straight line" approach) 	
R1	Approach radius - (only for "quadrant or semi-circle" approach)	mm
L1	Approach distance - (only for "straight line" approach)	mm
Retraction	Planar retraction mode: • Straight line • Quadrant: Part of a spiral (only with path milling left and right of the contour) • Semi-circle: Part of a spiral (only with path milling left and right of the contour)	
Retraction strategy	• Axis by axis • Spatial (not with perpendicular approach mode) 	
R2	Retraction radius - (only for "quadrant or semi-circle" retraction)	mm
L2	Retraction distance - (only for "straight line" retraction)	mm
Lift mode	If more than one depth infeed is necessary, specify the retraction height to which the tool retracts between the individual infeeds (at the transition from the end of the contour to the start). Lift mode before new infeed • Z0 + safety clearance • By the safety clearance • to RP • No retraction	
FZ (only for ShopMill)	Depth infeed rate – (only for axis-by-axis approach strategy)	mm/min mm/tooth
FZ - (only for G code)	Depth infeed rate – (only for axis-by-axis approach strategy)	*

Parameter	Description	Unit
FS	Chamfer width for chamfering - (only for chamfering machining)	mm
ZFS	Insertion depth of tool tip (abs or inc) - (for machining only)	mm

* Unit of feedrate as programmed before the cycle call

Note

Cylinder surface transformation with slot side compensation

For cylinder surface transformation with selected slot side compensation and approach/retraction in quadrant or semi-circle, the approach/retraction radius must be greater than "Offset to programmed path – tool radius".

10.4.8 Contour pocket/contour spigot (CYCLE63/64)

Contours for pockets or islands

Contours for pockets or islands must be closed, i.e. the starting point and end point of the contour are identical. You can also mill pockets that contain one or more islands. The islands can also be located partially outside the pocket or overlap each other. The first contour you specify is interpreted as the pocket contour and all the others as islands.

Automatic calculation / manual input of the starting point

Using "Automatic starting point" you have the option of calculating the optimum plunge point.

By selecting "Manual starting point", you define the plunge point in the parameter screen.

If the islands and the miller diameter, which must be plunged at various locations, are obtained from the pocket contour, then the manual entry only defines the first plunge point; the remaining plunge points are automatically calculated.

Contours for spigots

Contours for spigots must be closed, i.e. the starting point and end point of the contour are identical. You can define multiple spigots that can also overlap. The first contour specified is interpreted as a blank contour and all others as spigots.

Machining

You program the machining of contour pockets with islands/blank contour with spigots, e.g. as follows:

1. Enter the pocket contour/blank contour
2. Enter the island/spigot contour
3. Call the contour for pocket contour/blank contour or island/spigot contour (only for G code program)
4. Center (this is only possible for pocket contour)

5. Predrill (this is only possible for pocket contour)
6. Solid machine/machine pocket / spigot - roughing
7. Solid machine/machine remaining material - roughing
8. Finishing (base/edge)
9. Chamfering

Note

The following error messages can occur when chamfering inside contours:

Safety clearance in the program header too large

This error message appears when chamfering would, in principle, be possible with the parameters entered for FS and ZFS, but the safety clearance then could not be maintained.

Immersion depth too large

This error message appears when chamfering would be possible through the reduction of the immersion depth ZFS.

Tool diameter too large

This error message appears when the tool would already damage the edges during insertion. In this case, the chamfer FS must be reduced.

Software option

For removing residual stock, you require the option "residual material detection and machining".

Name convention

For multi-channel systems, cycles attach a "_C" and a two-digit, channel-specific number to the names of the programs to be generated, e.g. for channel 1 "_C01". This is the reason why the name of the main program must not end with "_C" and a two-digit number. This is monitored by the cycles.

For single-channel systems, cycles do not extend the name of the programs to be generated.

Note

G code programs

For G code programs, the programs to be generated, which do not include any path data, are saved in the directory in which the main program is located. In this case, it must be ensured that programs, which already exist in the directory and which have the same name as the programs to be generated, are overwritten.

10.4.9 Predrilling contour pocket (CYCLE64)

Function

In addition to predrilling, the cycle can be used for centering. The centering or predrilling program generated by the cycle is called for this purpose.

The number and positions of the required predrilled holes depends on the specific conditions, e.g. type of contour, tool, plane infeed, finishing allowances.

If you mill several pockets and want to avoid unnecessary tool changes, predrill all the pockets first and then remove the stock. In this case, for centering/predrilling, you also have to enter the parameters that appear when you press the "All parameters" softkey. These parameters must correspond to the parameters from the previous stock removal step.

Programming

When programming, proceed as follows:

1. Contour pocket 1
2. Centering
3. Contour pocket 2
4. Centering
5. Contour pocket 1
6. Predrilling
7. Contour pocket 2
8. Predrilling
9. Contour pocket 1
10. Stock removal
11. Contour pocket 2
12. Stock removal

If you are doing all the machining for the pocket at once, i.e. centering, rough-drilling and removing stock directly in sequence, and do not set the additional parameters for centering/ rough-drilling, the cycle will take these parameter values from the stock removal (roughing) machining step. When programming in G code, these values must be specifically re-entered.

Note

Execution from external media

If you want to execute programs from an external drive (e.g. local drive or network drive), you require the Execution from external storage (EES) function.

Procedure when centering

1. The part program or ShopMill program to be processed has been created and you are in the editor.
2. Press the "Cont. mill.", "Predrilling", and "Centering" softkeys.
   The "Centering" input window opens.

G code program parameters				ShopMill program parameters		
PRG	• Name of the program to be generated • Automatic Automatic generation of program names		T	Tool name	
PL	Machining plane		D	Cutting edge number	
Milling direction	• Synchronous operation • Up-cut operation		F	Feedrate	mm/min mm/tooth
RP	Retraction plane	mm	S / V	Spindle speed or constant cutting rate	rpm m/min
SC	Safety clearance	mm			
F	Feedrate	mm/min			

Parameter	Description	Unit
TR	Reference tool Tool, which is used in the "Stock removal" machining step. This is used to determine the plunge position.	
Z0	Reference point Z	mm
Z1	Depth with reference to Z0 (inc)	mm
DXY	• Maximum plane infeed • Maximum plane infeed as a percentage of the milling cutter diameter	mm %
UXY	Finishing allowance, plane	mm
Lift mode	Lift mode before new infeed If the machining operation requires several points of insertion, the retraction height to which the tool is retracted, is selected as follows: • To retraction plane • Z0 + safety clearance If there are no elements larger than Z0 in the pocket area, "Z0 + safety clearance" can be selected as the lift mode.	 mm mm

Predrilling procedure

1. The part program or ShopMill program to be processed has been created and you are in the editor.
2. Press the "Cont. mill.", "Predrilling", and "Predrilling" softkeys.
   The "Predrilling" input window opens.

G code program parameters				ShopMill program parameters		
PRG U	• Name of the program to be generated • Automatic Automatic generation of program names			T	Tool name	
PL U	Machining plane			D	Cutting edge number	
Milling direction U	• Synchronous operation • Up-cut operation			F U	Feedrate	mm/min mm/tooth
RP	Retraction plane	mm		S / V U	Spindle speed or constant cutting rate	rpm m/min
SC	Safety clearance	mm				
F	Feedrate	mm/min				

Parameter	Description	Unit
TR	Reference tool Tool, which is used in the "Stock removal" machining step. This is used to determine the plunge position.	
Z0	Reference point in the tool axis Z	mm
Z1 U	Pocket depth (abs) or depth referred to Z0 (inc)	mm
DXY U	• Maximum plane infeed • Maximum plane infeed as a percentage of the milling cutter diameter	mm %
UXY	Finishing allowance, plane	mm
UZ	Finishing allowance, depth	mm
Lift mode U	Lift mode before new infeed If the machining operation requires several points of insertion, the retraction height to which the tool is retracted, is selected as follows: • To retraction plane • Z0 + safety clearance If there are no elements larger than Z0 in the pocket area, "Z0 + safety clearance" can be selected as the lift mode.	 mm mm

10.4.10 Milling contour pocket (CYCLE63)

Function

Before you can machine a pocket with islands, you must enter the contour of the pocket and islands. The first contour you specify is interpreted as the pocket contour and all the others as islands.

From the programmed contours and the input screen form for stock removal, the cycle generates a program that removes the pockets with islands from inside to outside in parallel to the contour.

The islands can also be located partially outside the pocket or overlap each other.

Note

Execution from external media

If you want to execute programs from an external drive (e.g. local drive or network drive), you require the Execution from external storage (EES) function.

Input simple

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

Machine manufacturer

Various defined values can be pre-assigned using setting data.

Please refer to the machine manufacturer's specifications.

If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

Machining type

For solid machining, you can select the machining type (roughing or finishing). If you want to rough and then finish, you have to call the machining cycle twice (block 1 = roughing, block 2 = finishing). The programmed parameters are retained when the cycle is called for the second time.

During insertion with oscillation, the message "Ramp path too short" will appear if the tool is less than the milling cutter diameter away from the insertion point along the ramp, or the machining depth is not reached.

- Reduce the insertion angle if the tool remains too close to the insertion point.
- Increase the insertion angle if the tool does not reach the machining depth.
- If necessary, use a tool with a smaller radius of select a different insertion mode.

Procedure

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

2. Press the "Contour milling" and "Pocket" softkeys.
   The "Mill pocket" input window opens.

Parameters in the "Input complete" mode

G code program parameters				ShopMill program parameters		
Input			• complete			
PRG	• Name of the program to be generated • Automatic Automatic generation of program names			T	Tool name	
PL	Machining plane			D	Cutting edge number	
Milling direction	• Synchronous operation • Up-cut operation			F	Feedrate	mm/min mm/tooth
RP	Retraction plane	mm		S / V	Spindle speed or constant cutting rate	rpm m/min
SC	Safety clearance	mm				
F	Feedrate	mm/min				

Parameter	Description	Unit
Machining	The following machining operations can be selected: • ∇ (roughing) • ∇∇∇ base (base finishing) • ∇∇∇ edge (edge finishing) • Chamfering	
Z0	Reference point in the tool axis Z	mm
Z1	Pocket depth (abs) or depth relative to Z0 (inc) - (only for ∇, ∇∇∇ base or ∇∇∇ edge)	mm
DXY	• Maximum plane infeed • Maximum plane infeed as a percentage of the milling cutter diameter - (only for ∇ or ∇∇∇ base)	mm %
DZ	Maximum depth infeed – (only for ∇ or ∇∇∇ edge)	mm
UXY	Plane finishing allowance – (only for ∇, ∇∇∇ base or ∇∇∇ edge)	mm
UZ	Depth finishing allowance – (only for ∇ or ∇∇∇ base)	mm

Parameter	Description	Unit
Starting point	• Manual Starting point is manually entered • Automatic Starting point is automatically calculated - (only for ∇ or ∇∇∇ base)	
XS	Starting point X - (only for "manual" starting point)	mm
YS	Starting point Y - (only for "manual" starting point)	mm
Insertion	The following insertion modes can be selected – (only for ∇, ∇∇∇ base or ∇∇∇ edge): • **Vertical insertion** The calculated current infeed depth is executed at the calculated position for "automatic" starting point – or at the specified position for "manual" starting point. Note: This setting can be used only if the cutter can cut across center or if the pocket has been predrilled. • **Helical insertion** The cutter center point traverses along the helical path determined by the radius and depth per revolution (helical path). If the depth for one infeed has been reached, a full circle motion is executed to eliminate the inclined insertion path. • **Oscillating insertion** The cutter center point oscillates back and forth along a linear path until it reaches the depth infeed. When the depth has been reached, the path is traversed again without depth infeed in order to eliminate the inclined insertion path.	
FZ (only for ShopMill)	Depth infeed rate – (only for perpendicular insertion and ∇)	mm/min mm/tooth
FZ (only for G code)	Depth infeed rate – (only for perpendicular insertion and ∇)	*
EP	Maximum pitch of helix – (for helical insertion only)	mm/rev
ER	Radius of helix – (for helical insertion only) The radius cannot be any larger than the cutter radius; otherwise, material will remain.	mm
EW	Maximum insertion angle – (for insertion with oscillation only)	Degrees
Lift mode	Lift mode before new infeed - (only for ∇, ∇∇∇ base or ∇∇∇ edge) If the machining operation requires several points of insertion, the retraction height to which the tool is retracted, is selected as follows: • To retraction plane • Z0 + safety clearance If there are no elements larger than Z0 in the pocket area, "Z0 + safety clearance" can be selected as the lift mode.	 mm mm
FS	Chamfer width for chamfering - (only for chamfering machining)	mm
ZFS	Insertion depth of tool tip (abs or inc) - (for machining only)	mm

* Unit of feedrate as programmed before the cycle call

Note

When input manually, the starting point can also be located outside the pocket. This can be useful, for example, when machining a pocket which is open on one side. The machining operation then begins without insertion with a linear movement into the open side of the pocket.

Parameters in the "Input simple" mode

G code program parameters				ShopMill program parameters		
Input	• simple					
PRG • Name of the program to be generated • Automatic Automatic generation of program names				T	Tool name	
Milling direction • Synchronous operation • Up-cut operation				D	Cutting edge number	
RP	Retraction plane	mm		F	Feedrate	mm/min mm/rev
F	Feedrate	*		S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	
Machining	The following machining operations can be selected: • ∇ (roughing) • ∇∇∇ base (base finishing) • ∇∇∇ edge (edge finishing) • Chamfering	
Z0	Reference point in the tool axis Z	mm
Z1	Pocket depth (abs) or depth relative to Z0 (inc) - (only for ∇, ∇∇∇ base or ∇∇∇ edge)	mm
DXY	• Maximum plane infeed • Maximum plane infeed as a percentage of the milling cutter diameter (only for ∇ or ∇∇∇ base)	mm %
DZ	Maximum depth infeed – (only for ∇ and ∇∇∇ edge)	mm
UXY	Plane finishing allowance - (only for ∇, ∇∇∇ base or ∇∇∇ edge)	mm
UZ	Depth finishing allowance – (only for ∇ or ∇∇∇ base)	mm

Parameter	Description	
Insertion	The following insertion modes can be selected – (only for ∇, ∇∇∇ base or ∇∇∇ edge): • **Vertical** The calculated actual infeed depth is machined at the calculated position. Note: This setting can be used only if the cutter can cut across center or if the pocket has been predrilled. • **Helical** The cutter center point traverses along the helical path determined by the radius and depth per revolution (helical path). If the depth for one infeed has been reached, a full circle motion is executed to eliminate the inclined insertion path. • **Oscillation** The cutter center point oscillates back and forth along a linear path until it reaches the depth infeed. When the depth has been reached, the path is traversed again without depth infeed in order to eliminate the inclined insertion path.	
FZ (only for ShopMill)	Depth infeed rate – (only for perpendicular insertion and ∇)	mm/min mm/tooth
FZ (only for G code)	Depth infeed rate – (only for perpendicular insertion and ∇)	*
EP	Maximum pitch of helix – (for helical insertion only)	mm/rev
ER	Radius of helix – (for helical insertion only) The radius cannot be any larger than the cutter radius; otherwise, material will remain.	mm
EW	Maximum insertion angle - (only for insertion with oscillation)	Degrees
FS	Chamfer width for chamfering (inc) - (for chamfering only)	mm
ZFS	Insertion depth of tool tip (abs or inc) - (for chamfering only)	mm

* Unit of feedrate as programmed before the cycle call

Hidden parameters

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.

Parameter	Description	Value	Can be set in SD
PL (only for G code)	Machining plane	Defined in MD 52005	
SC (only for G code)	Safety clearance	1 mm	x
Starting point	Starting point is automatically calculated - (only for ∇ and ∇∇∇ base)	Automatic	
Lift mode	Lift mode before new infeed - (only for ∇, ∇∇∇ base or ∇∇∇ edge)	To RP	

Machine manufacturer

Please observe the information provided by the machine manufacturer.

10.4.11 Residual material contour pocket (CYCLE63)

Function

When you have removed stock from a pocket (with/without islands) and there is residual material, then this is automatically detected. You can use a suitable tool to remove this residual material without having to machine the whole pocket again, i.e. avoiding unnecessary non-productive motion. The finishing allowance should be set identically for all machining steps because it does not count as residual material.

The residual material is calculated on the basis of the milling cutter used for stock removal.

It is also possible to run multiple residual material steps one after the other. In this case, the milling tool should be selected to be smaller by a factor of no more than 3 for each new step.

If you mill several pockets and want to avoid unnecessary tool changes, remove stock from all the pockets first and then remove the residual material. In this case, for removing the residual material, you also have to enter a value for the reference tool TR parameter, which, for the ShopMill program, additionally appears when you press the "All parameters" softkey. When programming, proceed as follows:

1. Contour pocket 1
2. Stock removal
3. Contour pocket 2
4. Stock removal
5. Contour pocket 1
6. Remove residual material
7. Contour pocket 2
8. Remove residual material

Software option

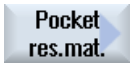
For removing residual stock, you require the option "residual material detection and machining".

Procedure

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

2. Press the "Contour milling" and "Pocket Res. Mat." softkeys.

   The "Pocket Res. Mat." input window opens.



3. For the ShopMill program, press the "All parameters" softkey if you want to enter additional parameters.

G code program parameters				ShopMill program parameters		
PRG ↻	• Name of the program to be generated • Automatic Automatic generation of program names		T	Tool name		
PL ↻	Machining plane		F ↻	Feedrate	mm/min mm/tooth	
Milling direction ↻	• Synchronous operation • Up-cut operation		S / V ↻	Spindle speed or constant cutting rate	rpm m/min	
RP	Retraction plane	mm				
SC	Safety clearance	mm				
F	Feedrate	mm/min				

Parameter	Description	Unit
Machining	The following machining operations can be selected: ∇ (roughing)	
TR	Reference tool Tool, which is used in the "Stock removal" machining step. Is used to determine the residual material.	
D ↻	Cutting edge number	
Z0	Reference point in the tool axis Z	mm
Z1 ↻	Pocket depth (abs) or depth referred to Z0 (inc)	mm
DXY ↻	• Maximum plane infeed • Maximum plane infeed as a percentage of the milling cutter diameter	mm %
DZ	Maximum depth infeed	mm
UXY	Finishing allowance, plane	mm
UZ	Finishing allowance, depth	mm
Lift mode ↻	Lift mode before new infeed If the machining operation requires several points of insertion, the retraction height to which the tool is retracted, is selected as follows: • To retraction plane • Z0 + safety clearance If there are no elements larger than Z0 in the pocket area, "Z0 + safety clearance" can be selected as the lift mode.	 mm mm

10.4.12 Milling contour spigot (CYCLE63)

Function

You can mill any spigot using the "Mill spigot" cycle.

Before you mill the spigot, you must first enter a blank contour and then one or more spigot contours. The blank contour defines the area, outside of which there is no material, i.e. the tool moves with rapid traverse there. Material is then removed between the blank contour and spigot contour.

Note

Execution from external media

If you want to execute programs from an external drive (e.g. local drive or network drive), you require the Execution from external storage (EES) function.

Input simple

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

Machine manufacturer

Various defined values can be pre-assigned using setting data.

Please observe the information provided by the machine manufacturer.

If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

Machining type

You can select the machining mode (roughing, base finishing, edge finishing, chamfer) for milling. If you want to rough and then finish, you have to call the machining cycle twice (Block 1 = roughing, Block 2 = finishing). The programmed parameters are retained when the cycle is called for the second time.

Approach/retraction

1. The tool approaches the starting point in rapid traverse at the height of the retraction plane and goes to the safety clearance. The cycle calculates the starting point.
2. The tool first infeeds to the machining depth and then approaches the spigot contour from the side in a quadrant at machining feedrate.
3. The spigot is machined in parallel with the contours from the outside in. The direction is determined by the machining direction (climbing or conventional).
4. When the first plane of the spigot has been machined, the tool retracts from the contour in a quadrant and then infeeds to the next machining depth.
5. The spigot is again approached in a quadrant and machine in parallel with the contours from outside in.
6. Steps 4 and 5 are repeated until the programmed spigot depth is reached.
7. The tool moves back to the safety clearance in rapid traverse.

Procedure

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

2. Press the "Contour milling" and "Spigot" softkeys.
   The "Mill spigot" input window opens.

3. Select the "Roughing" machining type.

Parameters in the "Input complete" mode

G code program parameters				ShopMill program parameters		
Input	• complete					
PRG ↻	• Name of the program to be generated • Automatic Automatic generation of program names			T	Tool name	
PL ↻	Machining plane			D	Cutting edge number	
Milling direction ↻	• Synchronous operation • Up-cut operation			F ↻	Feedrate	mm/min mm/tooth
RP	Retraction plane	mm		S / V ↻	Spindle speed or constant cutting rate	rpm m/min
SC	Safety clearance	mm				
F	Feedrate	mm/min				

Parameter	Description	Unit
Machining ↻	The following machining operations can be selected: • ∇ (roughing) • ∇∇∇ base (base finishing) • ∇∇∇ edge (edge finishing) • Chamfering	
Z0	Reference point in the tool axis Z	mm
Z1 ↻	Pocket depth (abs) or depth relative to Z0 (inc) - (only for ∇, ∇∇∇ base or ∇∇∇ edge)	mm
DXY ↻	• Maximum plane infeed • Maximum plane infeed as a percentage of the milling cutter diameter - (only for ∇ and ∇∇∇ base)	mm %
DZ	Maximum depth infeed – (only for ∇ or ∇∇∇ edge)	mm
UXY	Plane finishing allowance – (only for ∇, ∇∇∇ base or ∇∇∇ edge)	mm

Parameter	Description	Unit
UZ	Depth finishing allowance – (only for ∇ or ∇∇∇ base)	mm
Lift mode ↻	Lift mode before new infeed If the machining operation requires several points of insertion, the retraction height to which the tool is retracted, is selected as follows: • To retraction plane • Z0 + safety clearance If there are no elements larger than Z0 (X0) in the pocket area, then Z0 (X0) + safety clearance can be programmed as the lift mode.	 mm mm mm
FS	Chamfer width for chamfering - (only for chamfering machining)	mm
ZFS ↻	Insertion depth of tool tip (abs or inc) - (for machining only)	mm

Parameters in the "Input simple" mode

G code program parameters				ShopMill program parameters		
Input ↻	• simple					
PRG ↻	• Name of the program to be generated • Automatic Automatic generation of program names		T	Tool name		
Milling direction ↻	• Synchronous operation • Up-cut operation		D	Cutting edge number		
RP	Retraction plane	mm	F ↻	Feedrate	mm/min mm/rev	
F	Feedrate	*	S / V ↻	Spindle speed or constant cutting rate	rpm m/min	

Parameter	Description	
Machining ↻	The following machining operations can be selected: • ∇ (roughing) • ∇∇∇ base (base finishing) • ∇∇∇ edge (edge finishing) • Chamfering	
Z0	Reference point in the tool axis Z	mm
Z1 ↻	Spigot depth (abs) or depth relative to Z0 (inc) - (only for ∇, ∇∇∇ base and ∇∇∇ edge)	mm
DXY ↻	• Maximum plane infeed • Maximum plane infeed as a percentage of the milling cutter diameter- (only for ∇ and ∇∇∇ base)	mm %
DZ	Maximum depth infeed – (only for ∇ and ∇∇∇ edge)	mm

Parameter	Description	
UXY	Plane finishing allowance - (only for ∇, ∇∇∇ base or ∇∇∇ edge)	mm
UZ	Depth finishing allowance – (only for ∇ or ∇∇∇ base)	mm
FS	Chamfer width for chamfering (inc) - (for chamfering only)	mm
ZFS	Insertion depth of tool tip (abs or inc) - (for chamfering only)	mm

* Unit of feedrate as programmed before the cycle call

Hidden parameters

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.

Parameter	Description	Value	Can be set in SD
PL (only for G code)	Machining plane	Defined in MD 52005	
SC (only for G code)	Safety clearance	1 mm	x
Lift mode	Lift mode before new infeed - (only for ∇, ∇∇∇ base or ∇∇∇ edge)	To RP	

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

10.4.13 Residual material contour spigot (CYCLE63)

Function

When you have milled a contour spigot and residual material remains, then this is automatically detected. You can use a suitable tool to remove this residual material without having to machine the whole spigot again, i.e. avoiding unnecessary non-productive motion. The finishing allowance should be set identically for all machining steps because it does not count as residual material.

The residual material is calculated on the basis of the milling cutter used for clearing.

It is also possible to run multiple residual material steps one after the other. In this case, the milling tool should be selected to be smaller by a factor of no more than 3 for each new step.

If you mill several spigots and want to avoid unnecessary tool changes, clear all the spigots first and then remove the residual material. In this case, for removing the residual material, you also have to enter a value for the reference tool TR parameter, which, for the ShopMill program, additionally appears when you press the "All parameters" softkey. When programming, proceed as follows:

1. Contour blank 1
2. Contour spigot 1
3. Clear spigot 1

4. Contour blank 2
5. Contour spigot 2
6. Clear spigot 2
7. Contour blank 1
8. Contour spigot 1
9. Clear residual material spigot 1
10. Contour blank 2
11. Contour spigot 2
12. Clear residual material spigot 2

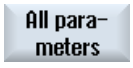
Software option

For removing residual stock, you require the option "residual material detection and machining".

Procedure

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

2. Press the "Contour milling" and "Spigot Res. Mat." softkeys.
   The "Spigot Res. Mat." input window opens.



3. For the ShopMill program, press the "All parameters" softkey if you want to enter additional parameters.

G code program parameters				ShopMill program parameters		
PRG	• Name of the program to be generated • Automatic Automatic generation of program names			T	Tool name	
PL	Machining plane			F	Feedrate	mm/min mm/tooth
Milling direction	• Synchronous operation • Up-cut operation			S / V	Spindle speed or constant cutting rate	rpm m/min
RP	Retraction plane	mm				
SC	Safety clearance	mm				
F	Feedrate	mm/min				

Parameter	Description	Unit
TR	Reference tool Tool, which is used in the "Stock removal" machining step. This is used to determine the residual corners.	
D	Cutting edge number	
Z0	Reference point in the tool axis Z	mm
Z1	Pocket depth (abs) or depth referred to Z0 (inc)	mm
DXY	• Maximum plane infeed • Maximum plane infeed as a percentage of the milling cutter diameter	mm %
DZ	Maximum depth infeed	mm
Lift mode	Lift mode before new infeed If the machining operation requires several points of insertion, the retraction height to which the tool is retracted, is selected as follows: • To retraction plane • Z0 + safety clearance If there are no elements larger than Z0 in the pocket area, "Z0 + safety clearance" can be selected as the lift mode.	 mm mm

10.5 Turning - milling/turning machine

10.5.1 General

In all turning cycles apart from contour turning (CYCLE95), it is possible to reduce the feedrate as a percentage when finishing in the combined roughing and finishing mode.

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

10.5.2 Stock removal (CYCLE951)

Function

You can use the "Stock removal" cycle for longitudinal or transverse stock removal of corners at outer or inner contours.

Note

Removing stock from corners

For this cycle, the safety clearance is additionally limited using setting data. The lower value is taken for machining.

Please refer to the machine manufacturer's specifications.

Machining type

- Roughing
  For roughing applications, paraxial cuts are machined to the finishing allowance that has been programmed. If no finishing allowance has been programmed, the workpiece is roughed down to the final contour.
  During roughing, the cycle reduces the programmed infeed depth D if necessary so that it is possible for cuts of an equal size to be made. For example, if the overall infeed depth is 10 and you have specified an infeed depth of 3, this would result in cuts of 3, 3, 3 and 1. The cycle now reduces the infeed depth to 2.5 so that four cuts of equal size are created.
  The angle between the contour and the tool cutting edge determines whether the tool rounds the contour at the end of each cut by the infeed depth D in order to remove residual corners, or whether it is raised immediately. The angle beyond which rounding is performed is stored in a machine data element.

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

If the tool does not round the corner at the end of the cut, it is raised by the safety clearance or a value specified in the machine data with rapid traverse. The cycle always observes the lower value; otherwise, stock removal at inner contours, for example, could cause the contour to be damaged.

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

- Finishing
  Finishing is performed in the same direction as roughing. The cycle automatically selects and deselects tool radius compensation during finishing.

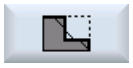
Procedure

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

2. Press the "Turning" softkey.

3. Press the "Stock removal" softkey.
   The "Stock Removal" input window opens.
4. Select one of the three stock removal cycles via the softkeys:

Simple straight stock removal cycle.
The "Stock removal 1" input window opens.
- OR -

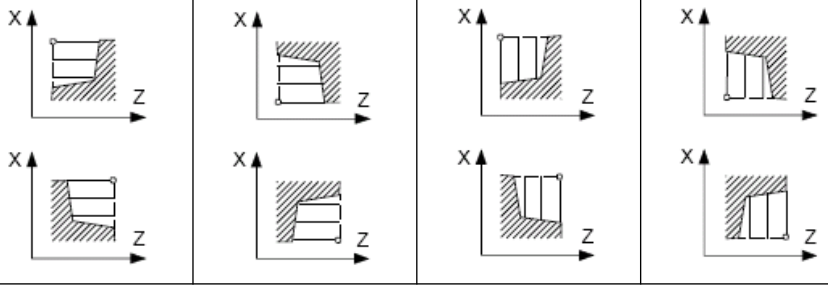


Straight stock removal cycle with radii or chamfers.
The "Stock removal 2" input window opens.
- OR -

Stock removal cycle with oblique lines, radii, or chamfers.
The "Stock Removal 3" input window opens.

G code program parameters				ShopMill program parameters		
PL	Machining plane			T		
SC	Safety clearance	mm		D	Cutting edge number	
F	Feedrate	*		F	Feedrate	mm/min mm/rev
				S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description		Unit
TC	Name of the swivel data set Note: The selection box only appears if more than one swivel data set has been set up.		
Retraction (for ShopMill program)	No	The axis is not retracted before swiveling	
	Z	Retraction in the direction of machine axis Z	
	Z XY	Retract towards the machine axis Z and then in the direction X, Y	
	max	Maximum retraction (up to the software end position) in the tool direction	
	ink	Incremental retraction (specified retraction distance, up to the software end position) in the tool direction	
	When retracting in the tool direction, in the swiveled machine state, several axes can move (travel).		
ZR (for ShopMill program)	Retraction path - only for incremental retraction in the tool direction		
Align tool through beta and gamma angles			
β (for ShopMill program)	Align tool with swivel axes • Input value The required angle can be freely entered • β = 0° • β = 90°		Degrees
γ (for ShopMill program)	• 0° • 180° • The required angle can be freely entered		Degrees
Directly position rotary axes			
B1 (for ShopMill program)	Directly align tool with swivel axes: The required angle can be freely entered		Degrees
C1 (for ShopMill program)	The required angle can be freely entered		Degrees
αC (for ShopMill program)	Rotational position for a pole position		Degrees
Hirth joint (for ShopMill program)	• Round off to the next Hirth joint for a minimum beta difference • Round beta up • Round beta down Note: For machines with a Hirth joint		

Parameter	Description	Unit
Tool	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.	
Preferred direction (for ShopMill program)	Preferred direction of the swivel axis for several possible alignments of the machine + –	
Machining	• ∇ (roughing) • ∇∇∇ (finishing)	
Position	Stock removal position: 	
Machining direction	Stock removal direction (longitudinal or transverse) in the coordinate system	
	Parallel to the Z axis (longitudinal) / Parallel to the X axis (transverse)	
	outside / inside / outside / inside 	
X0	Reference point in X ∅ (abs, always diameter)	mm
Z0	Reference point in Z (abs)	mm
X1	End point X (abs) or end point X in relation to X0 (inc)	mm
Z1	End point Z ∅ (abs) or end point Z in relation to Z0 (inc)	mm
D	Maximum depth infeed – (not for finishing)	mm
UX	Finishing allowance in X – (not for finishing)	mm
UZ	Finishing allowance in Z – (not for finishing)	mm
FS1...FS3 or R1...R3	Chamfer width (FS1...FS3) or rounding radius (R1...R3) - (not for stock removal 1)	mm

Parameter	Description	Unit
	Parameter selection of intermediate point The intermediate point can be determined through position specification or angle. The following combinations are possible - (not for stock removal 1 and 2) • XM ZM • XM α1 • XM α2 • α1 ZM • α2 ZM • α1 α2	
XM	Intermediate point X Ø (abs) or intermediate point X in relation to X0 (inc)	mm
ZM	Intermediate point Z (abs or inc)	mm
α1	Angle of the 1st edge	Degrees
α2	Angle of the 2nd edge	Degrees

* Unit of feedrate as programmed before the cycle call

10.5.3 Groove (CYCLE930)

Function

You can use the "Groove" cycle to machine symmetrical and asymmetrical grooves on any straight contour elements.

You have the option of machining outer or inner grooves, longitudinally or transversely (face). Use the "Groove width" and "Groove depth" parameters to determine the shape of the groove. If a groove is wider than the active tool, it is machined in several cuts. The tool is moved by a maximum of 80% of the tool width for each groove.

You can specify a finishing allowance for the groove base and the flanks; roughing is then performed down to this point.

The dwell time between recessing and retraction is stored in a setting data element.

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

Approach/retraction during roughing

Infeed depth D > 0

1. The tool first moves to the starting point calculated internally in the cycle at rapid traverse.
2. The tool cuts a groove in the center of infeed depth D.
3. The tool moves back by D + safety clearance with rapid traverse.
4. The tool cuts a groove next to the first groove with infeed depth 2 · D.

5. The tool moves back by D + safety clearance with rapid traverse.
6. The tool cuts alternating in the first and second groove with the infeed depth 2 · D, until the final depth T1 is reached.
   Between the individual grooves, the tool moves back by D + safety clearance with rapid traverse. After the last groove, the tool is retracted at rapid traverse to the safety distance.
7. All subsequent groove cuts are made alternating and directly down to the final depth T1.
   Between the individual grooves, the tool moves back to the safety distance at rapid traverse.

Approach/retraction during finishing

1. The tool first moves to the starting point calculated internally in the cycle at rapid traverse.
2. The tool moves at the machining feedrate down one flank and then along the bottom to the center.
3. The tool moves back to the safety distance at rapid traverse.
4. The tool moves at the machining feedrate along the other flank and then along the bottom to the center.
5. The tool moves back to the safety distance at rapid traverse.

Procedure

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

2. Press the "Turning" softkey.

3. Press the "Groove" softkey.
   The "Groove" input window opens.
4. Select one of the three groove cycles with the softkey:

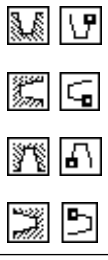
Simple groove cycle
The "Groove 1" input window opens.
- OR -

Groove cycle with inclines, radii, or chamfers.
The "Groove 2" input window opens.
- OR -

Groove cycle on an incline with inclines, radii or chamfers.
The "Groove 3" input window opens.

G code program parameters				ShopMill program parameters		
PL	Machining plane			T		
SC	Safety clearance	mm		D	Cutting edge number	
F	Feedrate	*		F	Feedrate	mm/min mm/rev
				S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
TC	Name of the swivel data set Note: The selection box only appears if more than one swivel data set has been set up.	
Retraction (for ShopMill program)	• No The axis is not retracted before swiveling • Z Z Retraction in the direction of machine axis Z • Z,X,Y Z XY Move machining axes to retraction position before swiveling • Tool direction, max. max Maximum retraction (up to the software end position) in the tool direction • Tool direction, inc. ink Incremental retraction (specified retraction distance, up to the software end position) in the tool direction When retracting in the tool direction, in the swiveled machine state, several axes can move (travel).	
ZR (for ShopMill program)	Retraction path - only for incremental retraction in the tool direction	mm
Align tool through beta and gamma angles		
β (For ShopMill program)	Align tool • Input value The required angle can be freely entered • β = 90° ↓ β = 0° ←	Degrees

Parameter	Description	Unit
γ (for ShopMill program)	• 0° • 180° • The required angle can be freely entered	
Directly position rotary axes		
B1 (for ShopMill program)	Directly align the tool with the swiveling axes: The required angle can be freely entered	Degrees
C1 (for ShopMill program)	Directly align tool with swivel axes: The required angle can be freely entered	Degrees
αC (for ShopMill program)	Rotational position for a pole position	
Hirth joint (for ShopMill program)	• Round to the next Hirth joint • Round up to Hirth joint • Round down to Hirth joint 	
Tool	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.	
Preferred direction (for ShopMill program)	Preferred direction of the swivel axis for several possible alignments of the machine 	
Machining	• ∇ (roughing) • ∇∇∇ (finishing) • ∇ + ∇∇∇ (roughing and finishing)	
Position	Groove position/reference point: 	
X0	Reference point in X ∅	mm
Z0	Reference point in Z	mm

Parameter	Description	Unit
B1	Groove width	mm
T1	Groove depth ∅ (abs) or groove depth referred to X0 or Z0 (inc)	mm
α1, α2	Edge angle 1 or edge angle 2 - (only for grooves 2 and 3) Asymmetric grooves can be described by separate angles. The angles can be between 0 and < 90°.	Degrees
α0	Angle of inclinations – (only for groove 3)	Degrees
FS1...FS4 or R1...R4	Chamfer width (FS1...FS4) or rounding radius (R1...R4) - (only for grooves 2 and 3)	mm
D	• Maximum depth infeed for insertion – (only for ∇ and ∇ + ∇∇∇) • For zero: Insertion in a cut – (only for ∇ and ∇ + ∇∇∇) D = 0: 1. cut is made directly to final depth T1 D > 0: 1st and 2nd cuts are made alternately to infeed depth D, in order to achieve a better chip flow and prevent the tool from breaking, see approaching/retraction when roughing. Alternate cutting is not possible if the tool can only reach the groove base at one position.	mm
UX or U	Finishing allowance in X or finishing allowance in X and Z – (only for ∇ and ∇ + ∇∇∇)	mm
UZ	Finishing allowance in Z – (for UX, only for ∇ and ∇ + ∇∇∇)	mm
N	Number of grooves (N = 1....65535)	
DP	Distance between grooves (inc) DP is not displayed when N = 1	mm

* Unit of feedrate as programmed before the cycle call

10.5.4 Undercut form E and F (CYCLE940)

Function

You can use the "Undercut form E" or "Undercut form F" cycle to turn form E or F undercuts in accordance with DIN 509.

Approach/retraction

1. The tool first moves to the starting point calculated internally in the cycle at rapid traverse.
2. The undercut is made in one cut at the machining feedrate, starting from the flank through to the cross-feed VX.
3. The tool moves back to the starting point at rapid traverse.

Procedure

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

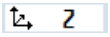
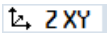
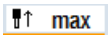
2. Press the "Turning" softkey.

3. Press the "Undercut" softkey.
   The "Undercut" input window opens.
4. Select one of the following undercut cycles via the softkeys:

Press the "Undercut form E" softkey.
The "Undercut form E (DIN 509)" input window opens.
- OR -

Press the "Undercut form F" softkey.
The "Undercut form F (DIN 509)" input window opens.

G code program (undercut, form E) parameters						
PL	Machining plane			T		
SC	Safety clearance	mm		D	Cutting edge number	
F	Feedrate	*		F	Feedrate	mm/min mm/rev
				S / V	Spindle speed or Constant cutting rate	rpm m/min

Parameter	Description	Unit
TC 	Name of the swivel data set Note: The selection box only appears if more than one swivel data set has been set up.	
Retraction (for ShopMill program) 	• No The axis is not retracted before swiveling • Z  Retraction in the direction of machine axis Z • Z,X,Y  Move machining axes to retraction position before swiveling • Tool direction, max.  Maximum retraction (up to the software end position) in the tool direction • Tool direction, inc. Incremental retraction (specified retraction distance, up to the software end position) in the tool direction When retracting in the tool direction, in the swiveled machine state, several axes can move (travel).	
ZR (for ShopMill program)	Retraction path - only for incremental retraction in	mm
Align tool through beta and gamma angles		
β (for ShopMill program) 	Align tool with swivel axes • Input value The required angle can be freely entered • β = 0° • β = 90° 	degrees
γ (for ShopMill program) 	• 0° • 180° • The required angle can be freely entered	degrees
Directly position rotary axes		
B1 (for ShopMill program)	Directly align the tool with the swiveling axes: The required angle can be freely entered	degrees
C1 (for ShopMill program)	The required angle can be freely entered	degrees

Parameter	Description	Unit
αC (for ShopMill program)	Rotational position for a pole position	degrees
Hirth joint	• Round to the next Hirth joint • Round up to Hirth joint • Round down to Hirth joint Note: For machines with a Hirth joint	
Tool	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.	
Preferred direction (for ShopMill program)	Preferred direction of the swivel axis for several possible alignments of the machine 	
Position	Form E machining position: 	
	Undercut size according to DIN table: E.g.: E1.0 x 0.4 (undercut form E)	
X0	Reference point X ∅	mm
Z0	Reference point Z	mm
X1	Allowance in X ∅ (abs) or allowance in X (inc)	mm
UX	Cross feed ∅ (abs) or cross feed (inc)	mm

* Unit of feedrate as programmed before the cycle call

G code program (undercut, form F) parameters						
PL	Machining plane			T	Tool name	
SC	Safety clearance	mm		D	Cutting edge number	
F	Feedrate	*		F	Feedrate	mm/min mm/rev
				S / V	Spindle speed or Constant cutting rate	rpm m/min

Parameter	Description	Unit
TC	Name of the swivel data set Note: The selection box only appears if more than one swivel data set has been set up.	
Retraction (for ShopMill program)	• No The axis is not retracted before swiveling • Z Z Retraction in the direction of machine axis Z • Z,X,Y Z XY Move machining axes to retraction position before swiveling • Tool direction, max. max Maximum retraction (up to the software end position) in the tool direction • Tool direction, inc. ink Incremental retraction (specified retraction distance, up to the software end position) in the tool direction When retracting in the tool direction, in the swiveled machine state, several axes can move (travel).	
ZR (for ShopMill program)	Retraction path - only for incremental retraction in	mm
Align tool through beta and gamma angles		
β (for ShopMill program)	Align tool with swivel axes • Input value The required angle can be freely entered • β = 0° ↓ • β = 90° ←	Degrees

Parameter	Description	Unit
γ (for ShopMill program)	• 0° • 180° • The required angle can be freely entered	Degrees
Directly position rotary axes		
B1 (for ShopMill program)	Directly align the tool with the swiveling axes: The required angle can be freely entered	Degrees
C1 (for ShopMill program)	The required angle can be freely entered	Degrees
αC (for ShopMill program)	Rotational position for a pole position	Degrees
Hirth joint	• Round to the next Hirth joint • Round up to Hirth joint • Round down to Hirth joint Note: For machines with a Hirth joint	
Tool	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.	
Preferred direction (for ShopMill program)	Preferred direction of the swivel axis for several possible alignments of the machine + –	
Position	Form F machining position:	

Parameter	Description	Unit
	Undercut size according to DIN table: e.g.: F0.6 x 0.3 (undercut form F)	
X0	Reference point X ∅	mm
Z0	Reference point Z	mm
X1	Allowance in X ∅ (abs) or allowance in X (inc)	mm
Z1	Allowance in Z (abs) or allowance in Z (inc) – (for undercut form F only)	mm
VX	Cross feed ∅ (abs) or cross feed (inc)	mm

* Unit of feedrate as programmed before the cycle call

10.5.5 Thread undercut (CYCLE940)

Function

The "Thread undercut DIN" or "Thread undercut" cycle is used to program thread undercuts to DIN 76 for workpieces with a metric ISO thread, or freely definable thread undercuts.

Approach/retraction

1. The tool first moves to the starting point calculated internally in the cycle at rapid traverse.
2. The first cut is made at the machining feedrate, starting from the flank and traveling along the shape of the thread undercut as far as the safety distance.
3. The tool moves to the next starting position at rapid traverse.
4. Steps 2 and 3 are repeated until the thread undercut is finished.
5. The tool moves back to the starting point at rapid traverse.

During finishing, the tool travels as far as cross-feed VX.

Procedure

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

2. Press the "Turning" softkey.

3. Press the "Undercut" softkey.

4. Press the "Thread undercut DIN" softkey.
   The "Thread Undercut (DIN 76)" input window opens.
   - OR -

   Press the "Thread undercut" softkey.
   The "Thread Undercut" input window opens.

G code program (undercut, thread DIN) parameters				ShopMill program parameters		
PL	Machining plane			T		
SC	Safety clearance	mm		D	Cutting edge number	
F	Feedrate	*		F	Feedrate	mm/min mm/rev
				S / V	Spindle speed or Constant cutting rate	rpm m/min

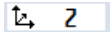
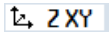
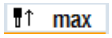
Parameter	Description	Unit
TC	Name of the swivel data set Note: The selection box only appears if more than one swivel data set has been set up.	
Retraction (for ShopMill program)	• No The axis is not retracted before swiveling • Z Z Retraction in the direction of machine axis Z • Z,X,Y Z XY Move machining axes to retraction position before swiveling • Tool direction, max. max Maximum retraction (up to the software end position) in the tool direction • Tool direction, inc. ink Incremental retraction (specified retraction distance, up to the software end position) in the tool direction When retracting in the tool direction, in the swiveled machine state, several axes can move (travel).	
ZR (for ShopMill program)	Retraction path - only for incremental retraction in	mm
Align tool through beta and gamma angles		
β (for ShopMill program)	Align tool with swivel axes • Input value The required angle can be freely entered • β = 0° ↓ • β = 90° ←	Degrees

Parameter	Description	Unit
γ (for ShopMill program) 	• 0° • 180° • The required angle can be freely entered	Degrees
Directly position rotary axes		
B1 (for ShopMill program)	Directly align the tool with the swiveling axes: The required angle can be freely entered	Degrees
C1 (for ShopMill program)	The required angle can be freely entered	Degrees
αC (for ShopMill program)	Rotational position for a pole position	Degrees
Hirth joint 	• Round to the next Hirth joint • Round up to Hirth joint • Round down to Hirth joint Note: For machines with a Hirth joint	
Tool 	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.	
Preferred direction (for ShopMill program) 	Preferred direction of the swivel axis for several possible alignments of the machine 	
Machining 	• ∇ (roughing) • ∇∇∇ (finishing) • ∇ + ∇∇∇ (roughing and finishing)	

Parameter	Description	Unit
Position ↻	Machining position: 	
Machining direction ↻	• Longitudinal • Parallel to the contour	
Form ↻	• Normal (form A) • Short (form B)	
P ↻	Thread pitch (select from the preset DIN table or enter)	mm/rev
X0	Reference point X ∅	mm
Z0	Reference point Z	mm
α	Insertion angle	Degrees
VX ↻	Cross feed ∅ (abs) or cross feed (inc) - (only for ∇∇∇ and ∇ + ∇∇∇)	mm
D	Maximum depth infeed – (only for ∇ and ∇ + ∇∇∇)	mm
U or UX ↻	Finishing allowance in X or finishing allowance in X and Z – (only for ∇ and ∇ + ∇∇∇)	mm
UZ	Finishing allowance in Z – (only for UX, ∇ and ∇ + ∇∇∇)	mm

* Unit of feedrate as programmed before the cycle call

G code program (undercut, thread) parameters				ShopMill program parameters		
PL ↻	Machining plane			T		
SC	Safety clearance	mm		D	Cutting edge number	
F	Feedrate	*		F ↻	Feedrate	mm/min mm/rev
				S / V ↻	Spindle speed or Constant cutting rate	rpm m/min

Parameter	Description	Unit
TC 	Name of the swivel data set Note: The selection box only appears if more than one swivel data set has been set up.	
Retraction (for ShopMill program) 	• No The axis is not retracted before swiveling • Z  Retraction in the direction of machine axis Z • Z,X,Y  Move machining axes to retraction position before swiveling • Tool direction, max.  Maximum retraction (up to the software end position) in the tool direction • Tool direction, inc. Incremental retraction (specified retraction distance, up to the software end position) in the tool direction When retracting in the tool direction, in the swiveled machine state, several axes can move (travel).	
ZR (for ShopMill program)	Retraction path - only for incremental retraction in	mm
Align tool through beta and gamma angles		
β (for ShopMill program) 	Align tool with swivel axes • Input value The required angle can be freely entered • β = 0° • β = 90° 	Degrees
γ (for ShopMill program) 	• 0° • 180° • The required angle can be freely entered	Degrees
Directly position rotary axes		
B1 (for ShopMill program)	Directly align the tool with the swiveling axes: The required angle can be freely entered	Degrees
C1 (for ShopMill program)	The required angle can be freely entered	Degrees

Parameter	Description	Unit
αC (for ShopMill program)	Rotational position for a pole position	Degrees
Hirth joint	• Round to the next Hirth joint • Round up to Hirth joint • Round down to Hirth joint Note: For machines with a Hirth joint	
Tool	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.	
Preferred direction (for ShopMill program)	Preferred direction of the swivel axis for several possible alignments of the machine 	
Machining	• ∇ (roughing) • ∇∇∇ (finishing) • ∇ + ∇∇∇ (roughing and finishing)	
Machining direction	• Longitudinal • Parallel to the contour	
Position	Machining position: 	
X0	Reference point X ∅	mm
Z0	Reference point Z	mm
X1	Undercut depth referred to X ∅ (abs) or undercut depth referred to X (inc)	

Parameter	Description	Unit
Z1	Allowance Z (abs or inc)	
R1	Rounding radius 1	mm
R2	Rounding radius 2	mm
α	Insertion angle	Degrees
VX	Cross feed ∅ (abs) or cross feed (inc) - (only for ∇∇∇ and ∇ + ∇∇∇)	
D	Maximum depth infeed – (only for ∇ and ∇ + ∇∇∇)	mm
U or UX	Finishing allowance in X or finishing allowance in X and Z – (only for ∇ and ∇ + ∇∇∇)	mm
UZ	Finishing allowance in Z – (only for UZ, ∇ and ∇ + ∇∇∇)	mm

* Unit of feedrate as programmed before the cycle call

10.5.6 Thread turning (CYCLE99), only for G code

Function

The "Longitudinal thread", "Tapered thread" or "Face thread" cycle is used to turn external or internal threads with a constant or variable pitch.

There may be single or multiple threads.

For metric threads (thread pitch P in mm/rev), the cycle assigns a value (calculated on the basis of the thread pitch) to the thread depth H1 parameter. You can change this value.

The default must be activated via setting data SD 55212 $SCS_FUNCTION_MASK_TECH_SET.

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

The cycle requires a speed-controlled spindle with a position measuring system.

Interruption of thread cutting

You have the option to interrupt thread cutting (for example if the cutting tool is broken).

1. Press the <CYCLE STOP> key.
   The tool is retracted from the thread and the spindle is stopped.
2. Replace the tool and press the <CYCLE START> key.
   The aborted thread cutting is started again with the interrupted cut at the same depth.

Thread re-machining

You have the option of subsequently machining threads. To do this, change into the "JOG" operating mode and carry out a thread synchronization.

Input simple

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

Machine manufacturer

Various defined values can be pre-assigned using setting data.

Please refer to the machine manufacturer's specifications.

If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

Approach/retraction

1. The tool moves to the starting point calculated internally in the cycle at rapid traverse.
2. Thread with advance:
   The tool moves at rapid traverse to the first starting position displaced by the thread advance LW.
   Thread with run-in:
   The tool moves at rapid traverse to the starting position displaced by the thread run-in LW2.
3. The first cut is made with thread pitch P as far as the thread run-out LR.
4. Thread with advance:
   The tool moves at rapid traverse to the return distance VR and then to the next starting position.
   Thread with run-in:
   The tool moves at rapid traverse to the return distance VR and then back to the starting position.
5. Steps 3 and 4 are repeated until the thread is finished.
6. The tool moves back to the retraction plane at rapid traverse.

Thread machining can be stopped at any time with the "Rapid lift" function. It ensures that the tool does not damage the thread when it is raised.

Start and end of thread

At the start of the thread, a distinction is made between thread lead (parameter LW) and thread run-in (parameter LW2).

If you program a thread lead, the programmed starting point is moved forward by this amount. You use the thread lead if the thread starts outside the material, for example, on the shoulder of a turned part.

If you program a thread run-in, an additional thread block is generated internally in the cycle. The thread block is inserted in front of the actual thread on which the tool is inserted. You require thread run-in if you want to cut a thread on the middle of a shaft.

If you program a thread run-out > 0, an additional thread block is generated at the end of the thread.

Note

Commands DITS and DITE

In CYCLE99, the commands DITS and DITE are not programmed. The setting data SD 42010 $SC_THREAD_RAMP_DISP[0] and [1] are not changed.

The parameters thread run-in (LW2) and thread run-out (LR) used in the cycles have a purely geometrical meaning. They do not influence the dynamic response of the thread blocks. The parameters result internally in a concatenation of several thread blocks.

Procedure for longitudinal thread, tapered thread, or face thread

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

2. Press the "Turning" softkey.

3. Press the "Thread" softkey.
   The "Thread" input window opens.

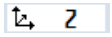
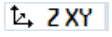
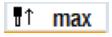
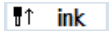
4. Press the "Longitudinal thread" softkey.
   The "Longitudinal Thread" input window opens.
   - OR -

   Press the "Tapered thread" softkey.
   The "Tapered Thread" input window opens.
   - OR -

   Press the "Face thread" softkey.
   The "Face Thread" input window opens.

Parameters in the "Complete" mode (longitudinal thread)

G code program parameters				**ShopMill program parameters**		
Input		• Complete				
PL	Machining plane			T	Tool name	
SC	Safety clearance	mm		D	Cutting edge number	
				S / V Constant cutting rate	Spindle speed or Constant cutting rate	rpm m/min

Parameter	Description	Unit
TC 	Name of the swivel data set Note: The selection box only appears if more than one swivel data set has been set up.	
Retraction (for ShopMill program) 	• No The axis is not retracted before swiveling • Z  Retraction in the direction of machine axis Z • Z,X,Y  Move machining axes to retraction position before swiveling • Tool direction, max.  Maximum retraction (up to the software end position) in the tool direction • Tool direction, inc.  Incremental retraction (specified retraction distance, up to the software end position) in the tool direction When retracting in the tool direction, in the swiveled machine state, several axes can move (travel).	
ZR (for ShopMill program)	Retraction path - only for incremental retraction in	mm
Align tool through beta and gamma angles		
β (for ShopMill program) 	Align tool with swivel axes • Input value The required angle can be freely entered • β = 0° • β = 90° 	Degrees
γ (for ShopMill program) 	• 0° • 180° • The required angle can be freely entered	Degrees
Directly position rotary axes		
B1 (for ShopMill program)	Directly align the tool with the swiveling axes: The required angle can be freely entered	Degrees
C1 (for ShopMill program)	The required angle can be freely entered	Degrees

Parameter	Description	Unit
αC (for ShopMill program)	Rotational position for a pole position	Degrees
Hirth joint 	• Round to the next Hirth joint • Round up to Hirth joint • Round down to Hirth joint Note: For machines with a Hirth joint	
Tool 	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.	
Preferred direction (for ShopMill program) 	Preferred direction of the swivel axis for several possible alignments of the machine 	
Table 	Thread table selection: • Without • ISO metric • Whitworth BSW • Whitworth BSP • UNC	
Selection - (not for table "Without") 	Data, table value, e.g. M10, M12, M14, ...	
P 	Select the thread pitch/turns for table "Without" or specify the thread pitch/turns corresponding to the selection in the thread table: • Thread pitch in mm/revolution • Thread pitch in inch/revolution • Thread turns per inch • Thread pitch in MODULUS	mm/rev in/rev turns/" MODULUS

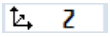
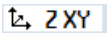
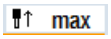
Parameter	Description	Unit
G	Change in thread pitch per revolution - (only for P = mm/rev or in/rev) G = 0: The thread pitch P does not change. G > 0: The thread pitch P increases by the value G per revolution. G < 0: The thread pitch P decreases by the value G per revolution. If the start and end pitch of the thread are known, the pitch change to be programmed can be calculated as follows: $G = \frac{\lvert P_e^2 - P^2 \rvert}{2 * Z_1}$ [mm/rev²] The meanings are as follows: P_e: End pitch of thread [mm/rev] P: Start pitch of thread [mm/rev] Z_1: Thread length [mm] A larger pitch results in a larger distance between the thread turns on the workpiece.	mm/rev²
Machining U	• ∇ (roughing) • ∇∇∇ (finishing) • ∇ + ∇∇∇ (roughing and finishing)	
Infeed (only for ∇ and ∇ + ∇∇∇) U	• Linear: Infeed with constant cutting depth • Degressive: Infeed with constant cutting cross-section	
Thread U	• Internal thread • External thread	
X0	Reference point X from thread table ∅ (abs)	mm
Z0	Reference point Z (abs)	mm
Z1 U	End point of the thread (abs) or thread length (inc) Incremental dimensions: The sign is also evaluated.	mm
Amount of crown U	Allowance to compensate for sag (- only for external thread and G= 0) • XS Segment height, crowned thread • RS Radius crowned thread Positive values: Convex Negative values: Concave Note: The pitch change per revolution "G" must be "0".	 mm mm

Parameter	Description	Unit
LW or LW2 or LW2 = LR	Thread advance (inc) The starting point for the thread is the reference point (X0, Z0) brought forward by the thread advance W. The thread advance can be used if you wish to begin the individual cuts slightly earlier in order to also produce a precise start of thread. Thread run-in (inc) The thread run-in can be used if you cannot approach the thread from the side and instead have to insert the tool into the material (e.g. lubrication groove on a shaft). Thread run-in = thread run-out (inc)	mm mm mm
LR	Thread run-out (inc) The thread run-out can be used if you wish to retract the tool obliquely at the end of the thread (e.g. lubrication groove on a shaft).	mm
H1	Thread depth from thread table (inc)	mm
DP or αP	Infeed slope as flank (inc) – (alternative to infeed slope as angle) DP > 0: Infeed along the rear flank DP < 0: Infeed along the front flank Infeed slope as angle – (alternative to infeed slope as flank) α > 0: Infeed along the rear flank α < 0: Infeed along the front flank α = 0: Infeed at right angle to cutting direction If you wish to infeed along the flanks, the maximum absolute value of this parameter may be half the flank angle of the tool.	 Degrees
 	Infeed along the flank Infeed with alternating flanks (alternative) Instead of infeed along one flank, you can infeed along alternating flanks to avoid always loading the same tool cutting edge. As a consequence you can increase the tool life. α > 0: Start at the rear flank α < 0: Start at the front flank	
D1 or ND (only for ∇ and ∇ + ∇∇∇)	First infeed depth or number of roughing cuts The respective value is displayed when you switch between the number of roughing cuts and the first infeed.	mm
½ or 1	Halve first infeed depth Normal first infeed depth	
rev	Finishing allowance in X and Z – (only for ∇ and ∇ + ∇∇∇)	mm
NN	Number of noncuts - (only for ∇∇∇ and ∇ + ∇∇∇)	
VR	Return distance (inc)	mm

Parameter	Description		Unit
Multiple threads	**No**		
	α0	Starting angle offset	Degrees
	Yes		
	N	Number of thread turns The thread turns are distributed evenly around the circumference of the turned part; the 1st thread turn is always located at 0°.	
	DA	Thread changeover depth (inc) First machine all thread turns sequentially to thread changeover depth DA, then machine all thread turns sequentially to depth 2 x DA, etc. until the final depth is reached. DA = 0: Thread changeover depth is not taken into account, i.e. finish machining each thread before starting the next thread.	mm
	Machining:	• Complete, or • From thread N1 N1 (1...4) start thread N1 = 1...N or • Only thread NX NX (1...4) 1 from N threads	

Parameters in the "input simple" mode (longitudinal thread)

G code program parameters				ShopMill program parameters		
Input		• simple				
				T	Tool name	
				D	Cutting edge number	
				S / V	Spindle speed or	rpm
					Constant cutting rate	m/min

Parameter	Description	Unit
TC 	Name of the swivel data set Note: The selection box only appears if more than one swivel data set has been set up.	
Retraction (for ShopMill program) 	• No The axis is not retracted before swiveling • Z  Retraction in the direction of machine axis Z • Z,X,Y  Move machining axes to retraction position before swiveling • Tool direction, max.  Maximum retraction (up to the software end position) in the tool direction • Tool direction, inc. Incremental retraction (specified retraction distance, up to the software end position) in the tool direction When retracting in the tool direction, in the swiveled machine state, several axes can move (travel).	
ZR (for ShopMill program)	Retraction path - only for incremental retraction in	mm
Align tool through beta and gamma angles		
β (for ShopMill program) 	Align tool with swivel axes • Input value The required angle can be freely entered • β = 0° • β = 90° 	Degrees
γ (for ShopMill program) 	• 0° • 180° • The required angle can be freely entered	Degrees
Directly position rotary axes		
B1 (for ShopMill program)	Directly align the tool with the swiveling axes: The required angle can be freely entered	Degrees
C1 (for ShopMill program)	The required angle can be freely entered	Degrees

Parameter	Description	Unit
αC (for ShopMill program)	Rotational position for a pole position	Degrees
Hirth joint	• Round to the next Hirth joint • Round up to Hirth joint • Round down to Hirth joint Note: For machines with a Hirth joint	
Tool	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.	
Preferred direction (for ShopMill program)	Preferred direction of the swivel axis for several possible alignments of the machine 	
P	Select the thread pitch/turns: • Thread pitch in mm/revolution • Thread pitch in inch/revolution • Thread turns per inch • Thread pitch in MODULUS	mm/rev in/rev turns/" MODULUS
Machining	• ∇ (roughing) • ∇∇∇ (finishing) • ∇ + ∇∇∇ (roughing and finishing)	
Infeed (only for ∇ and ∇ + ∇∇∇)	• Linear: Infeed with constant cutting depth • Degressive: Infeed with constant cutting cross-section	
Thread	• Internal thread • External thread	
X0	Reference point X from thread table ∅ (abs)	mm
Z0	Reference point Z (abs)	mm
Z1	End point of the thread (abs) or thread length (inc) Incremental dimensions: The sign is also evaluated.	mm

Parameter	Description	Unit
LW or LW2 or LW2 = LR	Thread advance (inc) The starting point for the thread is the reference point (X0, Z0) brought forward by the thread advance W. The thread advance can be used if you wish to begin the individual cuts slightly earlier in order to also produce a precise start of thread. Thread run-in (inc) The thread run-in can be used if you cannot approach the thread from the side and instead have to insert the tool into the material (e.g. lubrication groove on a shaft). Thread run-in = thread run-out (inc)	mm mm mm
LR	Thread run-out (inc) The thread run-out can be used if you wish to retract the tool obliquely at the end of the thread (e.g. lubrication groove on a shaft).	mm
H1	Thread depth from thread table (inc)	mm
DP or	Infeed slope as flank (inc) – (alternative to infeed slope as angle) DP > 0: Infeed along the rear flank DP < 0: Infeed along the front flank	
αP	Infeed slope as angle – (alternative to infeed slope as flank) α > 0: Infeed along the rear flank α < 0: Infeed along the front flank α = 0: Infeed at right angle to cutting direction If you wish to infeed along the flanks, the maximum absolute value of this parameter may be half the flank angle of the tool.	Degrees
	Infeed along the flank Infeed with alternating flanks (alternative) Instead of infeed along one flank, you can infeed along alternating flanks to avoid always loading the same tool cutting edge. As a consequence you can increase the tool life. α > 0: Start at the rear flank α < 0: Start at the front flank	
D1 or ND (only for ∇ and ∇ + ∇∇∇)	First infeed depth or number of roughing cuts The respective value is displayed when you switch between the number of roughing cuts and the first infeed.	mm
rev	Finishing allowance in X and Z – (only for ∇ and ∇ + ∇∇∇)	mm
NN	Number of noncuts - (only for ∇∇∇ and ∇ + ∇∇∇)	

Hidden parameters

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.

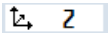
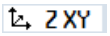
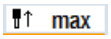
Parameter	Description	Value	Can be set in SD
PL	Machining plane	Defined in MD 52005	
Table	Thread table selection	Without	

Parameter	Description	Value	Can be set in SD
G	Change in thread pitch per revolution – (only for P = mm/rev or in/rev): Without change in thread pitch	0	
½	Halve first infeed depth	Yes	
VR	Return distance	2 mm	x
Multiple threads	1 thread	No	
α0	Starting angle offset	0°	

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

Parameters in the "complete" mode (face thread)

G code program parameters				ShopMill program parameters		
Input		• complete				
PL	Machining plane			T	Tool name	
				D	Cutting edge number	
				S / V 	Spindle speed or Constant cutting rate	rpm m/min

Parameter	Description	Unit
TC 	Name of the swivel data set Note: The selection box only appears if more than one swivel data set has been set up.	
Retraction (for ShopMill program) 	• No The axis is not retracted before swiveling • Z  Retraction in the direction of machine axis Z • Z,X,Y  Move machining axes to retraction position before swiveling • Tool direction, max.  Maximum retraction (up to the software end position) in the tool direction • Tool direction, inc. Incremental retraction (specified retraction distance, up to the software end position) in the tool direction When retracting in the tool direction, in the swiveled machine state, several axes can move (travel).	
ZR (for ShopMill program)	Retraction path - only for incremental retraction in	mm
Align tool through beta and gamma angles		
β (for ShopMill program) 	Align tool with swivel axes • Input value The required angle can be freely entered • β = 0° • β = 90° 	Degrees
γ (for ShopMill program) 	• 0° • 180° • The required angle can be freely entered	Degrees
Directly position rotary axes		
B1 (for ShopMill program)	Directly align the tool with the swiveling axes: The required angle can be freely entered	Degrees
C1 (for ShopMill program)	The required angle can be freely entered	Degrees

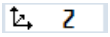
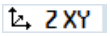
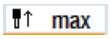
Parameter	Description	Unit
αC (for ShopMill program)	Rotational position for a pole position	Degrees
Hirth joint 	• Round to the next Hirth joint • Round up to Hirth joint • Round down to Hirth joint Note: For machines with a Hirth joint	
Tool 	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.	
Preferred direction (for ShopMill program) 	Preferred direction of the swivel axis for several possible alignments of the machine 	
P 	• Thread pitch in mm/revolution • Thread pitch in inch/revolution • Thread turns per inch • Thread pitch in MODULUS	mm/rev in/rev turns/" MODULUS
G	Change in thread pitch per revolution - (only for P = mm/rev or in/rev) G = 0: The thread pitch P does not change. G > 0: The thread pitch P increases by the value G per revolution. G < 0: The thread pitch P decreases by the value G per revolution. If the start and end pitch of the thread are known, the pitch change to be programmed can be calculated as follows: $G = \frac{\lvert P_e^2 - P^2 \rvert}{2 * Z_1}$ [mm/rev²] The meanings are as follows: P_e: End pitch of thread [mm/rev] P: Start pitch of thread [mm/rev] Z_1: Thread length [mm] A larger pitch results in a larger distance between the thread turns on the workpiece.	mm/rev²

Parameter	Description	Unit
Machining ○	• ∇ (roughing) • ∇∇∇ (finishing) • ∇ + ∇∇∇ (roughing and finishing)	
Infeed (only for ∇ and ∇ + ∇∇∇) ○	• Linear: Infeed with constant cutting depth • Degressive: Infeed with constant cutting cross-section	
Thread ○	• Internal thread • External thread	
X0	Reference point X ∅ (abs, always diameter)	mm
Z0	Reference point Z (abs)	mm
X1 ○	End point of the thread ∅ (abs) or thread length (inc) Incremental dimensions: The sign is also evaluated.	mm
LW ○ or LW2 ○ or LW2 = LR ○	Thread advance (inc) The starting point for the thread is the reference point (X0, Z0) brought forward by the thread advance W. The thread advance can be used if you wish to begin the individual cuts slightly earlier in order to also produce a precise start of thread. Thread run-in (inc) The thread run-in can be used if you cannot approach the thread from the side and instead have to insert the tool into the material (e.g. lubrication groove on a shaft). Thread run-in = thread run-out (inc)	mm mm mm
LR	Thread run-out (inc) The thread run-out can be used if you wish to retract the tool obliquely at the end of the thread (e.g. lubrication groove on a shaft).	mm
H1	Thread depth (inc)	mm
DP ○ or αP	Infeed slope as flank (inc) – (alternative to infeed slope as angle) DP > 0: Infeed along the rear flank DP < 0: Infeed along the front flank	
	Infeed slope as angle – (alternative to infeed slope as flank) α > 0: Infeed along the rear flank α < 0: Infeed along the front flank α = 0: Infeed at right angle to cutting direction If you wish to infeed along the flanks, the maximum absolute value of this parameter may be half the flank angle of the tool.	Degrees
   ○	Infeed along the flank Infeed with alternating flanks (alternative) Instead of infeed along one flank, you can infeed along alternating flanks to avoid always loading the same tool cutting edge. As a consequence you can increase the tool life. α > 0: Start at the rear flank α < 0: Start at the front flank	

Parameter	Description		Unit
D1 or ND (only for ∇ and ∇ + ∇∇∇)	First infeed depth or number of roughing cuts The respective value is displayed when you switch between the number of roughing cuts and the first infeed.		mm
½ or 1	Halve first infeed depth Normal first infeed depth		
rev	Finishing allowance in X and Z – (only for ∇ and ∇ + ∇∇∇)		mm
NN	Number of noncuts - (only for ∇∇∇ and ∇ + ∇∇∇)		
VR	Return distance (inc)		mm
Multiple threads	**No**		
	α0	Starting angle offset	Degrees
	Yes		
	N	Number of thread turns The thread turns are distributed evenly around the circumference of the turned part; the 1st thread turn is always located at 0°.	
	DA	Thread changeover depth (inc) First machine all thread turns sequentially to thread changeover depth DA, then machine all thread turns sequentially to depth 2 · DA, etc. until the final depth is reached. DA = 0: Thread changeover depth is not taken into account, i.e. finish machining each thread before starting the next thread.	mm
	Machining:	• Complete, or • From thread N1 N1 (1...4) start thread N1 = 1...N or • Only thread NX NX (1...4) 1 from N threads	

Parameters in the "input simple" mode (face thread)

G code program parameters				ShopMill program parameters		
Input		• simple				
				T	Tool name	
				D	Cutting edge number	
				S / V	Spindle speed or	rpm
					Constant cutting rate	m/min

Parameter	Description	Unit
TC 	Name of the swivel data set Note: The selection box only appears if more than one swivel data set has been set up.	
Retraction (for ShopMill program) 	• No The axis is not retracted before swiveling • Z  Retraction in the direction of machine axis Z • Z,X,Y  Move machining axes to retraction position before swiveling • Tool direction, max.  Maximum retraction (up to the software end position) in the tool direction • Tool direction, inc. Incremental retraction (specified retraction distance, up to the software end position) in the tool direction When retracting in the tool direction, in the swiveled machine state, several axes can move (travel).	
ZR (for ShopMill program)	Retraction path - only for incremental retraction in	mm
Align tool through beta and gamma angles		
β (for ShopMill program) 	Align tool with swivel axes • Input value The required angle can be freely entered • β = 0° • β = 90° 	Degrees
γ (for ShopMill program) 	• 0° • 180° • The required angle can be freely entered	Degrees
Directly position rotary axes		
B1 (for ShopMill program)	Directly align the tool with the swiveling axes: The required angle can be freely entered	Degrees
C1 (for ShopMill program)	The required angle can be freely entered	Degrees

Parameter	Description	Unit
αC (for ShopMill program)	Rotational position for a pole position	Degrees
Hirth joint 	• Round to the next Hirth joint • Round up to Hirth joint • Round down to Hirth joint Note: For machines with a Hirth joint	
Tool 	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.	
Preferred direction (for ShopMill program) 	Preferred direction of the swivel axis for several possible alignments of the machine 	
P 	Select the thread pitch/turns: • Thread pitch in mm/revolution • Thread pitch in inch/revolution • Thread turns per inch • Thread pitch in MODULUS	mm/rev in/rev turns/" MODULUS
Machining 	• ∇ (roughing) • ∇∇∇ (finishing) • ∇ + ∇∇∇ (roughing and finishing)	
Infeed (only for ∇ and ∇ + ∇∇∇) 	• Linear Infeed with constant cutting depth • Degressive Infeed with constant cutting cross-section	
Thread 	• Internal thread • External thread	
X0	Reference point X from thread table ∅ (abs)	mm
Z0	Reference point Z (abs)	mm
X1	End point of the thread ∅ (abs) or thread length (inc) Incremental dimensions: The sign is also evaluated.	mm

Parameter	Description	Unit
LW or LW2 or LW2 = LR	Thread advance (inc) The starting point for the thread is the reference point (X0, Z0) brought forward by the thread advance W. The thread advance can be used if you wish to begin the individual cuts slightly earlier in order to also produce a precise start of thread. Thread run-in (inc) The thread run-in can be used if you cannot approach the thread from the side and instead have to insert the tool into the material (e.g. lubrication groove on a shaft). Thread run-in = thread run-out (inc)	mm mm mm
LR	Thread run-out (inc) The thread run-out can be used if you wish to retract the tool obliquely at the end of the thread (e.g. lubrication groove on a shaft).	mm
H1	Thread depth from thread table (inc)	mm
DP or	Infeed slope as flank (inc) – (alternative to infeed slope as angle) DP > 0: Infeed along the rear flank DP < 0: Infeed along the front flank	
αP	Infeed slope as angle – (alternative to infeed slope as flank) α > 0: Infeed along the rear flank α < 0: Infeed along the front flank α = 0: Infeed at right angle to cutting direction If you wish to infeed along the flanks, the maximum absolute value of this parameter may be half the flank angle of the tool.	Degrees
	Infeed along the flank Infeed with alternating flanks (alternative) Instead of infeed along one flank, you can infeed along alternating flanks to avoid always loading the same tool cutting edge. As a consequence you can increase the tool life. α > 0: Start at the rear flank α < 0: Start at the front flank	
D1 or ND (only for ∇ and ∇ + ∇∇∇)	First infeed depth or number of roughing cuts The respective value is displayed when you switch between the number of roughing cuts and the first infeed.	mm
rev	Finishing allowance in X and Z – (only for ∇ and ∇ + ∇∇∇)	mm
NN	Number of noncuts - (only for ∇∇∇ and ∇ + ∇∇∇)	

Hidden parameters

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.

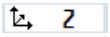
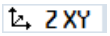
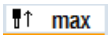
Parameter	Description	Value	Can be set in SD
PL	Machining plane	Defined in MD 52005	
G	Change in thread pitch per revolution – (only for P = mm/rev or in/rev): Without change in thread pitch	0	
½	Halve first infeed depth	Yes	
VR	Return distance	2 mm	x
Multiple threads	1 Thread	No	
α0	Starting angle offset	0°	

Machine manufacturer

Please observe the information provided by the machine manufacturer.

Parameters in the "complete" mode (tapered thread)

G code program parameters				ShopMill program parameters		
Input		• complete				
PL	Machining plane			T	Tool name	
				D	Cutting edge number	
				S / V	Spindle speed or	rpm
					Constant cutting rate	m/min

Parameter	Description	Unit
TC 	Name of the swivel data set Note: The selection box only appears if more than one swivel data set has been set up.	
Retraction (for ShopMill program) 	• No The axis is not retracted before swiveling • Z  Retraction in the direction of machine axis Z • Z,X,Y  Move machining axes to retraction position before swiveling • Tool direction, max.  Maximum retraction (up to the software end position) in the tool direction • Tool direction, inc. Incremental retraction (specified retraction distance, up to the software end position) in the tool direction When retracting in the tool direction, in the swiveled machine state, several axes can move (travel).	
ZR (for ShopMill program)	Retraction path - only for incremental retraction in	mm
Align tool through beta and gamma angles		
β (for ShopMill program) 	Align tool with swivel axes • Input value The required angle can be freely entered • β = 0° • β = 90° 	Degrees
γ (for ShopMill program) 	• 0° • 180° • The required angle can be freely entered	Degrees
Directly position rotary axes		
B1 (for ShopMill program)	Directly align the tool with the swiveling axes: The required angle can be freely entered	Degrees
C1 (for ShopMill program)	The required angle can be freely entered	Degrees

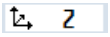
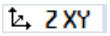
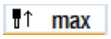
Parameter	Description	Unit
αC (for ShopMill program)	Rotational position for a pole position	Degrees
Hirth joint 	• Round to the next Hirth joint • Round up to Hirth joint • Round down to Hirth joint Note: For machines with a Hirth joint	
Tool 	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.	
Preferred direction (for ShopMill program) 	Preferred direction of the swivel axis for several possible alignments of the machine 	
P 	• Thread pitch in mm/revolution • Thread pitch in inch/revolution • Thread turns per inch • Thread pitch in MODULUS	mm/rev in/rev turns/" MODULUS
G	Change in thread pitch per revolution - (only for P = mm/rev or in/rev) G = 0: The thread pitch P does not change. G > 0: The thread pitch P increases by the value G per revolution. G < 0: The thread pitch P decreases by the value G per revolution. If the start and end pitch of the thread are known, the pitch change to be programmed can be calculated as follows: $G = \frac{\lvert P_e^2 - P^2 \rvert}{2 * Z_1}$ [mm/rev²] The meanings are as follows: P_e: End pitch of thread [mm/rev] P: Start pitch of thread [mm/rev] Z_1: Thread length [mm] A larger pitch results in a larger distance between the thread turns on the workpiece.	mm/rev²

Parameter	Description	Unit
Machining ↻	• ∇ (roughing) • ∇∇∇ (finishing) • ∇ + ∇∇∇ (roughing and finishing)	
Infeed (only for ∇ and ∇ + ∇∇∇) ↻	• Linear: Infeed with constant cutting depth • Degressive: Infeed with constant cutting cross-section	
Thread ↻	• Internal thread • External thread	
X0	Reference point X ∅ (abs, always diameter)	mm
Z0	Reference point Z (abs)	mm
X1 or X1α ↻	End point X ∅ (abs) or end point in relation to X0 (inc) or Thread taper Incremental dimensions: The sign is also evaluated.	mm or degrees
Z1 ↻	End point Z (abs) or end point in relation to Z0 (inc) Incremental dimensions: The sign is also evaluated.	mm
LW ↻ or LW2 ↻ or LW2 = LR ↻	Thread advance (inc) The starting point for the thread is the reference point (X0, Z0) brought forward by the thread advance W. The thread advance can be used if you wish to begin the individual cuts slightly earlier in order to also produce a precise start of thread. Thread run-in (inc) The thread run-in can be used if you cannot approach the thread from the side and instead have to insert the tool into the material (e.g. lubrication groove on a shaft). Thread run-in = thread run-out (inc)	mm mm mm
LR	Thread run-out (inc) The thread run-out can be used if you wish to retract the tool obliquely at the end of the thread (e.g. lubrication groove on a shaft).	mm
H1	Thread depth (inc)	mm
DP ↻ or αP	Infeed slope as flank (inc) – (alternative to infeed slope as angle) DP > 0: Infeed along the rear flank DP < 0: Infeed along the front flank	
	Infeed slope as angle – (alternative to infeed slope as flank) α > 0: Infeed along the rear flank α < 0: Infeed along the front flank α = 0: Infeed at right angle to cutting direction If you wish to infeed along the flanks, the maximum absolute value of this parameter may be half the flank angle of the tool.	Degrees

Parameter	Description		Unit
   	Infeed along the flank Infeed with alternating flanks (alternative) Instead of infeed along one flank, you can infeed along alternating flanks to avoid always loading the same tool cutting edge. As a consequence you can increase the tool life. α > 0: Start at the rear flank α < 0: Start at the front flank		
D1 or ND (only for ∇ and ∇ + ∇∇∇)	First infeed depth or number of roughing cuts The respective value is displayed when you switch between the number of roughing cuts and the first infeed.		mm
½ or 1	Halve first infeed depth Normal first infeed depth		
rev	Finishing allowance in X and Z – (only for ∇ and ∇ + ∇∇∇)		mm
NN	Number of noncuts - (only for ∇∇∇ and ∇ + ∇∇∇)		
VR	Return distance (inc)		mm
Multiple threads 	**No**		
	α0	Starting angle offset	Degrees
	Yes		
	N	Number of thread turns The thread turns are distributed evenly around the circumference of the turned part; the 1st thread turn is always located at 0°.	
	DA	Thread changeover depth (inc) First machine all thread turns sequentially to thread changeover depth DA, then machine all thread turns sequentially to depth 2 · DA, etc. until the final depth is reached. DA = 0: Thread changeover depth is not taken into account, i.e. finish machining each thread before starting the next thread.	mm
	Machining: 	• Complete, or • From thread N1 N1 (1...4) start thread N1 = 1...N or • Only thread NX NX (1...4) 1 from N threads	

Parameters in the "input simple" mode (tapered thread)

G code program parameters				ShopMill program parameters		
Input		• simple				
				T	Tool name	
				D	Cutting edge number	
				S / V 	Spindle speed or Constant cutting rate	rpm m/min

Parameter	Description	Unit
TC 	Name of the swivel data set Note: The selection box only appears if more than one swivel data set has been set up.	
Retraction (for ShopMill program) 	• No The axis is not retracted before swiveling • Z  Retraction in the direction of machine axis Z • Z,X,Y  Move machining axes to retraction position before swiveling • Tool direction, max.  Maximum retraction (up to the software end position) in the tool direction • Tool direction, inc. Incremental retraction (specified retraction distance, up to the software end position) in the tool direction When retracting in the tool direction, in the swiveled machine state, several axes can move (travel).	
ZR (for ShopMill program)	Retraction path - only for incremental retraction in	mm
Align tool through beta and gamma angles		
β (for ShopMill program) 	Align tool with swivel axes • Input value The required angle can be freely entered • β = 0° • β = 90° 	Degrees
γ (for ShopMill program) 	• 0° • 180° • The required angle can be freely entered	Degrees
Directly position rotary axes		
B1 (for ShopMill program)	Directly align the tool with the swiveling axes: The required angle can be freely entered	Degrees
C1 (for ShopMill program)	The required angle can be freely entered	Degrees

Parameter	Description	Unit
αC (for ShopMill program)	Rotational position for a pole position	Degrees
Hirth joint 	• Round to the next Hirth joint • Round up to Hirth joint • Round up to Hirth joint Note: For machines with a Hirth joint	
Tool 	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.	
Preferred direction (for ShopMill program) 	Preferred direction of the swivel axis for several possible alignments of the machine 	
P 	Select the thread pitch/turns: • Thread pitch in mm/revolution • Thread pitch in inch/revolution • Thread turns per inch • Thread pitch in MODULUS	mm/rev in/rev turns/" MODULUS
Machining 	• ∇ (roughing) • ∇∇∇ (finishing) • ∇ + ∇∇∇ (roughing and finishing)	
Infeed (only for ∇ and ∇ + ∇∇∇) 	• Linear Infeed with constant cutting depth • Degressive Infeed with constant cutting cross-section	
Thread 	• Internal thread • External thread	
X0	Reference point X from thread table ∅ (abs)	mm
Z0	Reference point Z (abs)	mm

Parameter	Description	Unit
X1 or X1α 	End point of the thread ∅ (abs) or thread length (inc) Incremental dimensions: The sign is also evaluated.	mm Degrees
Z1	End point Z (abs) or end point in relation to Z0 (inc)	mm
LW or LW2 or LW2 = LR 	Thread advance (inc) The starting point for the thread is the reference point (X0, Z0) brought forward by the thread advance W. The thread advance can be used if you wish to begin the individual cuts slightly earlier in order to also produce a precise start of thread. Thread run-in (inc) The thread run-in can be used if you cannot approach the thread from the side and instead have to insert the tool into the material (e.g. lubrication groove on a shaft). Thread run-in = thread run-out (inc)	mm mm mm
LR	Thread run-out (inc) The thread run-out can be used if you wish to retract the tool obliquely at the end of the thread (e.g. lubrication groove on a shaft).	mm
H1	Thread depth from thread table (inc)	mm
DP or αP	Infeed slope as flank (inc) – (alternative to infeed slope as angle) DP > 0: Infeed along the rear flank DP < 0: Infeed along the front flank	
	Infeed slope as angle – (alternative to infeed slope as flank) α > 0: Infeed along the rear flank α < 0: Infeed along the front flank α = 0: Infeed at right angle to cutting direction If you wish to infeed along the flanks, the maximum absolute value of this parameter may be half the flank angle of the tool.	Degrees
	Infeed along the flank Infeed with alternating flanks (alternative) Instead of infeed along one flank, you can infeed along alternating flanks to avoid always loading the same tool cutting edge. As a consequence you can increase the tool life. α > 0: Start at the rear flank α < 0: Start at the front flank	
D1 or ND (only for ∇ and ∇ + ∇∇∇)	First infeed depth or number of roughing cuts The respective value is displayed when you switch between the number of roughing cuts and the first infeed.	mm
rev	Finishing allowance in X and Z – (only for ∇ and ∇ + ∇∇∇)	mm
NN	Number of noncuts - (only for ∇∇∇ and ∇ + ∇∇∇)	

Hidden parameters

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.

Parameter	Description	Value	Can be set in SD
PL	Machining plane	Defined in MD 52005	
G	Change in thread pitch per revolution – (only for P = mm/rev or in/rev): Without change in thread pitch	0	
½	Halve first infeed depth	Yes	
VR	Return distance		x
Multiple threads	1 thread	No	
α0	Starting angle offset	0°	

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

10.5.6.1 Special aspects of the selection alternatives for infeed depths

Note the following special aspects of the selection alternatives for roughing between the parameters D1 "First maximum infeed depth" and ND "Number of roughing cuts" and the selection alternative "Halve first infeed" and "First infeed normal" with degressive infeed.

For programming the number of cuts via parameter ND, the depth for the individual infeeds from the thread depth is divided by the number of cuts (constant infeed mode) or is calculated internally in such a way that the cross-sectional area of cut remains constant (degressive infeed mode).

For the degressive infeed, you can choose between "Halve first infeed" and "First infeed normal". The calculation algorithm for n-infeeds for "Halve first infeed" is comparable with the calculation algorithm n-1 infeeds for "First infeed normal". The only difference is: The first infeed is halved, so one additional infeed is required in total.

For programming the first infeed via parameter D1, the division calculated internally in the cycle also depends on the selection constant or degressive infeed.

For constant infeed, the required number of infeeds is calculated based on D1 in such a way that infeed is always executed by the same value. This can cause a smaller infeed value to be applied than the value programmed in D1, resulting in an integer multiple of the actual cut depth from the thread depth.

Example 1

For a thread depth of 7 mm and a first infeed of 2 mm, 4 cuts of 1.75 mm each are made in this mode.

With degressive infeed, the cross-sectional area of cut for the total thread depth is divided into cuts in such a way as to produce an integer multiple of the cross-sectional area of cut per infeed actually executed. In many cases, this also means that the first infeed is less than the value programmed in D1. Here, too, you can choose between "Halve first infeed" and "First infeed

normal". When the first infeed is halved, double the value of D1 is used for calculating infeeds for the constant cross-sectional area of cut; if it is not, the single value of D1 is used.

Example 2

For a thread depth of 7 mm and D1=2 mm and "Halve first infeed", five infeeds and for "First infeed normal", 13 infeeds are executed.

However, the first infeed is less than 2 mm.

10.5.7 Thread chain (CYCLE98)

Function

With this cycle, you can produce several concatenated cylindrical or tapered threads with a constant pitch in longitudinal and face machining, all of which can have different thread pitches.

There may be single or multiple threads. With multiple threads, the individual thread turns are machined one after the other.

You define a right or left-hand thread by the direction of spindle rotation and the feed direction.

The infeed is performed automatically with a constant infeed depth or constant cutting cross-section.

- With a constant infeed depth, the cutting cross-section increases from cut to cut. The finishing allowance is machined in one cut after roughing.
  A constant infeed depth can produce better cutting conditions at small thread depths.
- With a constant cutting cross-section, the cutting pressure remains constant over all roughing cuts and the infeed depth is reduced.

The feedrate override has no effect during traversing blocks with thread. The spindle override must not be changed during the thread machining.

Interruption of thread cutting

You have the option to interrupt thread cutting (for example if the cutting tool is broken).

1. Press the <CYCLE STOP> key.
   The tool is retracted from the thread and the spindle is stopped.
2. Replace the tool and press the <CYCLE START> key.
   The aborted thread cutting is started again with the interrupted cut at the same depth.

Input simple

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

Machine manufacturer

Various defined values can be pre-assigned using setting data.

Please refer to the machine manufacturer's specifications.

If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

Approach/retraction

1. Approach of the starting point determined in the cycle at the beginning of the run-in path for the first thread with G0
2. Infeed for roughing according to the defined infeed type.
3. Thread cutting is repeated according to the programmed number of roughing cuts.
4. The finishing allowance is removed in the following step with G33.
5. This cut is repeated according to the number of noncuts.
6. The whole sequence of motions is repeated for each further thread.

Start and end of thread

At the start of the thread, a distinction is made between thread lead (parameter LW) and thread run-in (parameter LW2).

If you program a thread lead, the programmed starting point is moved forward by this amount. You use the thread lead if the thread starts outside the material, for example, on the shoulder of a turned part.

If you program a thread run-in, an additional thread block is generated internally in the cycle. The thread block is inserted in front of the actual thread on which the tool is inserted. You require thread run-in if you want to cut a thread on the middle of a shaft.

If you program a thread run-out > 0, an additional thread block is generated at the end of the thread.

Note

Commands DITS and DITE

In CYCLE99, the commands DITS and DITE are not programmed. The setting data SD 42010 $SC_THREAD_RAMP_DISP[0] and [1] are not changed.

The parameters thread run-in (LW2) and thread run-out (LR) used in the cycles have a purely geometrical meaning. They do not influence the dynamic response of the thread blocks. The parameters result internally in a concatenation of several thread blocks.

Procedure for thread chain

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

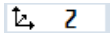
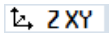
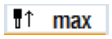
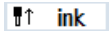
2. Press the "Turning" softkey.

3. Press the "Thread" softkey.
   The "Thread" input window opens.

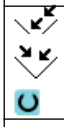
4. Press the "Thread chain" softkey.
   The "Thread Chain" input window opens.

Parameters in the "Input complete" mode

G code program parameters				**ShopMill program parameters (thread chain)**		
Input		• complete				
PL	Machining plane			T	Tool name	
				D	Cutting edge number	
				S / V Constant cutting rate	Spindle speed or Constant cutting rate	rpm m/min

Parameter	Description	Unit
TC 	Name of the swivel data set Note: The selection box only appears if more than one swivel data set has been set up.	
Retraction (for ShopMill program) 	• No The axis is not retracted before swiveling • Z  Retraction in the direction of machine axis Z • Z,X,Y  Move machining axes to retraction position before swiveling • Tool direction, max.  Maximum retraction (up to the software end position) in the tool direction • Tool direction, inc.  Incremental retraction (specified retraction distance, up to the software end position) in the tool direction When retracting in the tool direction, in the swiveled machine state, several axes can move (travel).	
ZR (for ShopMill program)	Retraction path - only for incremental retraction in	mm
Align tool through beta and gamma angles		
β (for ShopMill program) 	Align tool with swivel axes • Input value The required angle can be freely entered • β = 0° • β = 90° 	Degrees
γ (for ShopMill program) 	• 0° • 180° • The required angle can be freely entered	Degrees
Directly position rotary axes		
B1 (for ShopMill program)	Directly align the tool with the swiveling axes: The required angle can be freely entered	Degrees
C1 (for ShopMill program)	The required angle can be freely entered	Degrees

Parameter	Description	Unit
αC (for ShopMill program)	Rotational position for a pole position	Degrees
Hirth joint 	• Round to the next Hirth joint • Round up to Hirth joint • Round down to Hirth joint Note: For machines with a Hirth joint	
Tool 	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.	
Preferred direction (for ShopMill program) 	Preferred direction of the swivel axis for several possible alignments of the machine 	
Machining 	• ∇ (roughing) • ∇∇∇ (finishing) • ∇ + ∇∇∇ (roughing and finishing)	
Infeed (only for ∇ and ∇ + ∇∇∇) 	• Linear: Constant cutting depth infeed • Degressive: Constant cutting cross-section infeed	
Thread 	• Internal thread • External thread	
X0	Reference point X ⌀ (abs, always diameter)	mm
Z0	Reference point Z (abs)	mm
P0	Thread pitch 1	mm/rev in/rev turns/" MODULUS

Parameter	Description		Unit
X1 or X1α ↻	• Intermediate point 1 X ∅ (abs) or • Intermediate point 1 in relation to X0 (inc) or • Thread taper 1 Incremental dimensions: The sign is also evaluated.		mm Degrees
Z1 ↻	• Intermediate point 1 Z (abs) or • Intermediate point 1 in relation to Z0 (inc)		mm
P1	Thread pitch 2 (unit as parameterized for P0)		mm/rev in/rev turns/" MODULUS
X2 or X2α ↻	• Intermediate point 2 X ∅ (abs) or • Intermediate point 2 in relation to X1 (inc) or • Thread taper 2 Incremental dimensions: The sign is also evaluated.		mm Degrees
Z2 ↻	• Intermediate point 2 Z (abs) or • Intermediate point 2 in relation to Z1 (inc)		mm
P2	Thread pitch 3 (unit as parameterized for P0)		mm/rev in/rev turns/" MODULUS
X3 ↻	• End point X ∅ (abs) or • End point 3 in relation to X2 (inc) or • Thread taper 3		mm Degrees
Z3 ↻	• End point Z ∅ (abs) or • End point with reference to Z2 (inc)		mm
LW	Thread run-in		mm
LR	Thread run-out		mm
H1	Thread depth		mm
DP or αP ↻	Infeed slope (flank) or infeed slope (angle)		mm or degrees
 ↻	• Infeed along a flank • Infeed with alternating flanks		
D1 or ND ↻	First infeed depth or number of roughing cuts - (only for ∇ and ∇ + ∇∇∇)		mm
rev	Finishing allowance in X and Z - (only for ∇ and ∇ + ∇∇∇)		mm
NN	Number of noncuts - (only for ∇∇∇ and ∇ + ∇∇∇)		
VR	Return distance		mm
Multiple threads ↻	**No**		
	α0	Starting angle offset	Degrees
	Yes		
	N	Number of thread turns	
	DA	Thread changeover depth (inc)	mm

Parameters in the "Input simple" mode

G code program parameters				ShopMill program parameters (thread chain)		
Input		• simple				
				T	Tool name	
				D	Cutting edge number	
				S / V U	Spindle speed or Constant cutting rate	rpm m/min

Parameter	Description	Unit
TC U	Name of the swivel data set Note: The selection box only appears if more than one swivel data set has been set up.	
Retraction (for ShopMill program) U	• No The axis is not retracted before swiveling • Z Z Retraction in the direction of machine axis Z • Z,X,Y Z XY Move machining axes to retraction position before swiveling • Tool direction, max. max Maximum retraction (up to the software end position) in the tool direction • Tool direction, inc. ink Incremental retraction (specified retraction distance, up to the software end position) in the tool direction When retracting in the tool direction, in the swiveled machine state, several axes can move (travel).	
ZR (for ShopMill program)	Retraction path - only for incremental retraction in	mm
Align tool through beta and gamma angles		
β (for ShopMill program) U	Align tool with swivel axes • Input value The required angle can be freely entered • β = 0° ↓ • β = 90° ←	Degrees

Parameter	Description	Unit
γ (for ShopMill program)	• 0° • 180° • The required angle can be freely entered	Degrees
Directly position rotary axes		
B1 (for ShopMill program)	Directly align the tool with the swiveling axes: The required angle can be freely entered	Degrees
C1 (for ShopMill program)	The required angle can be freely entered	Degrees
αC (for ShopMill program)	Rotational position for a pole position	Degrees
Hirth joint	• Round to the next Hirth joint • Round up to Hirth joint • Round down to Hirth joint Note: For machines with a Hirth joint	
Tool	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.	
Preferred direction (for ShopMill program)	Preferred direction of the swivel axis for several possible alignments of the machine + –	
Machining	• ∇ (roughing) • ∇∇∇ (finishing) • ∇ + ∇∇∇ (roughing and finishing)	
Infeed (only for ∇ and ∇ + ∇∇∇)	• Linear Infeed with constant cutting depth • Degressive Infeed with constant cutting cross-section	
Thread	• Internal thread • External thread	

Parameter	Description	Unit
X0	Reference point X from thread table ∅ (abs)	mm
Z0	Reference point Z (abs)	mm
P0	Thread pitch 1	mm
X1 or X1α ○	• Intermediate point 1 X ∅ (abs) or • Intermediate point 1 in relation to X0 (inc) or • Thread taper 1 Incremental dimensions: The sign is also evaluated.	mm Degrees
Z1 ○	• Intermediate point 1 Z (abs) or • Intermediate point 1 in relation to Z0 (inc)	mm
P1	Thread pitch 2 (unit as parameterized for P0)	mm/rev in/rev turns/" MODULUS
X2 or X2α ○	• Intermediate point 2 X ∅ (abs) or • Intermediate point 2 in relation to X0 (inc) or • Thread taper 1 Incremental dimensions: The sign is also evaluated	mm Degrees
Z2 ○	• Intermediate point 2 Z (abs) or • Intermediate point 2 in relation to Z0 (inc)	mm
P2	Thread pitch 3 (unit as parameterized for P0)	mm/rev in/rev turns/" MODULUS
X3 ○	• End point X ∅ (abs) or • End point 3 in relation to X2 (inc) or • Thread taper 3	mm Degrees
Z3 ○	• End point Z ∅ (abs) or • End point with reference to Z2 (inc)	mm
LW	Thread advance (inc)	mm
LR	Thread run-out (inc)	mm
H1	Thread depth	mm
DP or αP ○	Infeed slope flank (inc) or infeed slope (angle)	mm Degrees
 ○	• Infeed along a flank • Infeed with alternating flanks	
D1 or ND ○	First infeed depth or number of roughing cuts - (only for ∇ and ∇ + ∇∇∇)	mm
rev	Finishing allowance in X and Z – (only for ∇ and ∇ + ∇∇∇)	mm
NN	Number of noncuts - (only for ∇∇∇ and ∇ + ∇∇∇)	

Hidden parameters

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.

Parameter	Description	Value	Can be set in SD
PL	Machining plane	Defined in MD 52005	
VR	Return distance		x
Multiple threads	1 Thread	No	
α0	Starting angle offset	0°	

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

10.5.7.1 Special aspects of the selection alternatives for infeed depths

Note the following special aspects of the selection alternatives for roughing between the parameters D1 "First maximum infeed depth" and ND "Number of roughing cuts" and the selection alternative "Halve first infeed" and "First infeed normal" with degressive infeed.

For programming the number of cuts via parameter ND, the depth for the individual infeeds from the thread depth is divided by the number of cuts (constant infeed mode) or is calculated internally in such a way that the cross-sectional area of cut remains constant (degressive infeed mode).

For the degressive infeed, you can choose between "Halve first infeed" and "First infeed normal". The calculation algorithm for n-infeeds for "Halve first infeed" is comparable with the calculation algorithm n-1 infeeds for "First infeed normal". The only difference is: The first infeed is halved, so one additional infeed is required in total.

For programming the first infeed via parameter D1, the division calculated internally in the cycle also depends on the selection constant or degressive infeed.

For constant infeed, the required number of infeeds is calculated based on D1 in such a way that infeed is always executed by the same value. This can cause a smaller infeed value to be applied than the value programmed in D1, resulting in an integer multiple of the actual cut depth from the thread depth.

Example 1

For a thread depth of 7 mm and a first infeed of 2 mm, 4 cuts of 1.75 mm each are made in this mode.

With degressive infeed, the cross-sectional area of cut for the total thread depth is divided into cuts in such a way as to produce an integer multiple of the cross-sectional area of cut per infeed actually executed. In many cases, this also means that the first infeed is less than the value programmed in D1. Here, too, you can choose between "Halve first infeed" and "First infeed normal". When the first infeed is halved, double the value of D1 is used for calculating infeeds for the constant cross-sectional area of cut; if it is not, the single value of D1 is used.

Example 2

For a thread depth of 7 mm and D1=2 mm and "Halve first infeed", five infeeds and for "First infeed normal", 13 infeeds are executed.

However, the first infeed is less than 2 mm.

10.5.8 Cut-off (CYCLE92)

Function

The "Cut-off" cycle is used when you want to cut off dynamically balanced parts (e.g. screws, bolts, or pipes).

You can program a chamfer or rounding on the edge of the machined part. You can machine at a constant cutting rate V or speed S up to a depth X1, from which point the workpiece is machined at a constant speed. As of depth X1, you can also program a reduced feedrate FR or a reduced speed SR, in order to adapt the velocity to the smaller diameter.

Use parameter X2 to enter the final depth that you wish to reach with the cut-off. With pipes, for example, you do not need to cut-off until you reach the center; cutting off slightly more than the wall thickness of the pipe is sufficient.

Approach/retraction

1. The tool first moves to the starting point calculated internally in the cycle at rapid traverse.
2. The chamfer or radius is machined at the machining feedrate.
3. Cut-off down to depth X1 is performed at the machining feedrate.
4. Cut-off is continued down to depth X2 at reduced feedrate FR and reduced speed SR.
5. The tool moves back to the safety distance at rapid traverse.

If your turning machine is appropriately set up, you can extend a workpiece drawer (part catcher) to accept the cut-off workpiece. Extension of the workpiece drawer must be enabled in a machine data element.

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

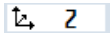
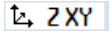
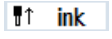
Procedure

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

2. Press the "Turning" softkey.

3. Press the "Cut-off" softkey.
   The "Cut-off" input window opens.

G code program parameters (cut-off)				ShopMill program parameters		
PL	Machining plane			T	Tool name	
SC	Safety clearance	mm		D	Cutting edge number	
F	Feedrate	*		F	Feedrate	mm/min mm/rev
				S / V	Spindle speed or Constant cutting rate	rpm m/min

Parameter	Description	Unit
TC	Name of the swivel data set Note: The selection box only appears if more than one swivel data set has been set up.	
Retraction (for ShopMill program)	• No The axis is not retracted before swiveling • Z  Retraction in the direction of machine axis Z • Z,X,Y  Move machining axes to retraction position before swiveling • Tool direction, max. Maximum retraction (up to the software end position) in the tool direction • Tool direction, inc.  Incremental retraction (specified retraction distance, up to the software end position) in the tool direction When retracting in the tool direction, in the swiveled machine state, several axes can move (travel).	
ZR (for ShopMill program)	Retraction path - only for incremental retraction in	mm
Align tool through beta and gamma angles		
β (for ShopMill program)	Align tool with swivel axes • Input value The required angle can be freely entered • β = 0° • β = 90° 	Degrees

Parameter	Description	Unit
γ (for ShopMill program) 	• 0° • 180° • The required angle can be freely entered	Degrees
Directly position rotary axes		
B1 (for ShopMill program)	Directly align the tool with the swiveling axes: The required angle can be freely entered	Degrees
C1 (for ShopMill program)	The required angle can be freely entered	Degrees
αC (for ShopMill program)	Rotational position for a pole position	Degrees
Hirth joint 	• Round to the next Hirth joint • Round up to Hirth joint • Round down to Hirth joint Note: For machines with a Hirth joint	
Tool 	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.	
Preferred direction (for ShopMill program) 	Preferred direction of the swivel axis for several possible alignments of the machine 	
DIR (only G code program) 	Direction of spindle rotation 	
S (only G code program)	Spindle speed	rpm
V (only G code program)	Constant cutting rate	mm/min
SV	Maximum speed limit - (only for constant cutting rate V)	rpm
X0	Reference point in X ∅ (abs, always diameter)	mm

Parameter	Description	Unit
Z0	Reference point in Z (abs)	mm
FS or R	Chamfer width or rounding radius	mm
X1	Depth for speed reduction ∅ (abs) or depth for speed reduction in relation to X0 (inc)	mm
FR	Reduced feedrate	*
SR	Reduced speed	rpm
X2	Final depth ∅ (abs) or final depth in relation to X1 (inc)	mm

* Unit of feedrate as programmed before the cycle call

10.6 Contour turning - Milling/turning machine

10.6.1 General information

Function

You can machine simple or complex contours with the "Contour turning" cycle. A contour comprises separate contour elements, whereby at least two and up to 250 elements result in a defined contour.

You can program chamfers, radii, undercuts or tangential transitions between the contour 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.

When you machine the contour, you can make allowance for a blank contour, which must be entered before the finished-part contour. Then select one of the the following machining technologies:

- Stock removal
- Grooving
- Plunge-turning

You can rough, remove residual material and finish for each of the three technologies above.

Note

Starting point or end point of the machining outside the retraction planes

For programs with contour machining from earlier software releases, for an NC start, it is possible that one of the alarms 61281 "Starting point of machining outside retraction planes" or 61282 "End point of machining outside retraction planes" is output.

In this case, adapt the retraction planes in the program header.

Programming

For example, the programming procedure for stock removal is as follows:

Note

When programming in the G code, it must be ensured that the contours are located after the end of program identifier!

1. Enter the unmachined-part contour
   If, when removing stock along the contour, you want to take into account an unmachined part contour (and no cylinder or no allowance) as unmachined part shape, then you must define the contour of the unmachined part before you define the finished-part contour. Compile the unmachined-part contour step-by-step from various contour elements.
2. Enter finished-part contour
   You build up the finished-part contour gradually from a series of different contour elements.
3. Contour call
4. Stock removal along the contour (roughing)
   The contour is machined longitudinally, transversely or parallel to the contour.
5. Remove residual material (roughing)
   For G code programming, when removing stock, it must first be decided whether to rough (machine) with residual material detection or not. A suitable tool will allow you to remove this without having to machine the contour again.
6. Stock removal along the contour (finishing)
   If you programmed a finishing allowance for roughing, the contour is machined again.

10.6.2 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.

Symbolic representation

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

Contour element	Symbol	Meaning
Starting point		Starting point of the contour
Straight line up		Straight line in 90° grid
Straight line down		Straight line in 90° grid
Straight line left		Straight line in 90° grid
Straight line right		Straight line in 90° grid
Straight line in any direction		Straight line with any gradient
Arc right		Circle
Arc left		Circle

Contour element	Symbol	Meaning
Pole		Straight diagonal or circle in polar coordinates
Finish contour	END	End of contour definition

The different colors of the symbols 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.

10.6.3 Creating a new contour

Function

For each contour that you want to cut, you must create a new contour.

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

Procedure

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

2. Press the "Turning" and "Contour turning" softkeys.

3. Press the "Contour" and "New contour" softkeys.
   The "New Contour" input window opens.

4. Enter a name for the new contour. The contour name must be unique.

5. Press the "Accept" softkey.
   The input window for the starting point of the contour appears.
   Enter the individual contour elements (see Section "Creating contour elements").

Parameter	Description	Unit
Z	Starting point Z (abs)	mm
X	Starting point X ∅ (abs)	mm
Transition to contour start 	Type of transition • Radius • Chamfer FS=0 or R=0: No transition element	
R	Transition to following element – radius	mm
FS	Transition to following element – chamfer	mm

Parameter	Description	Unit
Direction in front of the contour 	Direction of the contour element towards the starting point: • In the negative direction of the horizontal axis • In the positive direction of the horizontal axis • In the negative direction of the vertical axis • In the positive direction of the vertical axis	
Additional commands	You can enter additional commands in the form of G code for each contour element. You can enter the additional commands (max. 40 characters) in the extended parameter screens ("All parameters" softkey). The softkey is always available at the starting point, it only has to be pressed when entering additional contour elements. You can program feedrates and M commands, for example, using additional G code commands. However, carefully ensure that the additional commands do not collide with the generated G code of the contour and are compatible with the machining type required. 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. 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 straight X parallel and then enter "G9" (non-modal exact stop) for the additional command parameter. Then program the Z-parallel straight line. The corner will be machined exactly, as the feedrate at the end of the X-parallel straight line is briefly zero. **Note:** The additional commands are only effective for finishing!	

10.6.4 Creating contour elements

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

- Straight diagonal line
- Circle/arc

For each contour element, you must parameterize a separate parameter screen. Parameter entry is supported by various help screens that explain these parameters.

If you leave certain fields blank, the cycle 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 entering less parameters and allowing the cycle to calculate as many parameters as possible.

Contour transition elements

As transition element between two contour elements, you can select a radius or a chamfer or, in the case of linear contour elements, an undercut. 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.

Additional commands

You can enter additional commands in the form of G code for each contour element. You can enter the additional commands (max. 40 characters) in the extended parameter screens ("All parameters" softkey).

You can program feedrates and M commands, for example, using additional G-code commands. 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.

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 thus far, one of the options must be selected.
- Close contour

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

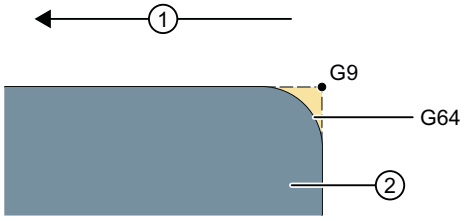
Producing exact contour transitions

The continuous path mode (G64) is used. This means, that contour transitions such as corners, chamfers or radii may not be machined precisely.

If you wish to avoid this, there are two different options when programming. Use the additional programs or program the special feedrate for the transition element.

- Additional command

For a contour, first program the vertical straight line and then enter "G9" (non-modal exact stop) for the additional command parameter. Then program the horizontal straight line. The corner will be machined exactly, since the feedrate at the end of the vertical straight line is briefly zero.



① Machining direction
② Workpiece

- Feedrate, transition element

If you have chosen a chamfer or a radius as the transition element, enter a reduced feedrate in the "FRC" parameter. The slower machining rate means that the transition element is machined more accurately.

Procedure for entering contour elements

1. The part program is opened. Position the cursor at the required input position, this is generally at the physical end of the program after M02 or M30.
2. Contour input using contour support:

2.1 Press the "Contour turning", "Contour" and "New contour" softkeys.

2.2 In the opened input window, enter a name for the contour, e.g. contour_1.

Press the "Accept" softkey.

2.3 The input screen to enter the contour opens, in which you initially enter a starting point for the contour. This is marked in the lefthand navigation bar using the "+" symbol.

Press the "Accept" softkey.

3. Enter the individual contour elements of the machining direction.
   Select a contour element via softkey.

   The "Straight (e.g. Z)" input window opens.

   - OR

   The "Straight (e.g. X)" input window opens.

   - OR

   The "Straight (e.g. ZX)" input window opens.

   - OR

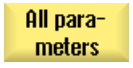
   The "Circle" input window opens.

4. Enter all the data available from the workpiece drawing in the input screen (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 into the process plan (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.

Contour element "Straight line e.g. Z"

Parameters	Description		Unit
Z	End point Z (abs or inc)		mm
α1	Starting angle to Z axis		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

Parameters	Description			Unit
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			

Contour element "Straight line e.g. X"

Parameters	Description			Unit
X	End point X ∅ (abs) or end point X (inc)			mm
α1	Starting angle to Z axis			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
CA	Grinding allowance • Grinding allowance to right of contour • Grinding allowance to left of contour			mm
Additional commands	Additional G code commands			

Contour element "Straight line e.g. ZX"

Parameters	Description		Unit
Z	End point Z (abs or inc)		mm
X	End point X ⌀ (abs) or end point X (inc)		mm
α1	Starting angle to Z 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
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"

Parameters	Description		Unit
Direction of rotation	• Clockwise direction of rotation • Counterclockwise direction of rotation 		
Z	End point Z (abs or inc)		mm
X	End point X ⌀ (abs) or end point X (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 Z axis		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
CA	Grinding allowance • Grinding allowance to right of contour • Grinding allowance to left of contour		mm
Additional commands	Additional G code commands		

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.

10.6.5 Changing the contour

Function

You can change a previously created contour later.

Individual contour elements can be

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

Procedure for changing a contour element

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

6. 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 executed.
2. Position the cursor on the contour element that you want to delete.

3. Press the "Delete element" softkey.

4. Press the "Delete" softkey.

10.6.6 Contour call (CYCLE62)

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.

Procedure

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

2. Press the "Turning" and "Contour turning" softkeys.

3. Press the "Contour" and "Contour call" softkeys.
   The "Contour Call" input window opens.

4. Assign parameters to the contour selection.

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

EXTCALL / EES

When calling a part program via EXTCALL without EES, the contour can only be called via “Contour name” and/or “Labels”. This is monitored in the cycle, which means that contour calls via "subprogram" or "labels in subprogram" are only possible if EES is active.

10.6.7 Stock removal (CYCLE952)

Function

You can use the "Stock removal" function to machine contours in the longitudinal or transverse direction or parallel to the contour.

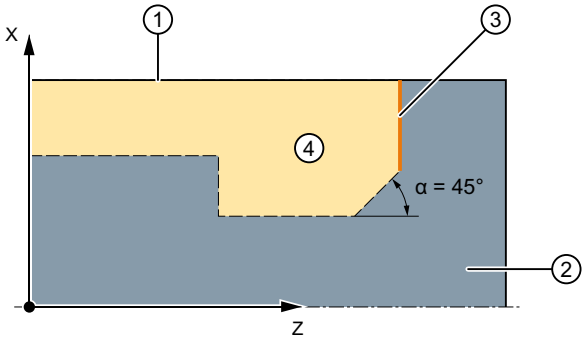
Blank

For stock removal, the cycle takes into account a blank that can comprise a cylinder, an allowance on the finished part contour or any other blank contour. You must define a blank contour as a separate closed contour in advance of the finished part contour.

Note

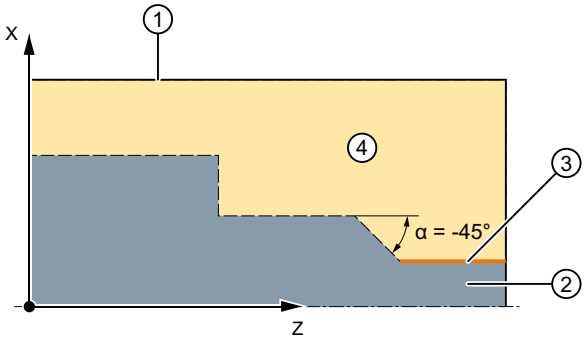
In order to avoid collisions between tools and workpieces due to positioning motions, the programmed blank contour must match the real blank.

If the blank and finished part contours do not intersect, the cycles defines the boundary between blank and finished part. If the angle between the straight line and the Z axis is greater than 1°, the boundary is placed at the top - and if the angle is less than or equal to 1°, the boundary is placed at the side.



① Blank
② Finished part
③ End of contour
④ Machining

Figure 10-10 α > 1: Boundary between unmachined and finished parts at the top



① Blank
② Finished part
③ End of contour
④ Machining

Figure 10-11 α ≤ 1°: Boundary between unmachined and finished parts at the side

Requirement

For a G code program, at least one CYCLE62 is required before CYCLE952.

If CYCLE62 is only present once, then this involves the finished part contour.

If CYCLE62 is present twice, then the first call is the blank contour and the second call is the finished part contour (also see Section "Programming (Page 592)").

Note

Execution from external media

If you want to execute programs from an external drive (e.g. local drive or network drive), you require the "Execution from external storage (EES)" function.

Rounding the contour

In order to avoid residual corners during roughing, you can enable the "Always round the contour" function. This will remove the protrusions that are always left at the end of the contour, due to the cut geometry. The "Round to the previous intersection" setting accelerates machining of the contour. However, any resulting residual corners will not be recognized or machined. Thus, it is imperative that you check the behavior before machining using the simulation.

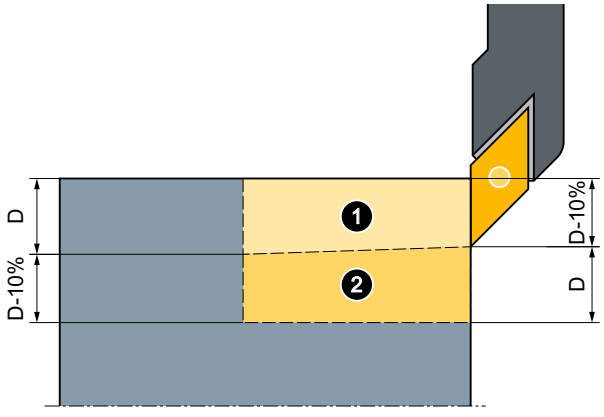
When set to "automatic", rounding is always performed if the angle between the cutting edge and the contour exceeds a certain value. The angle is set in a machine data element.

Machine manufacturer

Please observe the information provided by the machine manufacturer.

Alternating cutting depth

Instead of working with constant cutting depth D, you can use an alternating cutting depth to vary the load on the tool edge. As a consequence you can increase the tool life.



① First cut

② Second cut

Figure 10-12 Alternating cutting depth

The percentage for the alternating cutting depth is saved in a machine data element.

Machine manufacturer

Please observe the information provided by the machine manufacturer.

Cut segmentation

To avoid the occurrence of very thin cuts in cut segmentation due to contour edges, you can align the cut segmentation to the contour edges. During machining the contour is then divided by the edges into individual sections and cut segmentation is performed separately for each section.

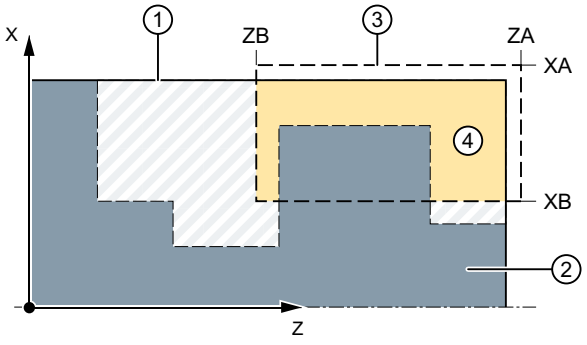
Set machining area limits

If, for example, you want to machine a certain area of the contour with a different tool, you can set machining area limits so that machining only takes place in the area of the contour you have selected. You can define between 1 and 4 limit lines.

The limit lines must not intersect the contour on the side facing the machining.

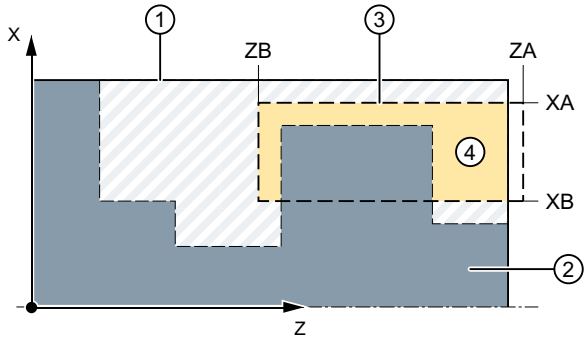
This limit has the same effect during roughing and finishing.

Example of the limit in longitudinal external machining



① Blank
② Finished part
③ Limit
④ Machining

Figure 10-13 Permitted limit: Limit line XA is outside the contour of the blank



① Blank
② Finished part
③ Limit
④ Machining

Figure 10-14 Impermissible limit: Limit line XA is inside the contour of the blank

Feedrate interruption

To prevent the occurrence of excessively long chips during machining, you can program a feedrate interruption. Parameter DI specifies the distance after which the feedrate interruption should occur.
The interruption time or retraction distance is defined in machine data.

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

Residual material machining / naming conventions

G code program

For multi-channel systems, cycles attach a "_C" and a two-digit, channel-specific number to the names of the programs to be generated, e.g. for channel 1 "_C01".

This is the reason why the name of the main program must not end with "_C" and a two-digit number. This is monitored by the cycles.

For programs with residual machining, when specifying the name for the file, which includes the updated blank contour, it must be ensured that this does not have the attached characters ("_C" and double-digit number).

For single-channel systems, cycles do not extend the name of the programs to be generated.

Note

G code program

For G code programs, the programs to be generated, which do not include any path data, are saved in the directory in which the main program is located. In this case, it must be ensured that programs, which already exist in the directory and which have the same name as the programs to be generated, are overwritten.

Input simple

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

Machine manufacturer

Various defined values can be pre-assigned using setting data.

Please observe the information provided by the machine manufacturer.

If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

Machining type

You can freely select the machining type (roughing, finishing or complete machining - roughing + finishing). During contour roughing, parallel cuts of maximum programmed infeed depth are created. Roughing is performed to the programmed allowance.

You can also specify a compensation allowance U1 for finishing operations, which allows you to either finish several times (positive allowance) or to shrink the contour (negative allowance). Finishing is performed in the same direction as roughing.

Procedure

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

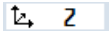
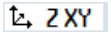
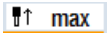
2. Press the "Turning" and "Contour turning" softkeys.

3. Press the "Stock removal" softkey.

   The "Stock Removal" input window opens.

Parameters in the "Input complete" mode

G code program parameters				ShopMill program parameters		
Input	• Complete					
PRG	• Name of the program to be generated • Automatic Automatic generation of program names			T	Tool name	
PL	Machining plane			D	Cutting edge number	
RP	Retraction plane – (only for machining direction, longitudinal, inner)	mm		F	Feedrate	mm/min mm/rev
SC	Safety clearance	mm		S / V	Spindle speed or constant cutting rate	rpm m/min
F	Feedrate	*				
Residual material	With subsequent residual material removal. • Yes • No					
CONR	Name to save the updated unmachined-part contour for residual material removal - (only "Yes" for residual material removal).					

Parameter	Description	Unit
TC	Name of the swivel data set Note: The selection box only appears if more than one swivel data set has been set up.	
Retraction (for ShopMill program)	• No The axis is not retracted before swiveling. • Z  Retraction in the direction of machine axis Z. • Z,X,Y  Move machining axes to retraction position before swiveling. • Tool direction, max.  Maximum retraction (up to the software end position) in the tool direction. • Tool direction, inc Incremental retraction (specified retraction distance, up to the software end position) in the tool direction. When retracting in the tool direction, in the swiveled machine state, several axes can move (traverse).	

Parameter	Description	Unit
ZR (for ShopMill program)	Retraction path - only for incremental retraction in	mm
Align tool through beta and gamma angles		
β (for ShopMill program) 	Align tool with swivel axes • Input value The required angle can be freely entered. • β = 0° • β = 90° 	Degrees
γ (for ShopMill program) 	• 0° • 180° • The required angle can be freely entered	Degrees
Directly position rotary axes		
B1 (for ShopMill program)	Directly align the tool with the swiveling axes: The required angle can be freely entered.	Degrees
C1 (for ShopMill program)	The required angle can be freely entered.	Degrees
αC (for ShopMill program)	Rotational position for a pole position.	Degrees
Hirth joint 	• Round to the next Hirth joint • Round up to Hirth joint • Round down to Hirth joint Note: For machines with a Hirth joint	
Tool 	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.	

Parameter	Description		Unit
Preferred direction (for ShopMill program)	Preferred direction of the swivel axis for several possible alignments of the machine. + –		
Machining	• ∇ (roughing) • ∇∇∇ (finishing) • ∇+∇∇∇ (complete machining)		
Machining direction	• Face • Longitudinal • Parallel to the contour	• From inside to outside • From outside to inside • From end face to rear side • From rear side to end face	
	The machining direction depends on the stock removal direction and choice of tool.		
Position	• Front • rear • Inside • Outside		
D	Maximum depth infeed - (only for ∇)		mm
DX	Maximum depth infeed - (only for parallel to the contour, as an alternative to D).		mm
	Do not round contour at end of cut. Always round contour at end of cut.		
	Uniform cut segmentation Round cut segmentation at the edge		
	Constant cutting depth Alternating cutting depth - (only with align cut segmentation to edge)		
DZ	Maximum depth infeed - (only for position parallel to the contour and UX)		mm

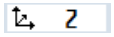
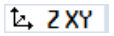
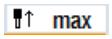
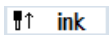
Parameter	Description	Unit
UX or U	Finishing allowance in X or finishing allowance in X and Z – (only for ∇)	mm
UZ	Finishing allowance in Z – (only for UX)	mm
DI	For zero: Continuous cut - (only for ∇)	mm
BL	Blank description (only for ∇) • Cylinder (described using XD, ZD) • Allowance (XD and ZD on the finished part contour) • Contour (additional CYCLE62 call with blank contour – e.g. cast iron mold)	
XD	- (only for ∇ machining) - (only for blank description, cylinder and allowance) • For blank description, cylinder – Version, absolute: Cylinder dimension ∅ (abs) – Version incremental: Allowance (inc) to maximum values of the CYCLE62 finished part contour • For blank description, allowance – Allowance on the CYCLE62 finished part contour (inc)	mm
ZD	- (only for ∇ machining) - (only for blank description, cylinder and allowance) • For blank description, cylinder – Version, absolute: Cylinder dimension (abs) – Version incremental: Allowance (inc) to maximum values of the CYCLE62 finished part contour • For blank description, allowance – Allowance on the CYCLE62 finished part contour (inc)	mm
Allowance	Allowance for pre-finishing - (only for ∇∇∇) • Yes U1 contour allowance • No	
U1	Compensation allowance in X and Z direction (inc) – (only for allowance) • Positive value: Compensation allowance is retained • Negative value: Compensation allowance is removed in addition to finishing allowance	mm
Set machining area limits	Set machining area limits • Yes • No	
XA XB ZA ZB	With limited machining area only, yes: 1. Limit XA ∅ 2. Limit XB ∅ (abs) or 2nd limit referred to XA (inc) 1. Limit ZA 2. Limit ZB (abs) or 2nd limit referred to ZA (inc)	mm

Parameter	Description	Unit
Relief cuts	Machine relief cuts • Yes • No	
FR	Insertion feedrate, relief cuts	*

* Unit of feedrate as programmed before the cycle call

Parameters in the "Input simple" mode

G code program parameters				ShopMill program parameters		
Input	• simple					
PRG	• Name of the program to be generated • Automatic Automatic generation of program names			T	Tool name	
F	Feedrate	*		D	Cutting edge number	
				F	Feedrate	mm/min mm/rev
				S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
TC 	Name of the swivel data set Note: The selection box only appears if more than one swivel data set has been set up.	
Retraction (for ShopMill program) 	• No The axis is not retracted before swiveling • Z  Retraction in the direction of machine axis Z • Z,X,Y  Move machining axes to retraction position before swiveling • Tool direction, max.  Maximum retraction (up to the software end position) in the tool direction • Tool direction, inc  Incremental retraction (specified retraction distance, up to the software end position) in the tool direction When retracting in the tool direction, in the swiveled machine state, several axes can move (traverse).	
ZR (for ShopMill program)	Retraction path - only for incremental retraction in	mm
Align tool through beta and gamma angles		
β (for ShopMill program) 	Align tool with swivel axes • Input value The required angle can be freely entered • β = 0° • β = 90° 	Degrees
γ (for ShopMill program) 	• 0° • 180° • The required angle can be freely entered	Degrees
Directly position rotary axes		
B1 (for ShopMill program)	Directly align the tool with the swiveling axes: The required angle can be freely entered	Degrees

Parameter	Description	Unit
C1 (for ShopMill program)	The required angle can be freely entered	Degrees
αC (for ShopMill program)	Rotational position for a pole position	Degrees
Hirth joint 	• Round to the next Hirth joint • Round up to Hirth joint • Round down to Hirth joint Note: For machines with a Hirth joint	
Tool 	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.	
Preferred direction (for ShopMill program) 	Preferred direction of the swivel axis for several possible alignments of the machine 	
Machining 	• ∇ (roughing) • ∇∇∇ (finishing) • ∇+∇∇∇ (complete machining)	

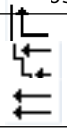
Parameter	Description		Unit
Machining direction ↻	• face ↻ • longitudinal ↻ • parallel to the contour ↻	• From inside to outside • From outside to inside • From end face to rear side • From rear side to end face 	
	The machining direction depends on the stock removal direction and choice of tool.		
RP	Retraction plane – (only for machining direction, longitudinal, inner)		mm
Position ↻	• front • rear • inside • outside		
D	Maximum depth infeed - (only for ∇)		mm
DX	Maximum depth infeed - (only for parallel to the contour, as an alternative to D)		mm
DZ	Maximum depth infeed - (only for position parallel to the contour and UX)		mm
UX or U ↻	Finishing allowance in X or finishing allowance in X and Z – (only for ∇)		mm
UZ	Finishing allowance in Z – (only for UX)		mm
DI	For zero: Continuous cut - (only for ∇)		mm
BL ↻	Blank description (only for ∇) • Cylinder (described using XD, ZD) • Allowance (XD and ZD on the finished part contour) • Contour (additional CYCLE62 call with blank contour – e.g. cast iron mold)		
XD	- (only for ∇ machining) - (only for blank description, cylinder and allowance) • For blank description, cylinder – Version, absolute: Cylinder dimension ⌀ (abs) – Version incremental: Allowance (inc) to maximum values of the CYCLE62 finished part contour • For blank description, allowance – Allowance on the CYCLE62 finished part contour (inc)		mm

Parameter	Description	Unit
ZD	- (only for ∇ machining) - (only for blank description, cylinder and allowance) • For blank description, cylinder – Version, absolute: Cylinder dimension (abs) – Version incremental: Allowance (inc) to maximum values of the CYCLE62 finished part contour • For blank description, allowance – Allowance on the CYCLE62 finished part contour (inc)	mm
Allowance	Allowance for pre-finishing - (only for ∇∇∇) • Yes U1 contour allowance • No	
U1	Compensation allowance in X and Z direction (inc) – (only for allowance) • Positive value: Compensation allowance is retained • Negative value: Compensation allowance is removed in addition to finishing allowance	mm
Relief cuts	• Yes (cannot be changed)	
FR	Insertion feedrate, relief cuts	*

* Unit of feedrate as programmed before the cycle call

Hidden parameters

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.

Parameter	Description	Value	Can be set in SD
Residual material	With subsequent residual material removal	No	
SC	Safety clearance		x
PL	Machining plane	Defined in MD 52005	
Selection	Always round on the contour Uniform cut segmentation Constant cutting depth		
DI	Continuous cut - (only for ∇)	0	
Set machining area limits	Set machining area limits	No	
Relief cuts	Machine relief cuts (grayed out)	Yes	

Machine manufacturer

Please observe the information provided by the machine manufacturer.

10.6.8 Stock removal residual (CYCLE952)

Function

Using the "Stock removal residual" function, you can remove material that has remained for stock removal along the contour.

During stock removal along the contour, the cycle automatically detects any residual material and generates an updated blank contour. For a G code program, for stock removal residual material, "Yes" must be programmed. Material that remains as part of the finishing allowance is not residual material. Using the "Stock removal residual material" function, you can remove unwanted material with a suitable tool.

The "Stock removal residual material" function is a software option.

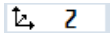
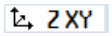
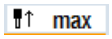
Procedure

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

2. Press the "Turning" and "Contour turning" softkeys.

3. Press the "Stock removal residual material" softkey.
   The "Stock removal residual material" input window opens.

G code program parameters				ShopMill program parameters		
PRG U	• Name of the program to be generated • Automatic Automatic generation of program names			T	Tool name	
PL U	Machining plane			D	Cutting edge number	
RP	Retraction plane – (only for machining direction, longitudinal, inner)	mm		F U	Feedrate	mm/min mm/rev
SC	Safety clearance	mm		S / V U	Spindle speed or constant cutting rate	rpm m/min
F	Feedrate	*				
CON	Name of the updated unmachined-part contour for residual material machining (without the attached character "_C" and double-digit number)					
Residual material U	With subsequent residual material removal. • Yes • No					
CONR	Name to save the updated unmachined-part contour for residual material removal - (only "Yes" for residual material removal).					

Parameter	Description	Unit
TC 	Name of the swivel data set Note: The selection box only appears if more than one swivel data set has been set up.	
Retraction (for ShopMill program) 	• No The axis is not retracted before swiveling. • Z  Retraction in the direction of machine axis Z. • Z,X,Y  Move machining axes to retraction position before swiveling. • Tool direction, max.  Maximum retraction (up to the software end position) in the tool direction. • Tool direction, inc Incremental retraction (specified retraction distance, up to the software end position) in the tool direction. When retracting in the tool direction, in the swiveled machine state, several axes can move (traverse).	
ZR (for ShopMill program)	Retraction path - only for incremental retraction in	mm
Align tool through beta and gamma angles		
β (for ShopMill program) 	Align tool with swivel axes • Input value The required angle can be freely entered. • β = 0° • β = 90° 	Degrees
γ (for ShopMill program) 	• 0° • 180° • The required angle can be freely entered	Degrees
Directly position rotary axes		
B1 (for ShopMill program)	Directly align the tool with the swiveling axes: The required angle can be freely entered.	Degrees

Parameter	Description	Unit
C1 (for ShopMill program)	The required angle can be freely entered.	Degrees
αC (for ShopMill program)	Rotational position for a pole position.	Degrees
Hirth joint 	• Round to the next Hirth joint • Round up to Hirth joint • Round down to Hirth joint Note: For machines with a Hirth joint	
Tool 	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.	
Preferred direction (for ShopMill program) 	Preferred direction of the swivel axis for several possible alignments of the machine. 	
Machining 	• ∇ (roughing) • ∇∇∇ (finishing)	
Machining direction 	• Face • Longitudinal • Parallel to the contour • From inside to outside • From outside to inside • From end face to rear side • From rear side to end face	
	The machining direction depends on the stock removal direction and choice of tool.	
Position 	• Front • rear • Inside • Outside	
D	Maximum depth infeed - (only for ∇)	mm
XDA	First grooving limit tool (abs) – (only for face machining direction).	mm
XDB	Second grooving limit tool (abs) – (only for face machining direction).	mm

Parameter	Description	Unit
DX	Maximum depth infeed - (only for parallel to the contour, as an alternative to D).	mm
↻	Do not round contour at end of cut. Always round contour at end of cut.	
↻	Uniform cut segmentation Round cut segmentation at the edge	
↻	only for align cut segmentation at the edge: Constant cutting depth Alternating cutting depth	
Allowance ↻	Allowance for pre-finishing - (only for ∇∇∇) • Yes U1 contour allowance • No	s
U1	Compensation allowance in X and Z direction (inc) – (only for allowance) • Positive value: Compensation allowance is retained • Negative value: Compensation allowance is removed in addition to finishing allowance	mm
Set machining area limits ↻	Set machining area limits • Yes • No	
XA XB ↻ ZA ZB ↻	With limited machining area only, yes: 1. Limit XA ∅ 2. Limit XB ∅ (abs) or 2nd limit referred to XA (inc) 1. Limit ZA 2. Limit ZB (abs) or 2nd limit referred to ZA (inc)	mm
Relief cuts ↻	Machine relief cuts • Yes • No	
FR	Insertion feedrate, relief cuts	*

* Unit of feedrate as programmed before the cycle call

10.6.9 Grooving (CYCLE952)

Function

The "Grooving" function is used to machine grooves of any shape.

Before you program the groove, you must define the groove contour.

If a groove is wider than the active tool, it is machined in several cuts. The tool is moved by a maximum of 80% of the tool width for each groove.

Blank

When grooving, the cycle takes into account a blank that can consist of a cylinder, an allowance on the finished part contour or any other blank contour.

Note

In order to avoid collisions between tools and workpieces due to positioning motions, the programmed blank contour must match the real blank.

Requirement

For a G code program, at least one CYCLE62 is required before CYCLE952.

If CYCLE62 is only present once, then this involves the finished part contour.

If CYCLE62 is present twice, then the first call is the blank contour and the second call is the finished part contour (also see Section "Programming (Page 592)").

Note

Execution from external media

If you want to execute programs from an external drive (e.g. local drive or network drive), you require the "Execution from external storage (EES)" function.

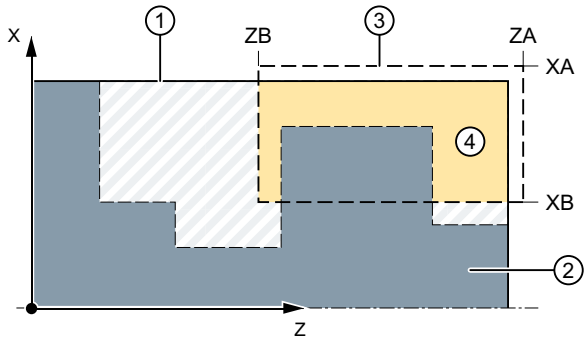
Set machining area limits

If, for example, you want to machine a certain area of the contour with a different tool, you can set machining area limits so that machining only takes place in the area of the contour you have selected.

The limit lines must not intersect the contour on the side facing the machining.

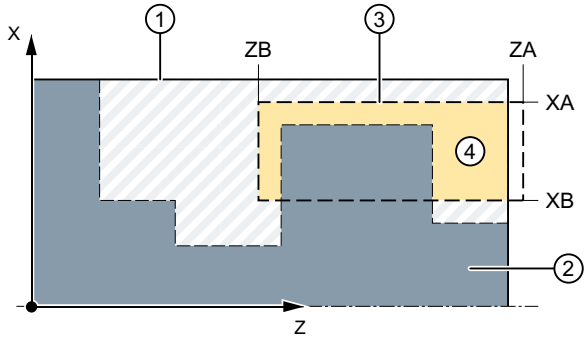
This limit has the same effect during roughing and finishing.

Example of the limit in longitudinal external machining



① Blank
② Finished part
③ Limit
④ Machining

Figure 10-15 Permitted limit: Limit line XA is outside the contour of the blank



① Blank
② Finished part
③ Limit
④ Machining

Figure 10-16 Impermissible limit: Limit line XA is inside the contour of the blank

Feedrate interruption

To prevent the occurrence of excessively long chips during machining, you can program a feedrate interruption.

Input simple

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

Machine manufacturer

Various defined values can be pre-assigned using setting data.

Please observe the information provided by the machine manufacturer.

If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

Machining type

You can freely select the machining type (roughing, finishing or complete machining).

For more detailed information, please refer to section "Stock removal".

Procedure

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

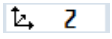
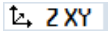
2. Press the "Turning" and "Contour turning" softkeys.

3. Press the "Grooving" softkey.
   The "Grooving" input window opens.

Parameters in the "Input complete" mode

G code program parameters				ShopMill program parameters		
Input	• Complete					
PRG ↻	• Name of the program to be generated • Automatic Automatic generation of program names			T	Tool name	
PL ↻	Machining plane			D	Cutting edge number	
RP	Retraction plane – (only for machining direction, longi-tudinal, inner)	mm		F ↻	Feedrate	mm/min mm/rev
SC	Safety clearance	mm		S / V ↻	Spindle speed or constant cutting rate	rpm m/min
F	Feedrate	*				

G code program parameters			ShopMill program parameters		
Residual material U	With subsequent residual material removal. • Yes • No				
CONR	Name to save the updated unmachined-part contour for residual material removal - (only "Yes" for residual material removal).				

Parameter	Description	Unit
TC U	Name of the swivel data set Note: The selection box only appears if more than one swivel data set has been set up.	
Retraction (for ShopMill program) U	• No The axis is not retracted before swiveling. • Z  Retraction in the direction of machine axis Z. • Z,X,Y  Move machining axes to retraction position before swiveling. • Tool direction, max. Maximum retraction (up to the software end position) in the tool direction. • Tool direction, inc Incremental retraction (specified retraction distance, up to the software end position) in the tool direction. When retracting in the tool direction, in the swiveled machine state, several axes can move (traverse).	
ZR (for ShopMill program)	Retraction path - only for incremental retraction in	mm
Align tool through beta and gamma angles		
β (for ShopMill program) U	Align tool with swivel axes • Input value The required angle can be freely entered. • β = 0° • β = 90° 	Degrees

Parameter	Description	Unit
γ (for ShopMill program) 	• 0° • 180° • The required angle can be freely entered	Degrees
Directly position rotary axes		
B1 (for ShopMill program)	Directly align the tool with the swiveling axes: The required angle can be freely entered.	Degrees
C1 (for ShopMill program)	The required angle can be freely entered.	Degrees
αC (for ShopMill program)	Rotational position for a pole position.	Degrees
Hirth joint 	• Round to the next Hirth joint • Round up to Hirth joint • Round down to Hirth joint Note: For machines with a Hirth joint	
Tool 	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.	
Preferred direction (for ShopMill program) 	Preferred direction of the swivel axis for several possible alignments of the machine. 	
Machining 	• ∇ (roughing) • ∇∇∇ (finishing) • ∇+∇∇∇ (complete machining)	
Machining direction 	• Face • Longitudinal	

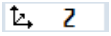
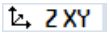
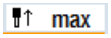
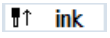
Parameter	Description	Unit
Position ○	• Front • rear • Inside • Outside	
D	Maximum depth infeed - (only for ∇)	mm
XDA	First grooving limit tool (abs) – (only for face machining direction).	mm
XDB	Second grooving limit tool (abs) – (only for face machining direction).	mm
UX or U ○	Finishing allowance in X or finishing allowance in X and Z – (only for ∇)	mm
UZ	Finishing allowance in Z – (only for UX)	mm
DI	For zero: Continuous cut - (only for ∇)	mm
BL ○	Blank description (only for ∇) • Cylinder (described using XD, ZD) • Allowance (XD and ZD on the finished part contour) • Contour (additional CYCLE62 call with blank contour – e.g. cast iron mold)	
XD	- (only for ∇ machining) - (only for blank description, cylinder and allowance) • For blank description, cylinder – Version, absolute: Cylinder dimension ∅ (abs) – Version incremental: Allowance (inc) to maximum values of the CYCLE62 finished part contour • For blank description, allowance – Allowance on the CYCLE62 finished part contour (inc)	mm
ZD	- (only for ∇ machining) - (only for blank description, cylinder and allowance) • For blank description, cylinder – Version, absolute: Cylinder dimension (abs) – Version incremental: Allowance (inc) to maximum values of the CYCLE62 finished part contour • For blank description, allowance – Allowance on the CYCLE62 finished part contour (inc)	mm
Allowance ○	Allowance for pre-finishing - (only for ∇∇∇) • Yes U1 contour allowance • No	mm
U1	Compensation allowance in X and Z direction (inc) – (only for allowance) • Positive value: Compensation allowance is retained • Negative value: Compensation allowance is removed in addition to finishing allowance	mm
Set machining area limits ○	Set machining area limits • Yes • No	

Parameter	Description	Unit
	With limited machining area only, yes:	mm
XA	1. Limit XA ∅	
XB	2. Limit XB ∅ (abs) or 2nd limit referred to XA (inc)	
ZA	1. Limit ZA	
ZB	2. Limit ZB (abs) or 2nd limit referred to ZA (inc)	
N	Number of grooves	
DP	Distance between grooves (inc)	mm

* Unit of feedrate as programmed before the cycle call

Parameters in the "Input simple" mode

G code program parameters				ShopMill program parameters		
Input	• simple					
PRG 	• Name of the program to be generated • Automatic Automatic generation of program names			T	Tool name	
RP	Retraction plane – (only for machining direction, longi-tudinal, inner)	mm		D	Cutting edge number	
F	Feedrate	*		F 	Feedrate	mm/min mm/rev
				S / V 	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
TC 	Name of the swivel data set Note: The selection box only appears if more than one swivel data set has been set up.	
Retraction (for ShopMill program) 	• No The axis is not retracted before swiveling • Z  Retraction in the direction of machine axis Z • Z,X,Y  Move machining axes to retraction position before swiveling • Tool direction, max.  Maximum retraction (up to the software end position) in the tool direction • Tool direction, inc  Incremental retraction (specified retraction distance, up to the software end position) in the tool direction When retracting in the tool direction, in the swiveled machine state, several axes can move (traverse).	
ZR (for ShopMill program)	Retraction path - only for incremental retraction in	mm
Align tool through beta and gamma angles		
β (for ShopMill program) 	Align tool with swivel axes • Input value The required angle can be freely entered • β = 0° • β = 90° 	Degrees
γ (for ShopMill program) 	• 0° • 180° • The required angle can be freely entered	Degrees
Directly position rotary axes		
B1 (for ShopMill program)	Directly align the tool with the swiveling axes: The required angle can be freely entered	Degrees

Parameter	Description	Unit
C1 (for ShopMill program)	The required angle can be freely entered	Degrees
αC (for ShopMill program)	Rotational position for a pole position	Degrees
Hirth joint 	• Round to the next Hirth joint • Round up to Hirth joint • Round down to Hirth joint Note: For machines with a Hirth joint	
Tool 	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.	
Preferred direction (for ShopMill program) 	Preferred direction of the swivel axis for several possible alignments of the machine 	
Machining 	• ∇ (roughing) • ∇∇∇ (finishing) • ∇+∇∇∇ (complete machining)	
Machining direction 	• Face • Longitudinal	
Position 	• front • rear • inside • outside	
D	Maximum depth infeed - (only for ∇)	mm
XDA	First grooving limit tool (abs) – (only for face machining direction)	mm
XDB	Second grooving limit tool (abs) – (only for face machining direction)	mm
UX or U	Finishing allowance in X or finishing allowance in X and Z – (only for ∇)	mm

Parameter	Description	Unit
UZ	Finishing allowance in Z – (only for UX)	mm
BL	Blank description (only for ∇) • Cylinder (described using XD, ZD) • Allowance (XD and ZD on the finished part contour) • Contour (additional CYCLE62 call with blank contour – e.g. cast iron mold)	
XD	- (only for ∇ machining) - (only for blank description, cylinder and allowance) • For blank description, cylinder – Version, absolute: Cylinder dimension ∅ (abs) – Version incremental: Allowance (inc) to maximum values of the CYCLE62 finished part contour • For blank description, allowance – Allowance on the CYCLE62 finished part contour (inc)	mm
ZD	- (only for ∇ machining) - (only for blank description, cylinder and allowance) • For blank description, cylinder – Version, absolute: Cylinder dimension (abs) – Version incremental: Allowance (inc) to maximum values of the CYCLE62 finished part contour • For blank description, allowance – Allowance on the CYCLE62 finished part contour (inc)	mm
Allowance	Allowance for pre-finishing - (only for ∇∇∇) • Yes U1 contour allowance • No	mm
U1	Compensation allowance in X and Z direction (inc) – (only for allowance) • Positive value: Compensation allowance is retained • Negative value: Compensation allowance is removed in addition to finishing allowance	mm

* Unit of feedrate as programmed before the cycle call

Hidden parameters

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.

Parameter	Description	Value	Can be set in SD
Residual material	Without subsequent residual material removal	No	
PL	Machining plane	Defined in MD 52005	
SC	Safety clearance		x

Parameter	Description	Value	Can be set in SD
DI	Continuous cut - (only for ∇)	0	
Set machining area limits	Set machining area limits	No	
N	Number of grooves	1	

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

10.6.10 Grooving residual material (CYCLE952)

Function

The "Grooving residual material" function is used when you want to machine the material that remained after grooving along the contour.

For a G code program, first select the "Grooving residual material" function. Material that remains as part of the finishing allowance is not residual material. The "Grooving residual material" function allows you to remove unwanted material with a suitable tool.

The "Grooving residual material" function is a software option.

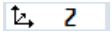
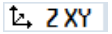
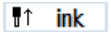
Procedure

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

2. Press the "Turning" and "Contour turning" softkeys.

3. Press the "Grooving residual material" softkey.
   The "Grooving residual material" input window is opened.

G code program parameters				**ShopMill program parameters**		
PRG ○	• Name of the program to be generated • Automatic Automatic generation of program names			T	Tool name	
PL ○	Machining plane			D	Cutting edge number	
RP	Retraction plane – (only for machining direction, longitudinal, inner)	mm		F ○	Feedrate	mm/min mm/rev
SC	Safety clearance	mm		S / V ○	Spindle speed or constant cutting rate	rpm m/min
F	Feedrate	*				
CON	Name of the updated unmachined-part contour for residual material machining (without the attached character "_C" and double-digit number)					
Residual material ○	With subsequent residual material removal. • Yes • No					
CONR	Name to save the updated unmachined-part contour for residual material removal - (only "Yes" for residual material removal).					

Parameter	Description	Unit
Retraction (for ShopMill program) 	• No The axis is not retracted before swiveling. • Z  Retraction in the direction of machine axis Z. • Z,X,Y  Move machining axes to retraction position before swiveling. • Tool direction, max. Maximum retraction (up to the software end position) in the tool direction. • Tool direction, inc  Incremental retraction (specified retraction distance, up to the software end position) in the tool direction. When retracting in the tool direction, in the swiveled machine state, several axes can move (traverse).	
ZR (for ShopMill program)	Retraction path - only for incremental retraction in	mm
Align tool through beta and gamma angles		
β (for ShopMill program) 	Align tool with swivel axes • Input value The required angle can be freely entered. • β = 0° • β = 90° 	Degrees
γ (for ShopMill program) 	• 0° • 180° • The required angle can be freely entered	Degrees
Directly position rotary axes		
B1 (for ShopMill program)	Directly align the tool with the swiveling axes: The required angle can be freely entered.	Degrees
C1 (for ShopMill program)	The required angle can be freely entered.	Degrees

Parameter	Description	Unit
αC (for ShopMill program)	Rotational position for a pole position.	Degrees
Hirth joint	• Round to the next Hirth joint • Round up to Hirth joint • Round down to Hirth joint Note: For machines with a Hirth joint	
Tool	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.	
Preferred direction (for ShopMill program)	Preferred direction of the swivel axis for several possible alignments of the machine. 	
Machining	• ∇ (roughing) • ∇∇∇ (finishing)	
Machining direction	• Face • Longitudinal	
Position	• Front • rear • Inside • Outside	
D	Maximum depth infeed - (only for ∇)	mm
XDA	First grooving limit tool (abs) – (only for face machining direction).	mm
XDB	Second grooving limit tool (abs) – (only for face machining direction).	mm
UX or U	Finishing allowance in X or finishing allowance in X and Z – (only for ∇)	mm
UZ	Finishing allowance in Z – (only for UX)	mm
DI	For zero: Continuous cut - (only for ∇)	mm

Parameter	Description	Unit
Allowance	Allowance for pre-finishing - (only for ∇∇∇) • Yes U1 contour allowance • No	mm
U1	Compensation allowance in X and Z direction (inc) – (only for allowance) • Positive value: Compensation allowance is retained • Negative value: Compensation allowance is removed in addition to finishing allowance	mm
Set machining area limits	Set machining area limits • Yes • No	
XA XB ZA ZB	With limited machining area only, yes: 1. Limit XA ∅ 2. Limit XB ∅ (abs) or 2nd limit referred to XA (inc) 1. Limit ZA 2. Limit ZB (abs) or 2nd limit referred to ZA (inc)	mm
N	Number of grooves	
DP	Distance between grooves (inc)	mm

* Unit of feedrate as programmed before the cycle call

10.6.11 Plunge turning (CYCLE952)

Function

Using the "Plunge turning" function, you can machine any shape of groove.

Contrary to grooving, the plunge turning function removes material on the sides after the groove has been machined in order to reduce machining time. Contrary to stock removal, the plunge turning function allows you to machine contours that the tool must enter vertically.

You will need a special tool for plunge turning. Before you program the "Plunge turning" cycle, you must define the contour.

Requirement

For a G code program, at least one CYCLE62 is required before CYCLE952.

If CYCLE62 is only present once, then this involves the finished part contour.

If CYCLE62 is present twice, then the first call is the blank contour and the second call is the finished part contour (also see Section "Programming (Page 592)").

Note

Execution from external media

If you want to execute programs from an external drive (e.g. local drive or network drive), you require the Execution from external storage (EES) function.

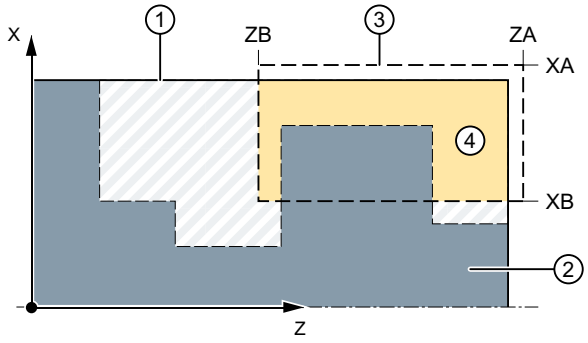
Set machining area limits

If, for example, you want to machine a certain area of the contour with a different tool, you can set machining area limits so that machining only takes place in the area of the contour you have selected.

The limit lines must not intersect the contour on the side facing the machining.

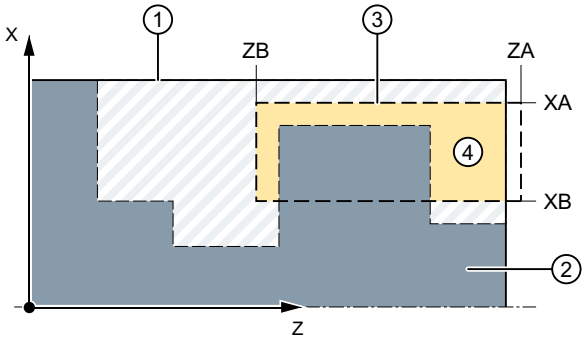
This limit has the same effect during roughing and finishing.

Example of the limit in longitudinal external machining



① Blank
② Finished part
③ Limit
④ Machining

Figure 10-17 Permitted limit: Limit line XA is outside the contour of the blank



① Blank
② Finished part
③ Limit
④ Machining

Figure 10-18 Impermissible limit: Limit line XA is inside the contour of the blank

Feedrate interruption

To prevent the occurrence of excessively long chips during machining, you can program a feedrate interruption.

Input simple

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

Machine manufacturer

Various defined values can be pre-assigned using setting data.

Please refer to the machine manufacturer's specifications.

If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

Machining type

You can freely select the machining type (roughing, finishing or complete machining).

For more detailed information, please refer to section "Stock removal".

Procedure

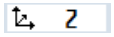
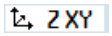
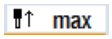
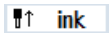
1. The part program to be executed has been created and you are in the editor.

2. Press the "Turning" and "Contour turning" softkeys.

3. Press the "Plunge turning" softkey.
   The "Plunge turning" input window opens.

Parameters in the "Input complete" mode

G code program parameters				ShopMill program parameters		
Input	• Complete					
PRG	• Name of the program to be generated • Automatic Automatic generation of program names			T	Tool name	
PL	Machining plane			D	Cutting edge number	
RP	Retraction plane – (only for machining direction, longitudinal, inner)	mm		S / V	Spindle speed or constant cutting rate	rpm m/min
SC	Safety clearance	mm				
Residual material	With subsequent residual material removal. • Yes • No					
CONR	Name to save the updated unmachined-part contour for residual material removal - (only "Yes" for residual material removal).					

Parameter	Description	Unit
TC 	Name of the swivel data set Note: The selection box only appears if more than one swivel data set has been set up.	
Retraction (for ShopMill program) 	• No The axis is not retracted before swiveling. • Z  Retraction in the direction of machine axis Z. • Z,X,Y  Move machining axes to retraction position before swiveling. • Tool direction, max.  Maximum retraction (up to the software end position) in the tool direction. • Tool direction, inc  Incremental retraction (specified retraction distance, up to the software end position) in the tool direction. When retracting in the tool direction, in the swiveled machine state, several axes can move (traverse).	
ZR (for ShopMill program)	Retraction path - only for incremental retraction in	mm
Align tool through beta and gamma angles		
β (for ShopMill program) 	Align tool with swivel axes • Input value The required angle can be freely entered. • β = 0° • β = 90° 	Degrees
γ (for ShopMill program) 	• 0° • 180° • The required angle can be freely entered	Degrees
Directly position rotary axes		
B1 (for ShopMill program)	Directly align the tool with the swiveling axes: The required angle can be freely entered.	Degrees
C1 (for ShopMill program)	The required angle can be freely entered.	Degrees

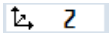
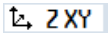
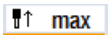
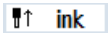
Parameter	Description	Unit
αC (for ShopMill program)	Rotational position for a pole position.	Degrees
Hirth joint U	• Round to the next Hirth joint • Round up to Hirth joint • Round down to Hirth joint Note: For machines with a Hirth joint	
Tool U	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.	
Preferred direction (for ShopMill program) U	Preferred direction of the swivel axis for several possible alignments of the machine. 	
FX	Feedrate in X direction	*
FZ	Feedrate in Z direction	*
Machining U	• ∇ (roughing) • ∇∇∇ (finishing) • ∇+∇∇∇ (complete machining)	
Machining direction U	• Face • Longitudinal	
Position U	• Front • rear • Inside • Outside	
D	Maximum depth infeed - (only for ∇)	mm
XDA	First grooving limit tool (abs) – (only for face machining direction).	mm
XDB	Second grooving limit tool (abs) – (only for face machining direction).	mm
UX or U U	Finishing allowance in X or finishing allowance in X and Z – (only for ∇)	mm
UZ	Finishing allowance in Z – (only for UX)	mm

Parameter	Description	Unit
DI	For zero: Continuous cut - (only for ∇)	mm
BL	Blank description (only for ∇) • Cylinder (described using XD, ZD) • Allowance (XD and ZD on the finished part contour) • Contour (additional CYCLE62 call with blank contour – e.g. cast iron mold)	
XD	- (only for ∇ machining) - (only for blank description, cylinder and allowance) • For blank description, cylinder – Version, absolute: Cylinder dimension ∅ (abs) – Version incremental: Allowance (inc) to maximum values of the CYCLE62 finished part contour • For blank description, allowance – Allowance on the CYCLE62 finished part contour (inc)	mm
ZD	- (only for ∇ machining) - (only for blank description, cylinder and allowance) • For blank description, cylinder – Version, absolute: Cylinder dimension (abs) – Version incremental: Allowance (inc) to maximum values of the CYCLE62 finished part contour • For blank description, allowance – Allowance on the CYCLE62 finished part contour (inc)	mm
Allowance	Allowance for pre-finishing - (only for ∇∇∇) • Yes U1 contour allowance • No	mm
U1	Compensation allowance in X and Z direction (inc) – (only for allowance) • Positive value: Compensation allowance is retained • Negative value: Compensation allowance is removed in addition to finishing allowance	mm
Set machining area limits	Set machining area limits • Yes • No	
XA XB ZA ZB	With limited machining area only, yes: 1st limit XA ∅ 2nd limit XB ∅ (abs) or 2nd limit referred to XA (inc) 1st limit ZA 2nd limit ZB (abs) or 2nd limit referred to ZA (inc)	mm
N	Number of grooves	
DP	Distance between grooves	mm

* Unit of feedrate as programmed before the cycle call

Parameters in the "Input simple" mode

G code program parameters				ShopMill program parameters		
Input	• simple					
PRG ↻	• Name of the program to be generated • Automatic Automatic generation of program names			T	Tool name	
				D	Cutting edge number	
				S / V ↻	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
TC ↻	Name of the swivel data set Note: The selection box only appears if more than one swivel data set has been set up.	
Retraction (for ShopMill program) ↻	• No The axis is not retracted before swiveling • Z  Retraction in the direction of machine axis Z • Z,X,Y  Move machining axes to retraction position before swiveling • Tool direction, max.  Maximum retraction (up to the software end position) in the tool direction • Tool direction, inc  Incremental retraction (specified retraction distance, up to the software end position) in the tool direction When retracting in the tool direction, in the swiveled machine state, several axes can move (traverse).	
ZR (for ShopMill program)	Retraction path - only for incremental retraction in	mm
Align tool through beta and gamma angles		

Parameter	Description	Unit
β (for ShopMill program) 	Align tool with swivel axes • Input value The required angle can be freely entered • β = 0° • β = 90° 	Degrees
γ (for ShopMill program) 	• 0° • 180° • The required angle can be freely entered	Degrees
Directly position rotary axes		
B1 (for ShopMill program)	Directly align the tool with the swiveling axes: The required angle can be freely entered	Degrees
C1 (for ShopMill program)	The required angle can be freely entered	Degrees
αC (for ShopMill program)	Rotational position for a pole position	Degrees
Hirth joint 	• Round to the next Hirth joint • Round up to Hirth joint • Round down to Hirth joint Note: For machines with a Hirth joint	
Tool 	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.	
Preferred direction (for ShopMill program) 	Preferred direction of the swivel axis for several possible alignments of the machine 	

Parameter	Description	Unit
FX	Feedrate in X direction	*
FZ	Feedrate in Z direction	*
Machining	• ∇ (roughing) • ∇∇∇ (finishing) • ∇+∇∇∇ (complete machining)	
Machining direction	• face • longitudinal	
RP	Retraction plane – (only for machining direction, longitudinal, inner)	mm
Position	• front • rear • inside • outside	
D	Maximum depth infeed - (only for ∇)	mm
XDA	First grooving limit tool (abs) – (only for face machining direction)	mm
XDB	Second grooving limit tool (abs) – (only for face machining direction)	mm
UX or U	Finishing allowance in X or finishing allowance in X and Z – (only for ∇)	mm
UZ	Finishing allowance in Z – (only for UX)	mm
BL	Blank description (only for ∇) • Cylinder (described using XD, ZD) • Allowance (XD and ZD on the finished part contour) • Contour (additional CYCLE62 call with blank contour – e.g. cast iron mold)	
XD	- (only for ∇ machining) - (only for blank description, cylinder and allowance) • For blank description, cylinder – Version, absolute: Cylinder dimension ∅ (abs) – Version incremental: Allowance (inc) to maximum values of the CYCLE62 finished part contour • For blank description, allowance – Allowance on the CYCLE62 finished part contour (inc)	mm
ZD	- (only for ∇ machining) - (only for blank description, cylinder and allowance) • For blank description, cylinder – Version, absolute: Cylinder dimension (abs) – Version incremental: Allowance (inc) to maximum values of the CYCLE62 finished part contour • For blank description, allowance – Allowance on the CYCLE62 finished part contour (inc)	mm

Parameter	Description	Unit
Allowance	Allowance for pre-finishing - (only for ∇∇∇) • Yes U1 contour allowance • No	mm
U1	Compensation allowance in X and Z direction (inc) – (only for allowance) • Positive value: Compensation allowance is retained • Negative value: Compensation allowance is removed in addition to finishing allowance	mm

* Unit of feedrate as programmed before the cycle call

Hidden parameters

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.

Parameter	Description	Value	Can be set in SD
Residual material	Without subsequent residual material removal	No	
PL	Machining plane	Defined in MD 52005	
SC	Safety clearance	1 mm	x
DI	Continuous cut - (only for ∇)	0	
Set machining area limits	Set machining area limits	No	
N	Number of grooves	1	

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

10.6.12 Plunge turning residual material (CYCLE952)

Function

The "Plunge turning residual material" function is used when you want to machine the material that remained after plunge turning.

For a G code program, select the function in the screen. Material that remains as part of the finishing allowance is not residual material. The "Plunge turning residual material" function allows you to remove unwanted material with a suitable tool.

The "Plunge turning residual material" function is a software option.

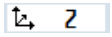
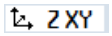
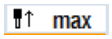
Procedure

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

2. Press the "Turning" and "Contour turning" softkeys.

3. Press the "Plunge turning residual material" softkey.
   The "Plunge turning residual material" input window opens.

G code program parameters				ShopMill program parameters		
PRG U	• Name of the program to be generated • Automatic Automatic generation of program names			T	Tool name	
PL U	Machining plane			D	Cutting edge number	
RP	Retraction plane – (only for machining direction, longitudinal, inner)	mm		S / V U	Spindle speed or constant cutting rate	rpm m/min
SC	Safety clearance	mm				
Residual material U	With subsequent residual material removal. • Yes • No					
CONR	Name to save the updated unmachined-part contour for residual material removal - (only "Yes" for residual material removal).					

Parameter	Description	Unit
TC 	Name of the swivel data set Note: The selection box only appears if more than one swivel data set has been set up.	
Retraction (for ShopMill program) 	• No The axis is not retracted before swiveling. • Z  Retraction in the direction of machine axis Z. • Z,X,Y  Move machining axes to retraction position before swiveling. • Tool direction, max.  Maximum retraction (up to the software end position) in the tool direction. • Tool direction, inc Incremental retraction (specified retraction distance, up to the software end position) in the tool direction. When retracting in the tool direction, in the swiveled machine state, several axes can move (traverse).	
ZR (for ShopMill program)	Retraction path - only for incremental retraction in	mm
Align tool through beta and gamma angles		
β (for ShopMill program) 	Align tool with swivel axes • Input value The required angle can be freely entered. • β = 0° • β = 90° 	Degrees
γ (for ShopMill program) 	• 0° • 180° • The required angle can be freely entered	Degrees
Directly position rotary axes		
B1 (for ShopMill program)	Directly align the tool with the swiveling axes: The required angle can be freely entered.	Degrees

Parameter	Description	Unit
C1 (for ShopMill program)	The required angle can be freely entered.	Degrees
αC (for ShopMill program)	Rotational position for a pole position.	Degrees
Hirth joint	• Round to the next Hirth joint • Round up to Hirth joint • Round down to Hirth joint Note: For machines with a Hirth joint	
Tool	Tool tip when swiveling • Tracking The position of the tool tip is maintained during swiveling. • No tracking The position of the tool tip is not maintained during swiveling.	
Preferred direction (for ShopMill program)	Preferred direction of the swivel axis for several possible alignments of the machine. 	
FX	Feedrate in X direction	*
FZ	Feedrate in Z direction	*
Machining	• ∇ (roughing) • ∇∇∇ (finishing)	
Machining direction	• Face • Longitudinal	
Position	• Front • rear • Inside • Outside	
D	Maximum depth infeed - (only for ∇)	mm
UX or U	Finishing allowance in X or finishing allowance in X and Z – (only for ∇)	mm
UZ	Finishing allowance in Z – (only for UX)	mm

Parameter	Description	Unit
XDA	First grooving limit tool ∅ (abs) – (end face or rear face only)	mm
XDB	Second grooving limit tool ∅ (abs) – (end face or rear face only)	mm
Allowance	Allowance for prefinishing • Yes U1 contour allowance • No	
DI	For zero: Continuous cut - (only for ∇)	mm
XD	- (only for blank description, cylinder and allowance) • For blank description, cylinder – Allowance or cylinder dimension ∅ (abs) – Allowance or cylinder dimension (inc) • For blank description, allowance – Allowance on the contour ∅ (abs) – Allowance on the contour (inc)	mm
ZD	- (only for blank description, cylinder and allowance) • For blank description, cylinder Allowance or cylinder dimension (abs or inc) • For blank description, allowance Allowance on the contour (abs or inc)	mm
U1	Compensation allowance in X and Z direction (inc) – (only for allowance) • Positive value: Compensation allowance is retained • Negative value: Compensation allowance is removed in addition to finishing allowance	mm
Set machining area limits	Set machining area limits • Yes • No	
XA XB ZA ZB	With limited machining area only, yes: 1st limit XA ∅ 2nd limit XB ∅ (abs) or 2nd limit referred to XA (inc) 1st limit ZA 2nd limit ZB (abs) or 2nd limit referred to ZA (inc)	mm
N	Number of grooves	
DP	Distance between grooves (inc)	mm

* Unit of feedrate as programmed before the cycle call

10.7 Further cycles and functions

10.7.1 Swivel plane/tool (CYCLE800)

The CYCLE800 swivel cycle is used to swivel to any surface in order to either machine or measure it. In this cycle, the active workpiece zeros and the work offsets are converted to the inclined surface taking into account the kinematic chain of the machine by calling the appropriate NC functions - and rotary axes (optionally) are positioned.

Swiveling can be realized:

- axis by axis
- via solid angle
- via projection angle
- directly

Before the rotary axes are positioned, the linear axes can be retracted if desired.

Swiveling always means three geometry axes.

In the basic version, the following functions

- 3 + 2 axes, inclined machining and
- toolholder with orientation capability

are available.

Setting/aligning tools for a G code program

The swivel function also includes the "Setting tool" and "Aligning milling tool" functions. Contrary to swiveling, when setting and aligning, the coordinate system (WCS) is not rotated at the same time.

Prerequisites before calling the swivel cycle

A tool (tool cutting edge D > 0) and the work offset (WO), with which the workpiece was scratched or measured, must be programmed before the swivel cycle is first called in the main program.

Example:

```
N1 T1D1
N2 M6
N3 G17 G54
N4 CYCLE800(1,"",0,57,0,0,0,0,0,0,0,0,1,0,1))       ;swivel ZERO to
                                                    ;initial position of the
                                                    ;machine kinematics
N5 WORKPIECE(,,,,"BOX",0,0,50,0,0,0,100,100)        ;blank agreement for
                                                    ;simulation and
                                                    ;recording
```

For machines where swivel is set-up, each main program with a swivel should start in the basic setting of the machine.

The definition of the blank (WORKPIECE) always refers to the currently effective work offset. For programs that use "swivel", a swivel to zero must be made before the blank is defined. For ShopMill programs, the blank in the program header is automatically referred to the unswiveled state.

In the swivel cycle, the work offset (WO) as well as the shifts and rotations of the parameters of the CYCLE800 are converted to the corresponding machining plane. The work offset is kept. Shifts and rotations are saved in system frames - the swivel frames - (displayed under parameter/ work offsets):

- Tool reference ($P_TOOLFRAME)
- Rotary table reference ($P_PARTFRAME)
- Workpiece reference ($P_WPFRAME)

The swivel cycle takes into account the actual machining plane (G17, G18, G19).

Swiveling on a machining or auxiliary surface always involves 3 steps:

- Shifting the WCS before rotation
- Rotating the WCS (axis-by-axis, ...)
- Shifting the WCS after rotation

The shifts and rotations refer to the coordinate system X, Y, Z of the workpiece and are therefore independent of the machine (with the exception of swivel "rotary axis direct").

No programmable frames are used in the swivel cycle. The frames programmed by the user are taken into account for additive swiveling.

On the other hand, when swiveling to a new swivel plane the programmable frames are deleted. Any type of machining operation can be performed on the swivel plane, e.g. by calling standard or measuring cycles.

The last swivel plane remains active after a program reset or when the power fails. The behavior at reset and power on can be set using machine data.

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

Block search when swiveling the plane / swiveling the tool

For block search with calculation, after NC start, initially, the automatic rotary axes of the active swivel data set are pre-positioned and then the remaining machine axes are positioned. This does not apply if a type TRACYL or TRANSMIT transformation is active after the block search. In this case, all axes simultaneously move to the accumulated positions.

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

Aligning tools

In contrast to "Swivel plane", no rotation is operative in the active frame chain (WCS) in the case of "Swivel tool" or "Align milling tool". Only the offsets calculated by the NC and the corresponding tool orientation are effective.

The maximum angular range for "Align milling tool" is limited by the traversing range of the participating rotary axes.

Name of the swivel data record

Selecting the swivel data set or deselecting the swivel data set.

The selection can be hidden by the machine data.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Approaching a machining operation

When approaching the programmed machining operation in the swiveled plane, under worst case conditions, the software limit switches could be violated. In this case, the system travels along the software limit switches above the retraction plane. In the event of violation below the retraction plane, for safety reasons, the program is interrupted with an alarm. To avoid this, before swiveling, e.g. move the tool in the X/Y plane and position it as close as possible to the starting point of the machining operation or define the retraction plane closer to the workpiece.

Retraction

Before swiveling the axes or deselecting the swivel data set, move the tool to a safe retraction position. The retraction versions available are defined when starting up the system (commissioning).

The retraction mode is modal. When a tool is changed or after a block search, the retraction mode last set is used.

Machine manufacturer

Please observe the information provided by the machine manufacturer.

⚠ WARNING

Risk of collision

You must select a retraction position that avoids a collision between the tool and workpiece when swiveling.

Tool

To avoid collisions, use 5-axis transformation (software option) to define the position of the tool tip during swiveling.

- Correct
  The position of the tool tip is corrected during swiveling (tracking function).
- No correction
  The position of the tool tip is not corrected (not tracked) during swiveling.

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

Swivel plane (only for G code programming)

- **New**

  Previously active swivel frames and programmed frames are deleted. A new swivel frame is formed according to the values specified in the input screen.
  Every main program must begin with a swivel cycle with the new swivel plane. This is in order to ensure that a swivel frame from another program is not active.

- **Additive**

  The swivel frame is added to the swivel frame from the last swivel cycle.
  If multiple swivel cycles are programmed in a program and programmable frames are also active between them (e.g. AROT ATRANS), they will be taken into account in the swivel frame.
  If a swivel data set is activated that was not previously active, the swivel frames are not deleted.

If the currently active work offset contains rotations, e.g. due to previous workpiece measuring operations, they will be taken into account in the swivel cycle.

Swivel mode

Swiveling can either be realized axis-by-axis, using the angle in space, using the projection angle or directly. The machine manufacturer determines when setting up the "Swivel plane/swivel tool" function which swivel methods are available.

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

- **Axis by axis**
  In the case of axis-by-axis swiveling, the coordinate system is rotated about each axis in turn, with each rotation starting from the previous rotation. The axis sequence can be freely selected.
- **Solid angle**
  With the solid angle swiveling option, the tool is first rotated about the Z axis and then about the Y axis. The second rotation starts from the first.
- **Projection angle**
  When swiveling using the projection angle, the angle value of the swiveled surface is projected onto the first two axes of the right-angle coordinate system. The user can freely select the axis rotation sequence.
  The 3rd rotation is based on the previous rotation. The active plane and the tool orientation must be taken into consideration when the projection angle is used:
  - For G17 projection angle XY, 3rd rotation around Z
  - For G18 projection angle ZX, 3rd rotation around Y
  - For G19 projection angle YZ, 3rd rotation around X

When programming projection angles around XY or YX, the new X-axis of the swiveled coordinate system lies in the old ZX plane.

When programming projection angles around XZ or ZX, the new Z-axis of the swiveled coordinate system lies in the old Y-Z plane.

When programming projection angles around YZ or ZY, the new Y-axis of the swiveled coordinate system lies in the old X-Y plane.

- **directly**
  For direct swiveling, the required positions of the rotary axes are specified. The HMI calculates a suitable new coordinate system based on these values. The tool axis is aligned in the Z direction. You can derive the resulting direction of the X and Y axis by traversing the axes.

Note

Direction of rotation

The positive direction of each rotation for the different swivel versions is shown in the help displays.

Axis sequence

Sequence of the axes which are rotated around:

XYZ or XZY or YXZ or YZX or ZXY or ZYX

Direction (minus/plus)

Direction reference of traversing direction of rotary axis 1 or 2 of the active swivel data set (machine kinematics). The NC calculates two possible solutions of the rotation / offset programmed in CYCLE800 using the angle traversing range of the rotary axes of the machine kinematics. Usually, only one of these solutions is technologically suitable. The solutions differ by 180 degrees in each case. Selecting the "minus" or "plus" direction determines which of the two possible solutions is to be applied.

- "Minus" → Lower rotary axis value
- "Plus" → Higher rotary axis value

Also in the basic setting (pole setting) of the machine kinematics, the NC calculates two solutions and these are approached by CYCLE800. The reference is the rotary axis that was set as direction reference when commissioning the "swivel" function.

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

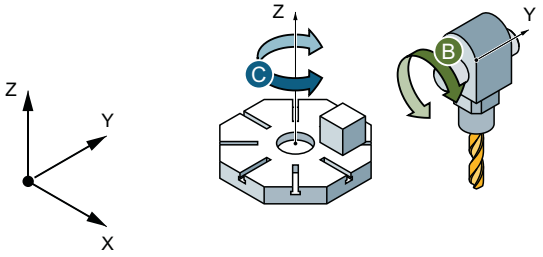
If one of the two positions cannot be reached for mechanical reasons, the alternative position is automatically selected irrespective of the setting of the "Direction" parameter.

Example 1: A rotary axis rotates in the basic position around the tool axis

- Machine kinematics with swivel head and swivel table.
- Swivel head with rotary axis 1 (B) rotates around machine axis Y.
  - Angular range of rotary axis B from -90 to +90 degrees.
- Swivel table with rotary axis 2 (C) rotates around machine axis Z.
  - Angular range of rotary axis 2 (C) from 0 to 360 degrees (modulo 360).
- Machine manufacturer has set the direction reference to rotary axis 1 (B) when he commissioned the swivel function.
- A rotation around X (WCS) of 10 degrees is programmed in the swivel cycle.

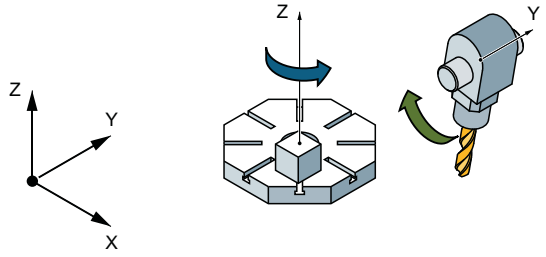
Basic position

Basic position (pole setting) of the kinematics (B = 0 degrees, C = 0 degrees)



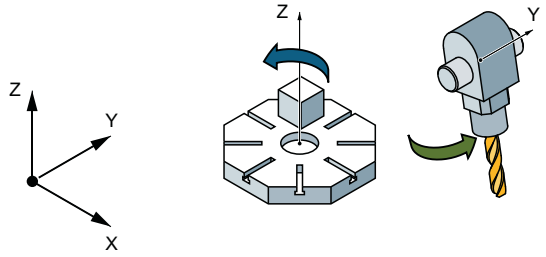
Swivel in the "+" (plus) direction

- Rotary axis B moves to +10 degrees in the positive direction.
- Rotary axis C moves to 270 degrees.



Swivel in the "-" (minus) direction

- Rotary axis B moves to -10 degrees in the negative direction.
- Rotary axis C moves to 90 degrees (rotation around X).



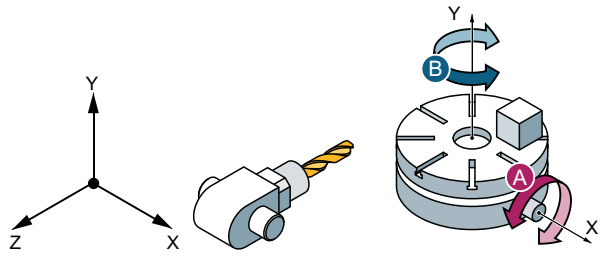
The two "Minus" or "Plus" direction settings enable a workpiece to be machined with swiveled planes. The two solutions calculated by the NC differ by 180 degrees (see rotary axis C).

Example 2: Horizontal machine - no rotary axis rotates in the basic position around the tool axis

- Machine kinematics with swivel table.
- 1st rotary axis (A) in the swivel table rotates around machine axis X.
  - Angular range, rotary axis 1 (A) from -90 degrees to +90 degrees.
- 2nd rotary axis (B) in the swivel table rotates around machine axis Y.
  - Angular range, rotary axis 2 (B) from 0 degrees to 360 degrees (modulo 360).
- Machine manufacturer has set the direction reference to rotary axis 1 (A) when he commissioned the swivel function.
- In the swivel cycle, a rotation (ZYX) is programmed through Y=30 degrees and X=10 degrees.

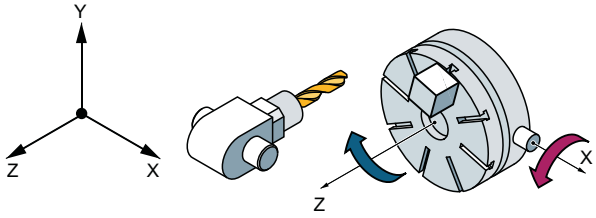
Basic position

Basic position of the kinematics (B = 0 degrees, A = 0 degrees)



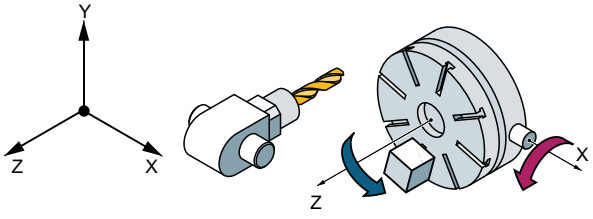
Direction "+" (plus)

- Rotary axis B moves to +30 degrees in the positive direction.
- Rotary axis A moves to 90 degrees.



Direction "-" (minus)

- Rotary axis B moves to 210 degrees (30 degrees - 180 degrees = -150 degrees = 210 degrees).
- Rotary axis A moves to 90 degrees.



The two "Minus" or "Plus" direction settings enable a workpiece to be machined with swiveled planes. The two solutions calculated by the NC differ by 180 degrees (see rotary axis B).

In this case, the "Plus" solution corresponds to the solution without additional rotation in the Actframe (180° in Z).

Procedure

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

2. Select the "Miscellaneous" softkey.

3. Press the "Swivel plane" softkey.
   The "Swivel plane" input window opens.

4. Press the "Basic setting" softkey if you wish to reestablish the initial state, i.e. you wish to set the values back to 0.
   You use this,for example, to swivel the coordinate system back into its original orientation.

G code program parameters				ShopMill program parameters		
PL	Machining plane			T	Tool name	
				D	Cutting edge number	
				F	Feedrate	mm/min mm/rev
				S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description		Unit
TC	Name of the swivel data record		
Retraction	No	No retraction before swiveling	
	ink	Incremental retraction in tool direction The retraction path is entered into parameter ZR.	
	max	Maximum retraction in tool direction	
	Z	Retraction in the direction of machine axis Z	
	Z XY	Retract towards the machine axis Z and then in the direction X, Y	
ZR	Retraction path - (only for incremental retraction in the tool direction)		
Swivel plane	• New: New swivel plane • Additive: Additive swivel plane		
X0	Reference point for rotation X		
Y0	Reference point for rotation Y		
Z0	Reference point for rotation Z		
Swivel mode	• Axis by axis: Rotate coordinate system axis-by-axis • Solid angle: Swivel via solid angle • Proj. angle: Swiveling via projection angle • Direct: Directly position rotary axes		

Parameter	Description		Unit
Axis sequence	Sequence of the axes which are rotated around: - (only for axis-by-axis swivel mode) XYZ or XZY or YXZ or YZX or ZXY or ZYX		
X	Rotation around X	- (only for axis sequence)	Degrees
Y	Rotation around Y		Degrees
Z	Rotation around Z		Degrees
Projection position	Position of the projection in space - (only for swivel mode, projection angle) Xα, Yα, Zβ or Yα, Zα, Zβ or Zα, Xα, Zβ		
Xα	Projection angle	- (only for projection position)	Degrees
Yα	Projection angle		Degrees
Zβ	Angle of rotation in the plane		Degrees
Name of rotary axis 1	Angle of rotation of rotary axis 1	- (only for direct swivel mode)	Degrees
Name of rotary axis 2	Angle of rotation of rotary axis 2		Degrees
Z	Angle of rotation in the plane		Degrees
X1	Zero point of rotated surface X		
Y1	Zero point of rotated surface Y		
Z1	Zero point of rotated surface Z		
Direction	Direction reference of traversing direction of rotary axis 1 or 2 - (not for direct swivel mode) • + • -		
Tool	Tool tip position when swiveling		
		Tracking The position of the tool tip is maintained during swiveling.	
		No tracking The position of the tool tip changes during swiveling.	

Call of an orientation transformation (TRAORI) after swiveling

If a program activating the orientation transformation (TRAORI) is to be executed on the swiveled machining plane, the system frames – tool reference and rotary table reference – for the swivel head or swivel table must be deactivated before TRAORI is called (see example). The workpiece reference (WPFRAME) is retained.

Example (machine with swivel table)

```
N1 G54
N2 T="MILL_10mm"
N3 M6
N4 CYCLE800(1,"",0,57,0,40,0,-45,0,0,0,0,0,-1)              ;Swivel cycle
```

```
N5 CYCLE71(50,24,2,0,0,0,80,60,0,4,10,5,0,2000,31,5)      ;Face milling
N6 TCARR=0                                                ;Swivel data set ;dese-
                                                          lection
N7 PAROTOF                                                ;Deactivate ;rotary ta-
                                                          ble reference
N8 TOROTOF                                                ;Deactivate ;tool refer-
                                                          ence
                                                          ;(only for ;swivel head
                                                          and ;mixed ;kinematics)
N9 TRAORI                                                 ;Activate the ;orienta-
                                                          tion ;transformation
N10 G54                                                   ;Activate ;work offset
N11 EXTCALL "ROLL"                                        ;Call 5-;axis machining
                                                          ;program with ;direc-
                                                          tion vectors ;(A3, B3,
                                                          C3)
N12 M2
```

10.7.2 Swiveling tool (CYCLE800)

10.7.2.1 Swiveling tool/preloading milling tools - only for G code program (CYCLE800)

After "Swivel plane", the tool orientation is always perpendicular on the machining plane. When milling with radial cutters, it can make technological sense to set the tool at an angle to the normal surface vector. In the swivel cycle, the setting angle is generated by an axis rotation (max. +/- 90 degrees) to the active swivel plane. When setting, the swivel plane is always “additive”. With "Setting tool", only rotations are displayed on the swivel cycle input screen form. The user can freely select the rotation sequence.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

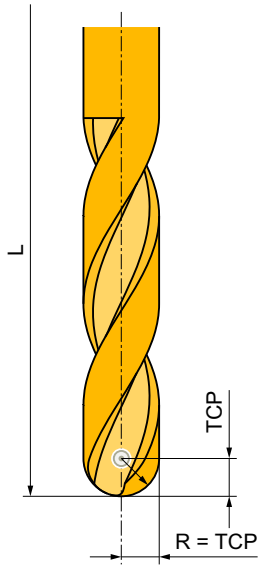


Figure 10-19 The length up to the TCP (Tool Center Point) must be entered as tool length of the radial cutter.

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 "Swivel tool" and "Setting milling tool" softkeys.
   The "Setting tool" input window opens.

Parameter	Description		Unit
PL	Plane for milling		
TC	Name of the swivel data record		
Retraction	No	No retraction before swiveling	
	ink	Incremental retraction in tool direction The retraction path is entered into parameter ZR.	
	max	Maximum retraction in tool direction	
	Z	Retraction in the direction of machine axis Z	
	Z XY	Retract towards the machine axis Z and then in the direction X, Y	
ZR	Retraction path - (only for incremental retraction in the tool direction)		
Axis sequence	Sequence of the axes which are rotated around XY or XZ or YX or YZ or ZX or ZY		
X	Rotation around X		Degrees
Y	Rotation around Y		Degrees

Parameter	Description		Unit
Tool	Tool tip position when swiveling		
		Tracking The position of the tool tip is maintained during swiveling.	
		No tracking The position of the tool tip changes during swiveling.	

10.7.3 Aligning turning tools (CYCLE800) - millling/turning machine

Function

The purpose of the "Align turning tool" function is to support combined milling-turning machines, i.e. milling machines which have been expanded to include turning functionality.

Tool orientation in the turning mode is implemented using a swivel axis (e.g. B1) with an associated milling spindle (e.g. SP1).

Unlike "Swivel plane", for "Align tool", the tool coordinate system is always adapted such that the Z-axis runs through the center of the workpiece holder.

The maximum angular range for "Align tool" is limited by the traversing range of the participating rotary axes and furthermore technically depending on the tool used.

For "Align tool", the data of the tool is calculated online based on the tool orientation. For a turning tool, this involves the cutting edge position (SL), the holder angle and the cutting direction.

The "Align turning tool" function is intended for turning applications. It must be programmed before the corresponding turning cycles in the NC program.

Definition of the β and γ angles

The beta and gamma angles orientate the turning tools. The angles are referred to WCS. If the workpiece coordinate system corresponds to the machine coordinate system, the tool data remains unchanged for β=0° / γ=0° (cutting edge position, holder angle, ...).

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

Turning tools fixed on the headstock

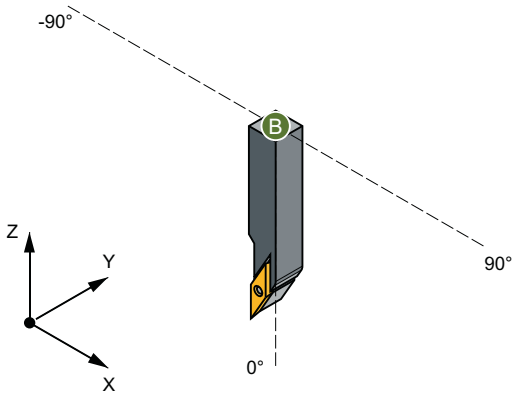
Turning tools that are directly mounted on the headstock cannot be turned via the tool spindle. Therefore the entry of γ is not possible.

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

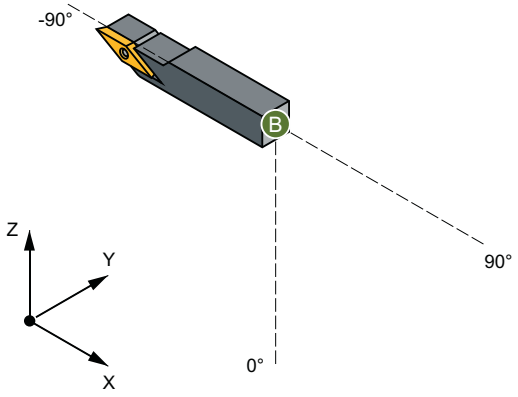
"Vertical milling machine" as example

Initial state of the kinematics B=0

- Align tool ß = 0°, γ = 0°, SL3 (SL = cutting edge position)



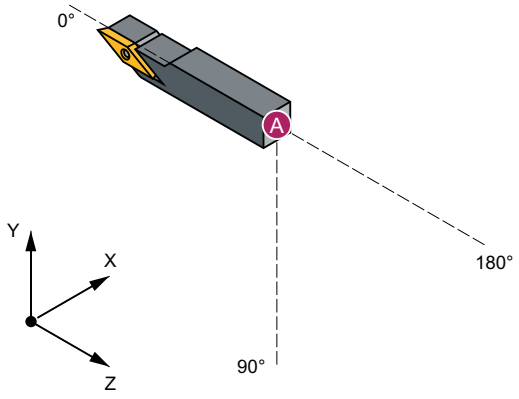
- Align tool ß = -90°, γ = 0°, SL4 (SL = cutting edge position)



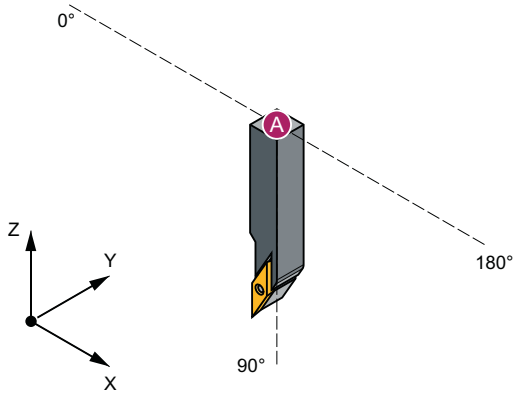
"Horizontal milling machine" as example

Initial state of the kinematics A = 0°

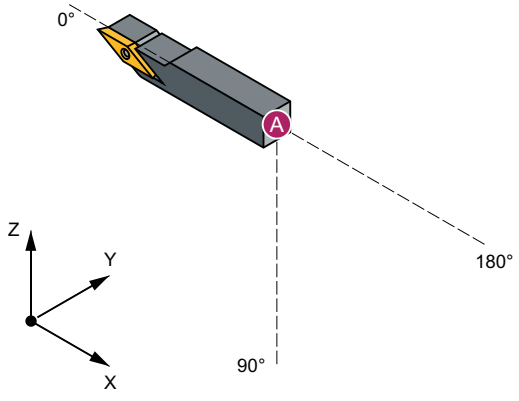
- Workpiece coordinate system when milling



- Align tool ß = 0°, A = 90°, γ = 0°, SL3 (SL = cutting edge position)



- Align tool ß = 90°, A = 0°, γ = 0°, SL4 (SL = cutting edge position)



SL = cutting edge position

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 "Swivel tool" and "Align turning tool" softkeys.
   The "Align turning tool" input window opens.

Parameter	Description		Unit
TC	Name of the swivel data record		
Retraction	No	No retraction before swiveling	
	Z	Retraction in the direction of machine axis Z	
	Z XY	Retract towards the machine axis Z and then in the direction X, Y	
	max	Maximum retraction in tool direction	
	ink	Incremental retraction in tool direction	
RD	Retraction path - (only for incremental retraction in the tool direction)		
Swivel mode	B	Swivel mode "direct": The position of the swivel axis is selected directly.	
	β	Swivel mode "align": The tool position is specified.	
β	Tool alignment with the swivel axis		Degrees
	↓	0°	
	←	90°	
	0	Value entry: The required angle can be freely entered	
	Note: The selected angle does not depend on the machine geometry.		
γ	Tool alignment with the swivel axis • 0.000° Tool alignment with tool spindle • 180.000° Tool alignment with tool spindle • fixed Tool is fixed on the tool headstock Note: Please observe the information provided by the machine manufacturer.		Degrees
B	Tool alignment directly with the swivel axis		

Parameter	Description		Unit
SP1	Tool alignment directly with the swivel axis • 0 Direct entry of the angle • fixed Tool is fixed on the tool headstock Note: Please observe the information provided by the machine manufacturer.		Degrees
Hirth gearing (only in the "align" swivel mode)		Round β to the next Hirth gearing	
		Round up β to the next Hirth gearing	
		Round off β to the next Hirth gearing	
Tool	Tool tip position when swiveling		
		Tracking The position of the tool tip is maintained during swiveling.	
		No tracking The position of the tool tip is not maintained during swiveling.	
Preferred direction	+	Select the preferred swivel direction if the kinematics facilitates two solutions for the selected position of the tool with respect to the workpiece	
	−		
Rotation plane	Alignment of the rotation plane (with swivel mode "direct" or γ = fixed)		
		Standard	
		Mirrored	
αC	Align the plane of rotation in the pole position - (only in the "align" swivel mode and β "zero degrees")		Degrees

10.7.4 High-speed settings (CYCLE832)

Function

The "High Speed Settings" function (CYCLE832) is used to preset data for the machining of free-form surfaces so that optimum machining is possible.

The call of CYCLE832 contains three parameters:

- Machining type (technology)
- Axis tolerance
- Input of the orientation tolerance (for 5-axis machines)

Machining of free-form surfaces involves high requirements for both velocity and precision and surface quality.

With the "High Speed Settings" function, you can achieve optimum velocity control depending on the type of machining (roughing, semi-finishing, finishing/speed or fine finishing/precision). It is also possible to machine and process very fine structures. For this purpose, the cycle

activates the compressor COMPCAD (for Advanced Surface option) or COMPSURF (for TOP Surface option).

Note

Programming a cycle

Program the cycle in the technology program before the geometry program is called.

If CYCLE832 is called before CYCLE800 in the part program sequence, the use of incorrect round axes may be considered for the compressor. A required CYCLE800 call must be made before a CYCLE832 call.

Software option

To use the "High Speed Settings" (CYCLE832) function, you require the "Advanced Surface" software option.

Default values

You can use the "Default values" softkey to assign default values to the tolerance parameters.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Surface smoothing

For the "High Speed Settings" (CYCLE832) function, there are two ways in which the surface quality of free-form surfaces can be improved. To smooth the surface, the continuous-path control is optimized within a defined contour tolerance.

Software option

To smooth contours with the "High Speed Settings" (CYCLE832) function, you require the "Top Surface" software option.

Machining methods

You can choose between the following technological machining operations:

- "Roughing"
- "Semi-finishing"
- "Finishing/speed"
- "Fine finishing/precision"
- "Deselected" (default setting)

Note

Plain text entry

You can enter the parameters in plain text in the "Machining" selection box. Plain text is generated for the "Machining mode" parameter when the input screen is closed (e.g. _ROUGH for roughing).

For CAM programs in the HSC range, the four machining types directly relate to the accuracy and speed of the path contour (see help screen).

The operator/programmer uses the tolerance value to give a corresponding weighting.

Corresponding to the appropriate G commands, the four machining types are assigned to technology G group 59:

Machining type	Technology G group 59
Roughing	DYNROUGH
Semi-finishing	DYNSEMIFIN
Finishing/speed	DYNFINISH
Fine finishing/precision	DYNPREC
Deselection	DYNNORM

In the "Machine" operating area, the G functions that are active in the part program are shown in the "G functions" window.

Orientation tolerance

You can enter the orientation tolerance for applications on machines with the dynamic multi-axis orientation transformation (TRAORI).

MD note

Additional G commands that are available for use in machining free-form surfaces, are also activated in the High Speed Settings cycle.

When deselecting CYCLE832, the G groups are programmed to the settings - during the program run time - that are declared in the machine data for the reset state.

More information

More information on the "High Speed Settings" (CYCLE832) function can be found here:

- SINUMERIK Operate Commissioning Manual
- Programming Manual NC Programming

Procedure

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

Press the "Various" softkey.
3.

Press the "High Speed Settings" softkey.
The "High Speed Settings" input window is opened.
4.

Press the "Default values" softkey if you want to store default values for axis tolerance values depending on the machining.

Parameter	Description	Unit
Machining	• ∇ (roughing) • ∇∇ (semi-finishing) • ∇∇∇ (finishing/speed) • ∇∇∇∇ (fine finishing/precision) • Deselection	
Mold-making function	• Advanced Surface • Top Surface **Note** The field can be hidden. Please observe the information provided by the machine manufacturer.	
Contour tolerance	• Input of the maximum allowance from the programmed contour. • Standard default values depending on the type of machining via the "Default values" softkey: – ∇ (roughing): 0.100 – ∇∇ (semi-finishing): 0.050 – ∇∇∇ (finishing/speed): 0.010 – ∇∇∇∇ (fine finishing/precision): 0.005 **Note** The default values may have been changed by the manufacturer. Please observe the information provided by the machine manufacturer.	
Smoothing (not for "Advanced Surface")	• Yes Optimized path within the contour tolerance • No Path close to contour **Note** The field can be hidden. Please observe the information provided by the machine manufacturer.	

Parameter	Description	Unit
Multi-axis program 	Multi-axis program for 5-axis machines • Yes The orientation tolerance > 0 degrees can be entered here • No The value 1 is entered automatically **Note** The field can be hidden. Please observe the information provided by the machine manufacturer.	
ORI tolerance	Specification of the maximum allowance from the programmed tool orientation (for 5-axis machines). Standard default values depending on the type of machining via the "Default values" softkey: • ∇ (roughing): 1 • ∇∇ (semi-finishing): 0.5 • ∇∇∇ (finishing/speed): 0.3 • ∇∇∇∇ (fine finishing/precision): 0.1	

10.7.5 Subroutines

If you require the same machining steps when programming different workpieces, you can define these machining steps in a separate subprogram. You can then call this subprogram in any program.

Identical machining steps therefore only have to be programmed once.

A distinction is not made between the main program and subprograms. This means that you can call a "standard" ShopMill or G code program in another ShopMill program as a subprogram.

You can also call another subprogram in the subprogram. The maximum nesting depth is 15 subprograms.

Note

You cannot insert subprograms into linked blocks.

If you want to call a ShopMill program as a subprogram, the program must already have been calculated once (load or simulate program in the "Machine Auto" mode). This is not necessary for G code subprograms.

Program clipboard

If you use the "Execution from external storage (EES)" software option, the subprogram can be stored locally or externally in an arbitrary program memory configured for EES.

If you use the "CNC user memory extended" software option, the subprogram can be stored on the system CF card in a program memory configured for EES.

Without these two software options, the subprogram must always be stored in the NCK work memory (in a separate "XYZ" directory or in the "Subprograms" directory). If you still want to call a subprogram located on another drive, you can use G code command "EXTCALL".

Program header

Please note that when a subprogram is called, the settings in the program header of the subprogram are evaluated. These settings also remain active even after the subprogram has ended.

If you wish to activate the settings from the program header of the main program again, you can make the settings again in the main program after calling the subprogram.

Procedure

1. Generate a ShopMill or G code program that you wish to call as a subprogram in another program.
2. Position the cursor in the work plan or in the program view of the main program on the program block after which you wish to call the subprogram.
3.

Press the "Various" and "Subroutine" softkeys.
4. Enter the path of the subprogram if the desired subprogram is not stored in the same directory as the main program.
The subprogram is thus executed in the position pattern.
5.

Press the "Accept" softkey.
The subprogram call is inserted in the main program.

Parameter	Description	
Path/workpiece	Path of the subprogram if the desired subprogram is not stored in the same directory as the main program.	
Program name	Name of the subprogram that is to be inserted.	

Programming example

```
N10 T1 D1                          ;Load tool
N11 M6
N20 G54 G710                       ;Select work offset
N30 M3 S12000                      ;Switch-on spindle
N40 CYCLE832(0.05,3,1)             ;Tolerance value 0.05 mm, machining type,
                                   roughing
N50 EXTCALL"CAM_SCHRUPP"           Externally call subprogram CAM_SCHRUPP
N60 T2 D1                          ;Load tool
N61 M6
```

```
N70 CYCLE832(0.005,1,1)                    ;Tolerance value 0.005 mm, machining
                                           type, finishing
N80 EXTCALL"CAM_SCHLICHT"                  ;Call subprogram CAM_SCHLICHT
N90 M30                                    ;End of program
```

The subprograms CAM_SCHRUPP.SPF, CAM_SCHLICHT.SPF contain the workpiece geometry and the technological values (feedrates). These are externally called due to the program size.

10.7.6 Adapt to load (CYCLE782)

Function

The clamping and weight of the workpiece affect the dynamic response of your machine. Using the "Adjust to load" function, you can automatically adapt the controller setting of the drive or the dynamic response parameters of an axis to a specific situation.

Software option

To use the function "Adjust to load" (CYCLE782), you need the software option "Intelligent load matching".

To adjust to the load, you can either use a fixed value for the moment of inertia or calculate the load automatically during program execution. To measure the moment of inertia, acceleration movements are performed. You can also view the measurement result during program execution.

Machine manufacturer

Please observe the information provided by the machine manufacturer.

If the "Adjust to load" function was previously executed, you can transfer the result of the last measurement to the screen form with a softkey. This allows the adjustment be made without having to redetermine the moment of inertia again each time the program starts.

Procedure

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

Press the "Various" softkey.
3.
Press the ">>" and "Adjust to load" softkeys.

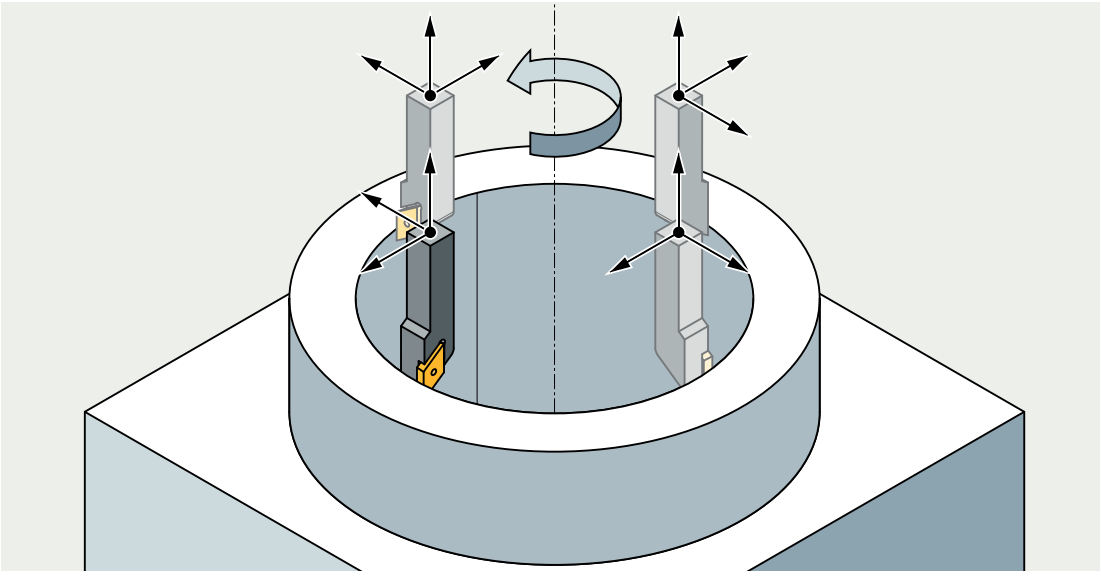
The input window "Adjust to load" opens.

Parameter	Description	Unit
Axis 	Channel axis name	
Axis dynamics 	• Default Adaptation deactivated. The default controller settings are used. • Adapt Adaptation activated. The controller settings are adapted to the currently active load.	
Entire load 	Currently active load • Measure: Travel movements are executed to derive the load. • Set: Fixed value for the moment of inertia is applied.	
Measurement result display 	Measurement result display • On • Off	
Display mode 	Duration of measurement result display • autom. 8s Disappears automatically after 8 s • NC Start Acknowledge the display with the <NC-START> key	
Moment of inertia	Moment of inertia of the entire load The value of the most recent measurement can be applied with the softkey "Insert last result".	kgm^2
Mass	Mass of the entire load (only for linear axes with linear drives)	kg

10.7.7 Interpolation turning (CYCLE806)

10.7.7.1 Function

With the interpolation turning function, it is possible to carry out turning on a machine tool with at least three linear axes and a positioning-capable spindle. In interpolation turning, the workpiece is not rotated for turning. Instead, the tool rotates around the workpiece. The linear axes perform the circular motion and the tool spindle holds the cutting edge perpendicular to the circle. The axis of rotation can be located at any workpiece position and on 4 or 5-axis machines, freely oriented in space. This permits you to create turning surfaces, for example, on flanges that are off-center or even diagonally in space. This allows you to reduce setup times.



Interpolation turning is available on milling machines and turning machines.

Software option

To use the "Interpolation turning" function you require the software option "InterPolation Turning".

If interpolation turning is enabled, turning tools will be offered in the tool list.

The CYCLE800 can be swiveled so that the turning tool is perpendicular to the machining plane.

CYCLE806 can be used to activate interpolation turning. After that, certain turning cycles can be used.

Machining while interpolation turning is active always takes place in plane G18.

Note

The "Interpolation turning" function is not available for working with "Manual machine".

Cycles for interpolation turning

You can use the following standard cycles for interpolation turning:

- Stock removal (CYCLE951), all variants
- Stock removal (CYCLE952): Contour roughing, contour grooving
- Groove (CYCLE930), all variants
- Undercut (CYCLE940), all variants

You can also use the same turning operations under ShopMill and ShopTurn.

Spindle positioning by CYCLE806

When activating the interpolation turning function, the cutting edge of the turning tool in the WCS must be set to Y = 0. In the CYCLE806, if necessary, the WCS and the tool spindle are rotated and thus the position of the turning tool is adjusted. Before starting the interpolation turning function, the tool must be at a safe distance from the workpiece.

More information

Further information on the "Interpolation turning" function is provided in the Transformations Function Manual.

See also

Parameter (Page 678)

10.7.7.2 Selecting/deselecting interpolation turning - CYCLE806

1. Load the turning tool.
2. Set the swivel plate, if applicable.
3. Preposition the tool.
4. Start interpolation turning with the CYCLE806:
   - Specify the center of rotation (XC/YC).
   - The turning tool is oriented in the direction of the center of rotation.

   The compensating movement starts with the start of the spindle (M3/M4).
   In the next step you can use standard cycles, user cycles or G code.
5. End interpolation turning with the CYCLE806:
   - The turning tool is turned back to the starting position of interpolation turning.

10.7.7.3 Calling the cycle

Procedure

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

Press the softkeys ">" and "Interpol. turning".
The input window opens.
3.

Press the "Sel./Desel. interpol.turn." softkey.

10.7.7.4 Parameter

The following table shows the available parameters for interpolation turning (CYCLE806):

Parameters G code program				Parameters ShopTurn/ShopMill program		
PL	Machining plane G17 (XY)			T	Tool name	
				F	Feedrate	mm/rev
γ	• 0° • 180°	Degrees		γ	• 0° • 180°	Degrees
S/V	Spindle speed or constant cutting speed	rpm Path/min		S/V	Spindle speed or constant cutting speed	rpm m/min
DIR	• Spindle rotates clockwise • Spindle rotates counter-clockwise			Face Y/Face B (for ShopTurn program)	Machining planes for aligning and swiveling tools on the main spindle and counterspindle or for machining inclined surfaces	
XC	Define machining center point in the current plane			XC	Define machining center point in the current plane	
YC	Define machining center point in the current plane			YC	Define machining center point in the current plane	

See also

Function (Page 676)
Calling the cycle (Page 678)

10.8 Additional cycles and functions in ShopMill

10.8.1 Transformations

To make programming easier, you can transform the coordinate system. Use this possibility, for example, to rotate the coordinate system.

Coordinate transformations only apply in the actual program. You can define shift, rotation, scaling or mirroring. You can select between a new or an additive coordinate transformation.

In the case of a new coordinate transformation, all previously defined coordinate transformations are deselected. An additive coordinate transformation acts in addition to the currently selected coordinate transformations.

Note

Transformations with virtual axes

Please note that when selecting TRANSMIT or TRACYL offsets, scaling and mirroring, the real Y axis is not transferred into the virtual Y axis.

Offsets, scalings and mirroring of the virtual Y axis are deleted for TRAFOOF.

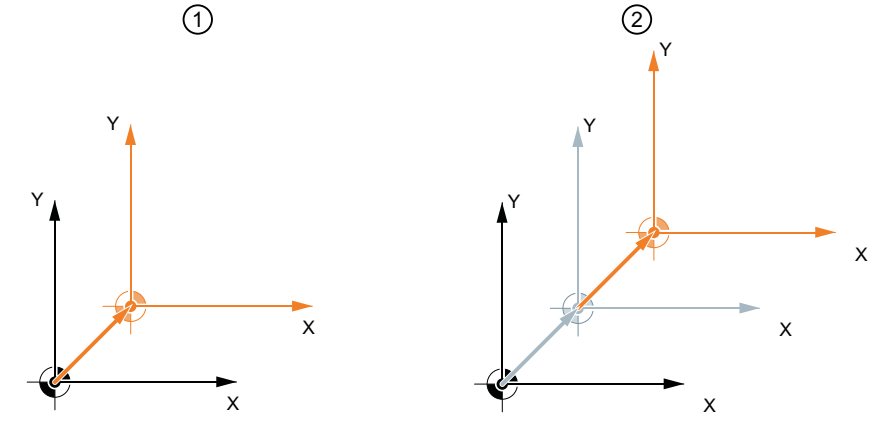
Procedure for work offset, offset, rotation, scaling or mirroring

1. The ShopMill program has been created and you are in the editor.
2. Press the "Various" and "Transformation" softkeys.

3. Press the "Work offsets" softkey.
   The "Work offsets" input window opens.
   - OR -
   Press the "Offset" softkey.
   The "Offset" input window opens.
   - OR -
   Press the "Rotation" softkey.
   The "Rotate" input window opens.
   - OR -
   Press the "Scaling" softkey.
   The "Scaling" input window opens.
   - OR -
   Press the "Mirroring" softkey.
   The "Mirroring" input window opens.

10.8.2 Translation

For each axis, you can program an offset of the zero point.

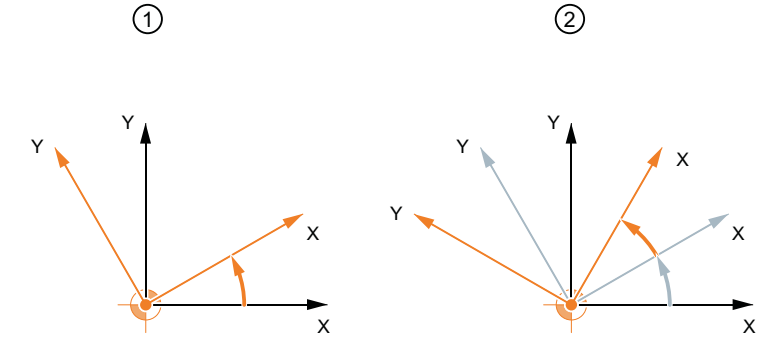


① New offset
② Additive offset

Parameter	Description	Unit
Offset 	• New New offset • Additive Additive offset	
X	Offset X	mm
Y	Offset Y	mm
Z	Offset Z	mm

10.8.3 Rotation

You can rotate every axis through a specific angle. A positive angle corresponds to counterclockwise rotation.

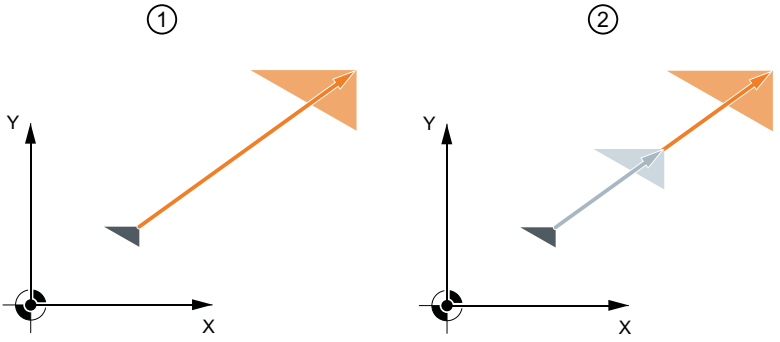


① New rotation
② Additive rotation

Parameter	Description	Unit
Rotation	• New New rotation • Additive Additive rotation	
X	Rotation around X	Degrees
Y	Rotation around Y	Degrees
Z	Rotation around Z	Degrees

10.8.4 Scaling

You can specify a scale factor for the active machining plane as well as for the tool axis. The programmed coordinates are then multiplied by this factor.



① New scaling
② Additive scaling

Parameter	Description	Unit
Scaling	• New New scaling • Additive Additive scaling	
XY	Scale factor XY	
Z	Scale factor Z	

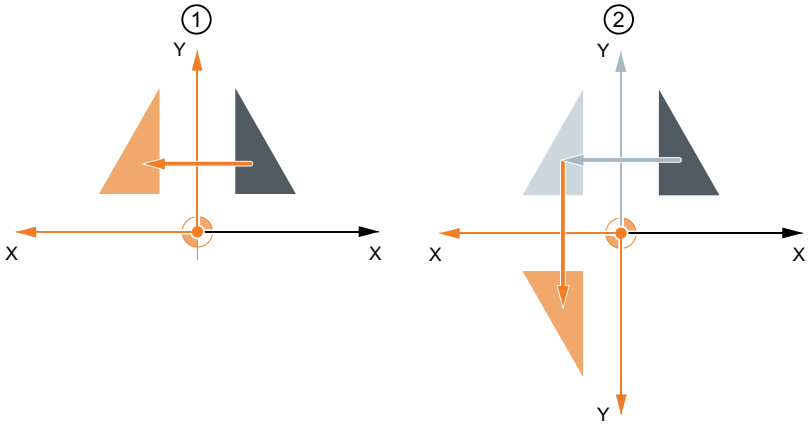
10.8.5 Mirroring

Furthermore, you can mirror all axes. Enter the axis to be mirrored.

Note

Travel direction of the milling cutter

Note that with mirroring, the travel direction of the cutting tool (conventional/climb) is also mirrored.



① New mirroring
② Additive mirroring

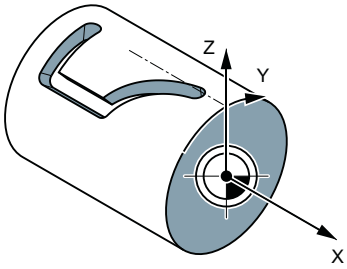
Parameter	Description	Unit
Mirroring U	• New New mirroring • Additive Additive mirroring	
X U	Mirroring of the X axis, on/off	
Y U	Mirroring of the Y axis, on/off	
Z U	Mirroring of the Z axis, on/off	

10.8.6 Cylinder surface transformation

You require the cylinder surface transformation to machine

- Longitudinal grooves on cylindrical bodies,
- Transverse grooves on cylindrical objects
- grooves with any path on cylindrical bodies.

The path of the grooves is programmed with reference to the unwrapped, level surface of the cylinder. The groove can be programmed using line/circle, drilling or milling cycles or with contour milling (free contour programming).



Supporting several data sets

- If several Tracyl data sets have been set up, then the corresponding rotary axes can be selected.
- If several Tracyl data sets have been set up around a rotary axis, then the corresponding numbers of the data sets can be selected.

Slot side compensation

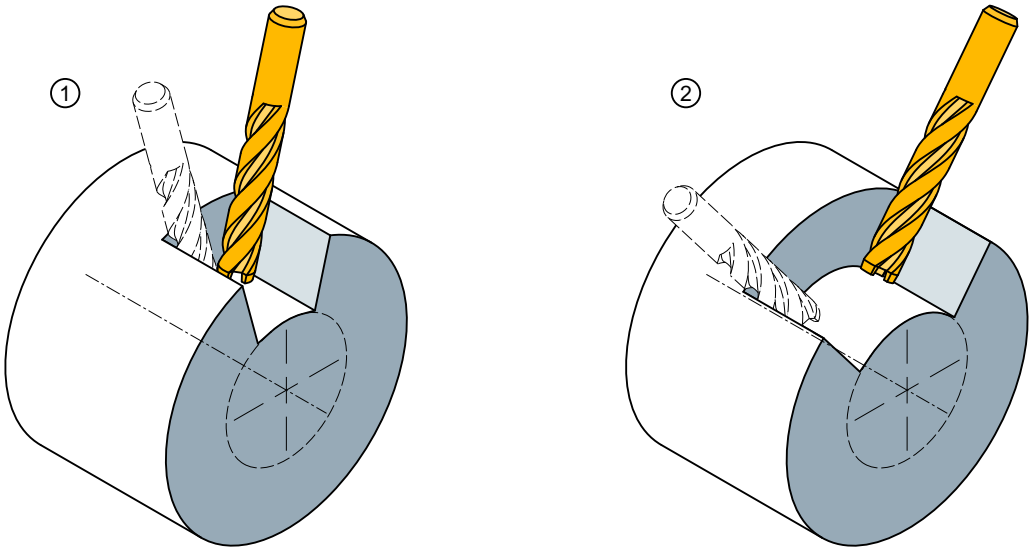
Cylinder surface transformation is available in the following versions:

Slot side compensation off

When slot side compensation is deactivated, any type of slot with parallel sides can be machined if the tool diameter is equal to the slot width.

The slot sides are not parallel if the slot width is larger than the tool diameter.

The slot contour is programmed for machining purposes.



① Longitudinal groove
② Transverse groove

Slot side compensation on

This function is only permissible during path milling with switched-on radius compensation.

When slot side compensation is on, slots with parallel sides are machined even if the slot width is larger than the tool diameter.

The slot contour must not be programmed for machining purposes, but the imaginary center point path of a bolt guided in the slot; whereby the bolt must move along every side. The slot width is determined by parameter D.

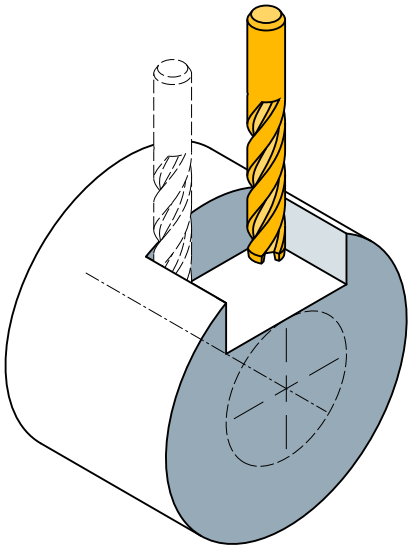


Figure 10-20 Longitudinal groove with parallel limitation

Note

Selecting slot side compensation

The slot side compensation selection depends on the transformation type.

General procedure

The basic programming procedure is as follows:

- Select work offset for cylinder surface transformation (e.g. offset the zero point on the center point of the cylinder end face)
- Position the Y axis (Y axis must be positioned prior to cylinder surface transformation because it is defined differently after transformation)
- Activate cylinder surface transformation
- Select work offset for machining on the developed cylinder surface (e.g. shift zero point to the zero point on the workpiece drawing)
- Program machining operation (e.g. enter contour and path milling)
- Deactivate cylinder surface transformation

The programmed cylinder surface transformation is simulated only as developed peripheral surface.

Note

The work offsets active prior to selection of cylinder surface transformation are no longer active after the function has been deselected.

Procedure

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

Press the "Various" softkey.
3.

Press the "Transformations" and "Cylinder surface" softkeys.

Parameters	Description	Unit
Cylinder surface trans. 	• Yes Activates the cylinder surface transformation for programming. For several rotary axes, instead of "yes", the name of the rotary axis (A or B) is displayed. • No Deactivates cylinder surface transformation after programming.	
Transformation number	Selects the Tracyl data set, if there are several Tracyl data sets.	
⌀	Cylinder diameter - only for "Cylinder surface transformation yes"	
Slot side compensation 	• on - only for "Cylinder surface transformation yes" Activates slot side compensation. • off - only for "Cylinder surface transformation yes" Deactivates slot side compensation. The selection for "slot side compensation" depends on the transformation type.	
D	Offset to programmed path - only for "slot side compensation on"	

10.8.7 Straight or circular machining

When you want to perform straight or circular path movements or machining without defining a complete contour, you can use the functions "Straight line" or "Circle" respectively.

General sequence

To program simple machining operations, proceed as follows:

- Specify the tool and the spindle speed
- Program the machining operation

Machining options

The following machining options are available:

- Straight line
- Circle with known center point
- Circle with known radius
- Helix
- Straight line with polar coordinates
- Circle with polar coordinates

If you want to program a straight line or a circle using polar coordinates, you must define the pole first.

⚠ CAUTION

If you use a straight or circular path movement to move the tool into the retraction zone specified in the program header, you must also move the tool out again. Otherwise a collision could occur as a result of the traversing movements in a subsequently programmed cycle.

Before you can program a straight line or circle, you have to select the tool, spindle speed and machining plane.

If you program a sequence of different straight or circular path movements, the settings for the tool and spindle speed remain active until you change these again.

Procedure

1. The ShopMill program to be edited has been created and you are in the editor.

2. Press the menu forward key and the "Straight Circle" softkey.

3. Press the "Tool" softkey.
   The parameter screen "Tool" is opened.
4. Enter a tool in parameter field "T".
   - OR -

Select tool

Press the "Select tool" softkey.

The "Tool selection" window is opened.

To program

Position the cursor on the tool that you wish to use for machining and press the "To program" softkey.

The tool is copied into the "T" parameter field.

- OR -

Press the "Tool list" and "New tool" softkeys.

Using the softkeys on the vertical softkey bar, select the required tool and press the "To program" softkey.

The tool is copied into the "T" parameter field.

5. Select the tool cutting edge number D if the tool has several cutting edges.
6. Enter the spindle speed or cutting rate.
7. Enter an allowance in the "DR" field.

Accept

Press the "Accept" softkey.

The values are saved and the parameterization screen form is closed. The process plan is displayed and the newly generated program block is marked.

Parameter	Description	Unit
T	Tool name	
D	Cutting edge number	
S / V	Spindle speed or Constant cutting rate	rev/min m/min
DR	Allowance, tool radius	mm

10.8.8 Programming a straight line

The tool moves at the programmed feedrate or with rapid traverse from its actual position to the programmed end position.

Radius compensation

Alternatively, you can implement the straight line with radius compensation. The radius compensation acts modally, therefore you must deactivate the radius compensation again when you want to traverse without radius compensation. Where several straight line blocks with radius compensation are programmed sequentially, you may only select radius compensation in the first program block.

When executing the first path motion with radius compensation, the tool traverses without compensation at the starting point and with compensation at the end point. This means that if a vertical path is programmed, the tool traverses an oblique path. The compensation is not applied over the entire traversing path until the second programmed path motion with radius compensation is executed. The reverse effect occurs when radius compensation is deactivated.

Procedure

1. The ShopMill program to be edited has been created and you are in the editor.

2. Press the menu forward key and the "Straight Circle" softkey.

3. Press the "Straight line" softkey.

4. Press the "Rapid traverse" softkey to enter the feedrate in rapid traverse.

Parameter	Description		Unit
X	Target position X (abs) or target position X referred to the last programmed position (inc)		mm
Y	Target position Y (abs) or target position Y referred to the last programmed position (inc)		mm
Z	Target position Z (abs) or target position Z referred to the last programmed position (inc)		mm
	Note Incremental dimensions: The sign is also evaluated.		
F	Machining feedrate		mm/rev mm/min mm/tooth
Radius compensation	Input defining which side of the contour the cutter travels in the programmed direction:		
		Radius compensation to right of contour	
		Radius compensation to left of contour	
		Radius compensation off	
		The previously programmed setting for radius compensation is used.	

10.8.9 Programming a circle with known center point

The tool travels along a circular path from its actual position to the programmed circle end point. You must know the position of the circle center point. The control calculates the radius of the circle/arc on the basis of your interpolation parameter settings.

The circle can only be traversed at machining feedrate. You must program a tool before the circle can be traversed.

Procedure

1. The ShopMill program to be edited has been created and you are in the editor.
2. Press the menu forward key and the "Straight Circle" softkey.

3.

Press the "Circle center point" softkey.

Parameter	Description		Unit
Direction of rotation	The tool travels in the programmed direction from the circle starting point to its end point. You can program this direction as clockwise or counter-clockwise.		
	Clockwise direction of rotation		
	Counter-clockwise direction of rotation		
X	Target position X (abs) or target position X referred to the last programmed position (inc)		mm
Y	Target position Y (abs) or target position Y referred to the last programmed position (inc)		mm
I	Distance between circle starting point and center point in X direction (inc.)		mm
J	Distance between circle starting point and center point in Y direction (inc.)		mm
F	Machining feedrate		mm/rev mm/min mm/tooth
PL	Plane: The circle is traversed in the set plane with the relevant interpolation parameters: XYIJ: XY plane with interpolation parameters I and J ZXKI: ZX plane with interpolation parameters K and I YZJK: YZ plane with interpolation parameters J and K		 mm mm mm

10.8.10 Programming a circle with known radius

The tool traverses a circular path with the programmed radius from its actual position to the programmed circle end point. The control system calculates the circle center point. You do not need to program interpolation parameters.

The circle can only be traversed at machining feedrate.

Procedure

1. The ShopMill program to be edited has been created and you are in the editor.

2. Press the menu forward key and the "Straight Circle" softkey.

3. Press the "Circle radius" softkey.

Parameter	Description		Unit
Direction of rotation	The tool travels in the programmed direction from the circle starting point to its end point. You can program this direction as clockwise or counter-clockwise.		
		Clockwise direction of rotation	
		Counter-clockwise direction of rotation	
X	Target position X (abs) or target position X referred to the last programmed position (inc)		mm
Y	Target position Y (abs) or target position Y referred to the last programmed position (inc)		mm
R	Radius of arc. You can select the arc of your choice by entering a positive or a negative sign.		mm
F	Machining feedrate		mm/rev mm/min mm/tooth

10.8.11 Helix

With helical interpolation, a circular movement is overlaid in the plane with a linear motion in the tool axis, i.e. a spiral is created.

Procedure

1. The ShopMill program to be edited has been created and you are in the editor.

2. Press the menu forward key and the "Straight Circle" softkey.

3. Press the "Helix" softkey.

Parameter	Description		Unit
Direction of rotation	The tool travels in the programmed direction from the circle starting point to its end point. You can program this direction as clockwise or counterclockwise.		
	Clockwise direction of rotation		
	Counterclockwise direction of rotation		
I	Center point of the helix in the X direction (abs or inc)		mm
J	Center point of the helix in the Y direction (abs or inc)		mm
P	Helix pitch The pitch is programmed in mm per revolution.		mm/rev
Z	Target position of the helical end point (abs or inc)		mm
F	Machining feedrate		mm/rev mm/min mm/tooth

10.8.12 Polar coordinates

If a workpiece has been dimensioned from a central point (pole) with radius and angles, you will find it helpful to program these as polar coordinates.

You can program straight lines and circles as polar coordinates.

Defining a pole

You must define the pole before you can program a straight line or circle in polar coordinates. This pole acts as the reference point of the polar coordinate system.

The angle for the first line or circle then needs to be programmed in absolute coordinates. You can program the angles for any additional straight lines and circles as either absolute or incremental coordinates.

Procedure

1. The ShopMill program to be edited has been created and you are in the editor.

2. Press the menu forward key and the "Polar" softkey.

3. Press the "Pole" softkey.

Parameter	Description	Unit
X	Pole X (abs) or pole X referred to the last programmed position (inc)	mm
Y	Pole Y (abs) or pole Y referred to the last programmed position (inc)	mm

10.8.13 Straight polar

A straight line in the polar coordinate system is defined by a radius (L) and an angle (α). The angle refers to the X axis.

The tool moves from its actual position along a straight line to the programmed end point at the machining feedrate or in rapid traverse.

The 1st straight line in polar coordinates entered after the pole must be programmed with an absolute angle. You can program any additional straight lines or circles with incremental coordinates.

Procedure

1. The ShopMill program to be edited has been created and you are in the editor.

2. Press the menu forward key and the "Straight Circle" softkey.

3. Press the "Polar" and "Straight polar" softkeys.

4. Press the "Rapid traverse" softkey to enter the feedrate in rapid traverse.

Parameter	Description		Unit
L	Distance to the pole, end point		mm
α	Polar angle to the pole, end point (abs) or change in polar angle to the pole, end point (inc)		Degrees
F	Machining feedrate		mm/rev mm/min mm/tooth
Radius compensation	Input defining which side of the contour the cutter travels in the programmed direction:		
		Radius compensation to left of contour	
		Radius compensation to right of contour	
		Radius compensation off	
		The set radius compensation remains as previously set	

10.8.14 Circle polar

A circle in the polar coordinate system is defined by an angle (α). The angle refers to the X axis.

The tool moves from its actual position on a circular path to the programmed end point (angle) at the machining feedrate. The radius corresponds to the distance from the actual tool position to the defined pole, i.e. the circle starting and end point positions are at the same distance from the defined pole.

The 1st arc in polar coordinates entered after the pole must be programmed with an absolute angle. You can program any additional straight lines or circles with incremental coordinates.

Procedure

1. The ShopMill program to be edited has been created and you are in the editor.
2. Press the menu forward key and the "Straight Circle" softkey.

3. Press the "Polar" and "Circle polar" softkeys.

Parameter	Description		Unit
Direction of rotation	The tool travels in the programmed direction from the circle starting point to its end point. You can program this direction as clockwise (right) or counter-clockwise (left).		
		Clockwise direction of rotation	
		Counter-clockwise direction of rotation	
	Polar angle to the pole, end point (abs) or change in polar angle to the pole, end point (inc)		Degrees
F	Machining feedrate		mm/rev mm/min mm/tooth

10.8.15 Obstacle

Function

If there is an obstacle between 2 position patterns, it can be crossed. The height of the obstacle is programmed in absolute terms.

If all positions in the 1st pattern have been machined, the tool axis travels with rapid traverse to a height corresponding to the obstacle height + safety clearance. The new position is approached with rapid traverse at this height. The tool axis then approaches a position according to Z0 of the position pattern + safety clearance with rapid traverse.

Procedure

1. The ShopMill program to be edited has been created and you are in the editor.

2. Press the "Drilling" softkey.

3. Press the "Positions" and "Obstacle" softkeys.
   The "obstacle" input window opens.

Note

Obstacles are only taken into consideration if they lie between 2 position patterns. If the tool change point and the programmed retraction plane are positioned below the obstacle, the tool travels to the retraction plane height and on to the new position without taking the obstacle into account. The obstacle must not be higher than the retraction plane.

Parameter	Description	Unit
Z0	Obstacle height (abs)	

Multi-channel view 11

11.1 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

See also

Editor settings (Page 229)

11.2 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.

11.3 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.

11.4 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.

Collision avoidance

12

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

12.1 Activate collision avoidance

Precondition

- Collision avoidance is setup and an active machine model is available.
- The setting "Collision avoidance" has been selected for the AUTO operating mode or for the JOG and MDA operating modes.

Procedure

1. Select the "Machine" operating area.

2. Press the <AUTO> key.

3. Press the "Sim. rec." softkey.

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

During simultaneous recording, an active machine model is displayed.

12.2 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 43)

Tool management 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:

- acc. to the magazine
- acc. to the name (tool identifier, alphabetic)
- according to the tool type
- according to the T number (tool identifier, numerical)
- According to 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

Machine manufacturer

Please refer to the machine manufacturer's specifications.

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.

Note

Turning on milling machine

If you are working on a milling/turning machine, when creating tools, turning tools are also available in addition to drilling, milling and special tools.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Tool types

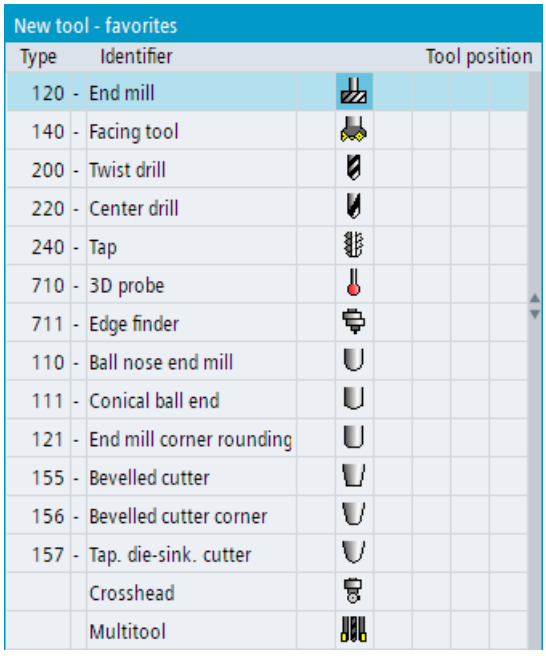
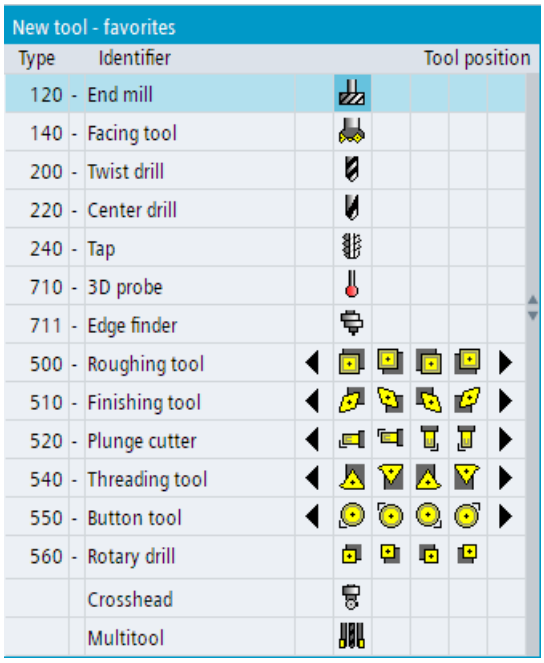


Figure 13-1 Favorites-standard selection for a milling machine



Favorites-standard selection for milling/turning machine

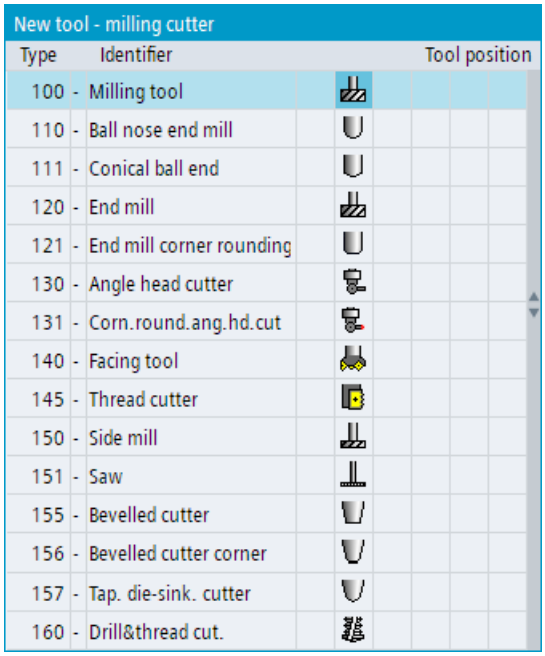


Figure 13-2 Available tools in the "New Tool - Milling Cutter" window

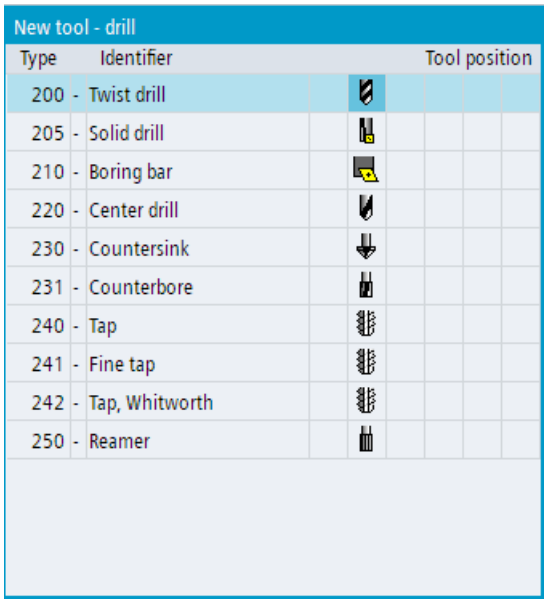


Figure 13-3 Available tools in the "New Tool - Drill" window

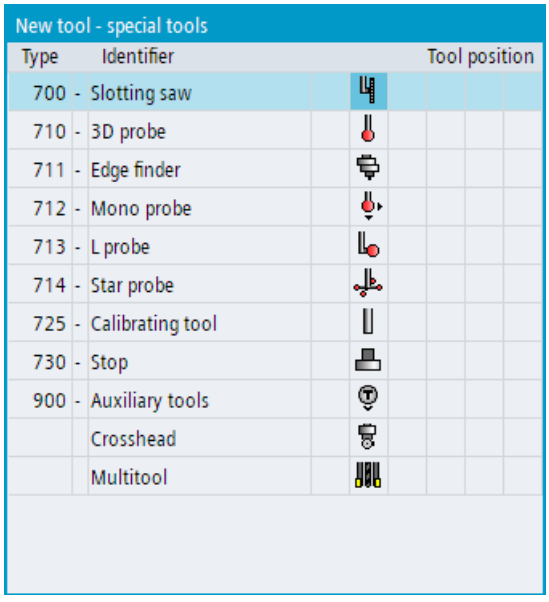


Figure 13-4 Available tools in the "New Tool - Special Tools" window

13.4 Tool dimensioning

This section provides an overview of the dimensioning of tools.

Tool types

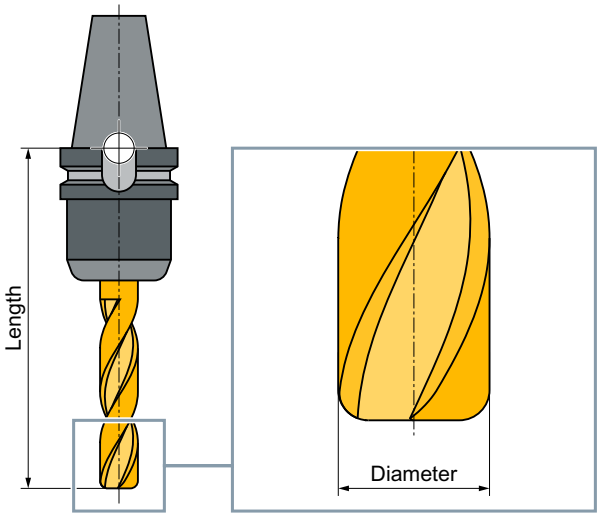


Figure 13-5 End mill (Type 120)

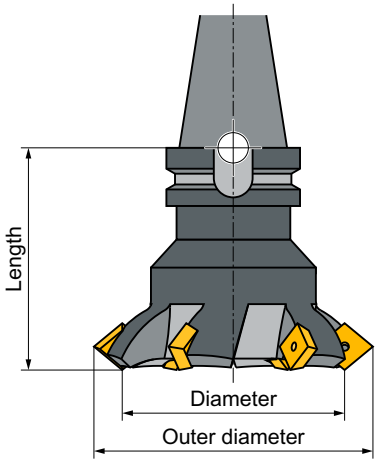


Figure 13-6 Face mill (Type 140)

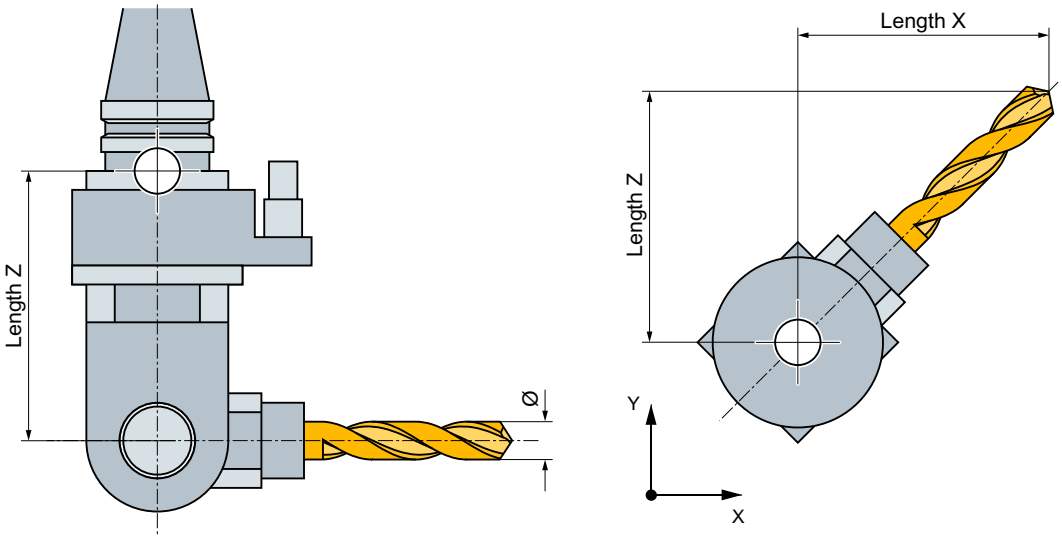


Figure 13-7 Angle head cutter (Type 130)

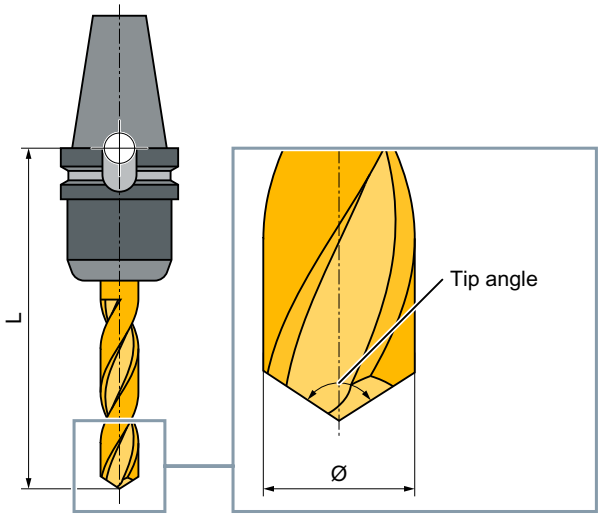


Figure 13-8 Drill (Type 200)

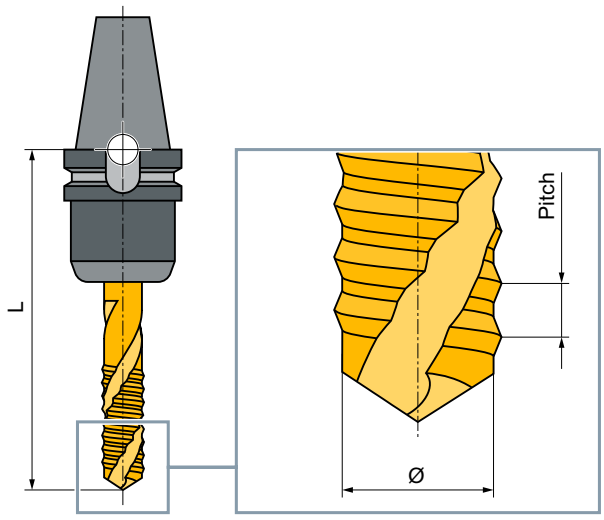


Figure 13-9 Tap (Type 240)

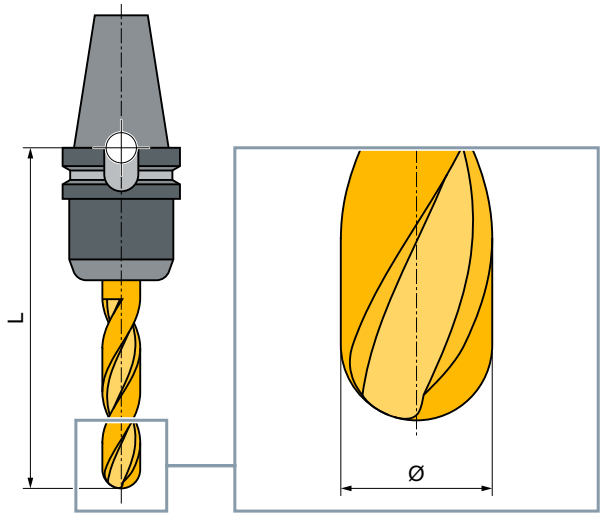


Figure 13-10 3D tool with an example of a cylindrical die-sinking cutter (Type 110)

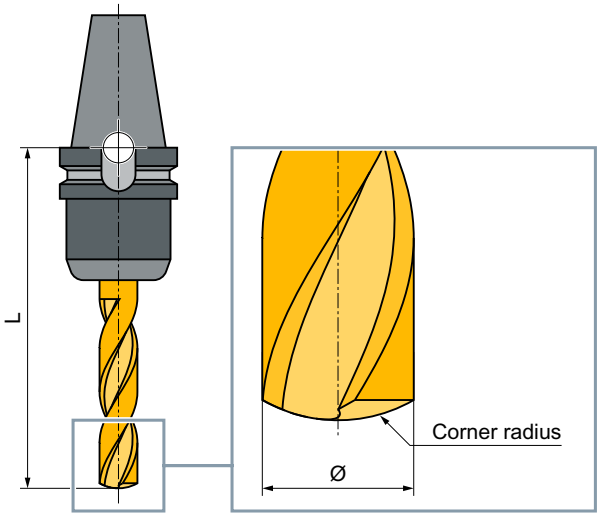


Figure 13-11 3D tool type with an example of a ballhead cutter (Type 111)

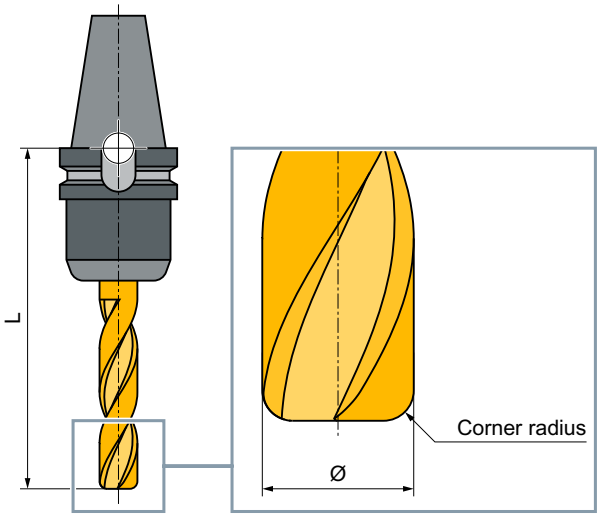


Figure 13-12 3D tool with an example of an end mill with corner rounding (Type 121)

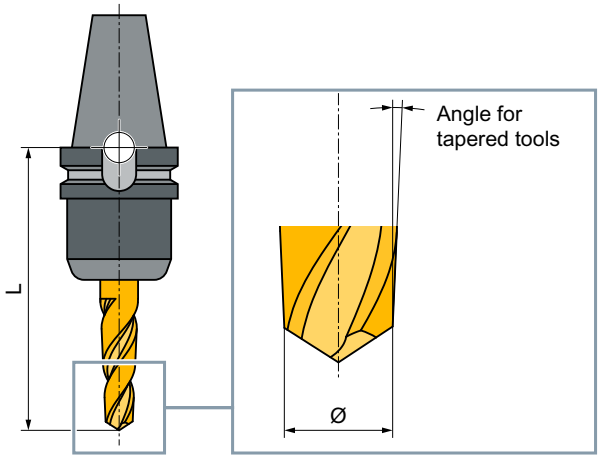


Figure 13-13 3D tool type with an example of a bevel cutter (Type 155)

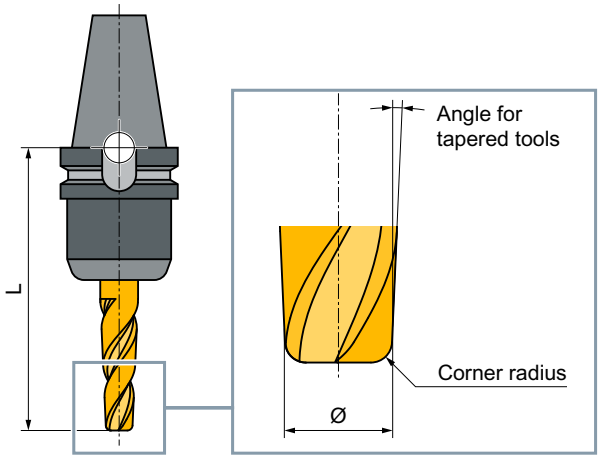


Figure 13-14 3D tool with an example of a bevel cutter with corner rounding (Type 156)

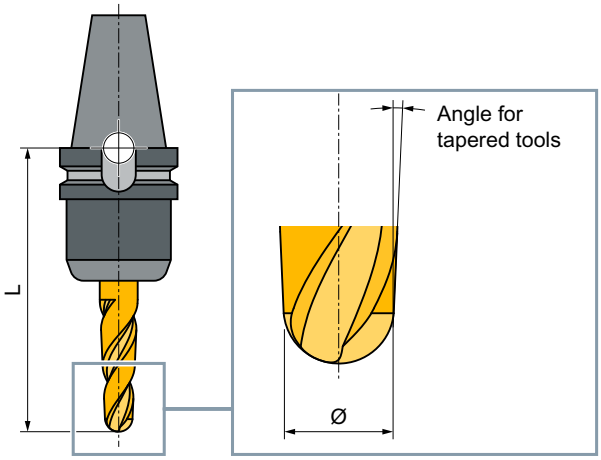
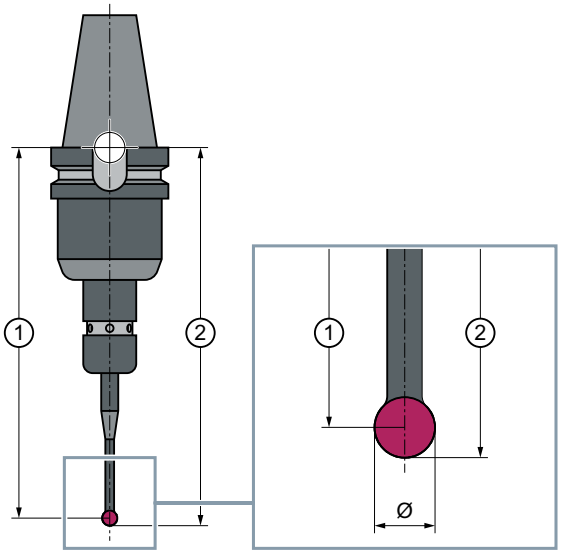


Figure 13-15 3D tool with an example of a tapered die-sinking cutter (Type 157)



① Length m
② Length u

Figure 13-16 3D probe

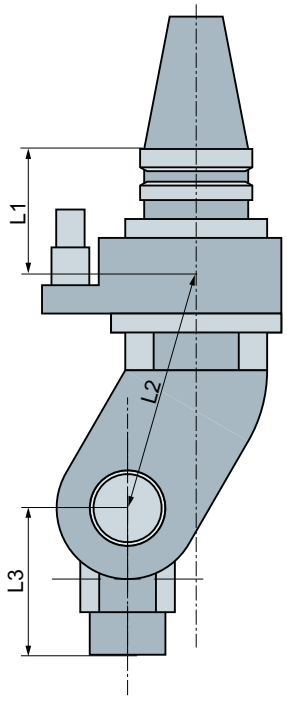


Figure 13-17 Crossover head adapter

L1, L2, L3 are offset geometry lengths.

Machine manufacturer

The tool length of the workpiece probe is measured to the center of the ball (length m) or to the ball circumference (length u).

Please observe the information provided by the machine manufacturer.

Note

An electronic workpiece probe must be calibrated before use.

13.5 Tool list

All parameters and functions that are required to create and set up the tools are displayed in the tool list.

Each tool is uniquely identified by the tool identifier and the sister tool number.

Tool parameters

Column heading	Meaning
Location BS * If activated in magazine selection	Magazine/location number • The magazine location numbers The magazine number is specified first, followed by the location number in the magazine. If there is only one magazine, only the location number is displayed. • Load position in the load magazine The following icons can also be displayed for other magazine types (e.g. for a chain): • Spindle location as an icon • Locations for gripper 1 and gripper 2 (applies only when a spindle with dual gripper is used) as icons.
Type	Tool type Specific tool offset data is displayed depending on the tool type (represented as an icon). For milling/turning machines, the icon identifies the position of the tool; this was selected when the tool was created.
	You have the option of changing the tool type or the tool position using the <SELECT> key.
Tool name	The tool is identified by the name and the sister tool number. You may enter the names as text or numbers. **Note**: The maximum length of tool names is 31 ASCII characters. The number of characters is reduced for Asian characters or Unicode characters. The following special characters are not permitted: | # ".
ST	Sister tool number (for sister tool strategy)
D	Cutting edge number
Length X, length Y, length Z	Tool length Geometry length length X, length Y, length Z
Radius	Tool radius
∅	Tool diameter

Column heading	Meaning
Width/ Tip width/ Tip angle / Pitch Drilling radius	Cutting edge for Type 150 - side milling cutter and Type 151 - saw Tip width for Type 520 - plunge cutter and Type 530 - cut-off tool Tip angle for Type 200 – twist drill and Type 220 – centering tool and Type 230 – countersink Pitch for Type 240 - tap Drilling radius for Type 560 - rotary drill. Holder angle and cutting tip angle are fixed.
   	Cutting edge graphic The cutting edge graphic shows the positioning defined by the holder angle, cut direction and cutting tip angle. Holder angle for Type 500 - rougher and Type 510 - finisher. The reference direction for the holder angle specifies the cut direction. In addition to the holder angle, the cutting tip angle is also specified.
N	Number of teeth for Type 100 - milling tool, Type 110 - ball end mill for cylindrical die-sinking cutter, Type 111 - ball end mill or tapered die-sinking cutter, Type 120 - end mill, Type 121 - end mill with corner rounding, Type 130 - angle head cutter, Type 131 - angle head cutter with corner rounding, Type 140 - facing tool, Type 150 - side mill, Type 155 - bevel cutter, Type 156 - bevel cutter with corner rounding and Type 157 - tapered die-sinking cutter.
Tip length	Tip length of a cutting tool or grooving cutter The tip length is required for displaying the tools during the simulation of the program processing.
	Direction of spindle rotation Spindle is not switched on CW spindle rotation CCW spindle rotation
	Coolants 1 and 2 (e.g. internal and external cooling) can be activated/deactivated. The coolant infeed at the machine does not necessarily have to be set-up.
M1 - M4	Other tool-specific functions such as additional coolant infeed, monitoring functions for speed, tool breakage, etc.

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.

Software option

In order to be able to manage the parameter spindle direction of rotation, coolant and tool-specific functions (M1-M4), you require the "ShopTurn/ShopMill" option.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Tool change / block search in ShopMill program step

If a tool is inserted by a tool change in a ShopMill program step, the corresponding M function for the coolant is automatically output.

After a block search, the coolant specified here is output even if a different coolant has been programmed since, e.g. via G code or via the "Machine functions" window.

More information

More information on the configuration and setup of the tool list can be found in the Tool Management Function Manual.

Icons in the tool list

Icon/ Marking		**Meaning**
Tool type		
Red "X"		The tool is disabled.
Yellow triangle pointing downward		The prewarning limit has been reached.
Yellow triangle pointing upward		The tool is in a special state. Place the cursor on the marked tool. A tooltip provides a short description.
Green border		The tool is preselected.
Magazine/location number		
Green double arrow		The magazine location is positioned at the change position.
Gray double arrow (configurable)		The magazine location is positioned at the loading position.
Red "X"		The magazine location is disabled.

Procedure

1. Select the "Parameter" operating area.

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

See also

Displaying tool details (Page 749)
Changing a tool type (Page 753)

13.5.1 Additional data

The following tool types require geometry data that is not included in the tool list display.

Tools with additional geometry data

Tool type	Additional parameters
111 Conical ballhead cutter	Corner radius
121 End mill with corner rounding	Corner radius
130 Angle head cutter	Geometry length (length X, length Y, length Z) Wear length (Δ length X, Δ length Y, Δ length Z) Adapter length (length X, length Y, length Z) V (direction vector 1 - 6) Vector X, vector Y, vector Z
131 Angle head cutter with corner rounding	Geometry length (length X, length Y, length Z) Corner radius Wear length (Δ length X, Δ length Y, Δ length Z) Adapter length (length X, length Y, length Z) V (direction vector 1 - 6) Vector X, vector Y, vector Z
140 Face milling	External radius Tool angle
155 Bevel cutter	Taper angle
156 Bevel cutter with corner rounding	Corner radius Taper angle
157 Conical die-milling cutter	Taper angle
585 Calibration tool	Geometry length (length X, length Y, length Z) Wear length (Δ length X, Δ length Y, Δ length Z)

Tool type	Additional parameters
700 Slotting saw	Geometry length (length X, length Y, length Z) Wear length (Δ length X, Δ length Y, Δ length Z) Adapter length (length X, length Y, length Z) Geometry (slot width, projection) Wear (slot width, projection)
710 3D-probe milling	Geometry length (length X, length Y, length Z) Wear length (Δ length X, Δ length Y, Δ length Z)
712 Mono-probe	Geometry length (length X, length Y, length Z) Wear length (Δ length X, Δ length Y, Δ length Z)
713 L-probe	Geometry length (length X, length Y, length Z) Wear length (Δ length X, Δ length Y, Δ length Z) Boom length (length)
714 Star-type probe	Geometry length (length X, length Y, length Z) Wear length (Δ length X, Δ length Y, Δ length Z) Outer diameter (∅)
Crossover head	V1 - V3: Direction vectors L1 - L3: Offset geometry length L2A, L2B: Geometry lengths

You can use the configuration file to specify the data to be displayed for specific tool types in the "Additional Data" window.

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

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.

13.5.2 Creating a new tool

When creating a new tool, the "New tool - favorites" window offers you a number of selected tool types, known as "favorites".

If you do not find the desired tool type in the favorites list, then select the milling, drilling or special tool using the corresponding softkeys.

Procedure

1. The tool list is opened.

2. Place the cursor in the tool list at the position where the new tool should be stored.
   For this, you can select an empty magazine location or the NC tool memory outside of the magazine.
   You may also position the cursor on an existing tool in the area of the NC tool memory. Data from the displayed tool will not be overwritten.

3. Press the "New tool" softkey.

The "New tool - favorites" window opens.

- OR -

If you want to create a tool that is not in the "Favorites" list, press the softkey "Cutters 100-199", "Drill 200-299", or "Spec.tool 700-900".

...

The "New tool - milling cutter", "New tool - drill", or "New tool - special tools" window opens.

4. Select the tool by placing the cursor on the corresponding icon.

5. Press the "OK" softkey.
   The tool is added to the tool list with a predefined name. If the cursor is located on an empty magazine location in the tool list, then the tool is loaded to this magazine location.

The tool creation sequence can be defined differently.

Multiple load points

If you have configured several loading points for a magazine, then the "Select loading point" window appears when a tool is created directly in an empty magazine location or when the "Load" softkey is pressed.

Select the required load point and confirm with the "OK" softkey.

Additional data

If configured accordingly, the "New tool" window opens after the required tool has been selected and confirmed with "OK".

You can define the following data in this window:

- Names
- Tool location type
- Size of tool

More information

More information on the configuration options can be found in the Tool Management Function Manual.

13.5.3 Measuring the tool

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 is opened.

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 "T" field in the "Length Manual" screen.

3. Select the cutting edge number D and the number of the replacement tool ST of the tool.

4. Approach the workpiece in the Z direction, scratch it with a turning spindle and enter the set position Z0 of the workpiece edge.

5. Press the "Set length" softkey.

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

13.5.4 Managing several cutting edges

In the case of tools with more than one cutting edge, a separate set of offset data is assigned to each cutting edge. The number of possible cutting edges depends on the controller configuration.

Tool cutting edges that are not required can be deleted.

Procedure

1. The tool list is opened.

2. Position the cursor on the tool for which you would like to store more cutting edges.

3. Press the "Edges" softkey in the "Tool list".

4. Press the "New cutting edge" softkey.
   A new data set is stored in the list.
   The cutting edge number is incremented by one and the offset data is assigned the values of the cutting edge on which the cursor is positioned.
5. Enter the offset data for the 2nd cutting edge.
6. Repeat this process if you wish to create more tool edge offset data.

7. Position the cursor on the cutting edge that you want to delete and press the "Delete cutting edge" softkey.
   The data set is deleted from the list. The first tool cutting edge cannot be deleted.

13.5.5 Delete 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.

13.5.6 Loading and unloading tools

You can load and unload tools to and from a magazine via the tool list. When a tool is loaded, it is taken to a magazine location. When it is unloaded, it is removed from the magazine and stored in the NC memory.

When you are loading a tool, the application automatically suggests an empty location. You may also directly specify an empty magazine location.

You can unload tools from the magazine that you are not using at present. HMI then automatically saves the tool data in the NC memory.

Should you want to use the tool again later, simply load the tool with the tool data into the corresponding magazine location again. Then the same tool data does not have to be entered more than once.

Procedure

1. The tool list is opened.
2. Place the cursor on the tool that you want to load into the magazine (if the tools are sorted according to magazine location number you will find it at the end of the tool list).

3. Press the "Load" softkey.

   The "Load to... " window opens.

   The "... Location" field is defaulted with the number of the first empty magazine location.

4. Press the "OK" softkey to load the tool into the suggested location.

   - OR -

   Enter the location number you require and press the "OK" softkey.

   - OR -

   Press the "Spindle" softkey.

   The tool is loaded into the specified magazine location or spindle.

Loading empty magazine location directly with tool

1. Position the cursor at an empty magazine location where you want to load a tool and press the "Load" softkey.

   The "Load with ..." window opens.

   Select the desired tool in the " ... Tool" field and press the "OK" softkey.

Several magazines

If you have configured several magazines, the "Load to ..." window appears after pressing the "Load" softkey.

If you do not want to use the suggested empty location, then enter your desired magazine and magazine location. Confirm your selection with "OK".

Multiple load points

If you have configured several loading points for a magazine, then the "Loading Point Selection" window appears after pressing the "Load" softkey.

Select the required loading point and confirm with "OK".

Unloading tools

1. Place the cursor on the tool that you would like to unload from the magazine and press the "Unload" softkey.
2. Select the required load point in the "Loading Point Selection" window.

3. Confirm your selection with "OK".

- OR -

Undo your selection with "Cancel".

13.5.7 Selecting a magazine

You can directly select the buffer memory, the magazine, or the NC memory.

Procedure

1. The tool list is opened.

2. Press the "Magazine selection" softkey.

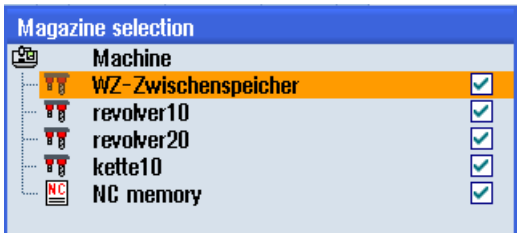
If there is only one magazine, you will move from one area to the next (i.e. from the buffer memory to the magazine, from the magazine to the NC memory, and from the NC memory back to the buffer memory) each time you press the softkey. The cursor is positioned at the beginning of the magazine each time.

- OR -

If there is more than one magazine, the "Magazine Selection" window opens. Position the cursor on the desired magazine in this window and press the "Go to" softkey.

The cursor jumps directly to the beginning of the specified magazine.

Hiding magazines



Deactivate the checkbox next to the magazines that you do not want to appear in the magazine list.

The magazine selection behavior with multiple magazines can be configured in different ways.

Machine manufacturer

Please observe the information provided by the machine manufacturer.

Further information

Additional information on the configuring options is provided in the Tool Management Function Manual.

13.5.8 Code carrier connection

You have the option of configuring a code carrier connection.

This means that the following functions are available in SINUMERIK Operate:

- Creating a new tool from code carrier
- Unloading tools on code carrier

Software option

In order to use the functions, you require the option "Tool Ident Connection".

Further information

Additional information on tool management with code carrier and the configuration of the user interface in SINUMERIK Operate is provided in the Tool Management Function Manual.

With a code carrier connection, in the list of favorites, there is also a tool available.

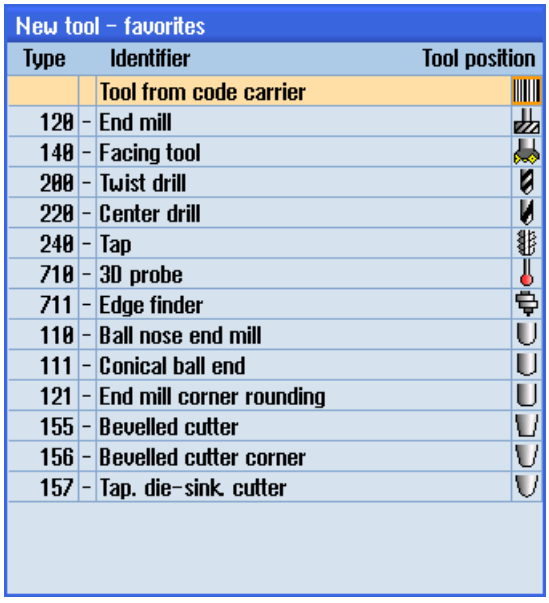


Figure 13-18 New tool from code carrier in the list of favorites

Creating a new tool from code carrier

1. The tool list is opened.

2. Place the cursor in the tool list at the position where the new tool should be created.

   To do this, you can select an empty magazine location or the NC tool memory outside of the magazine.

   You may also position the cursor on an existing tool in the area of the NC tool memory. Data from the displayed tool will not be overwritten.

3. Press the "New tool" softkey.

   The "New Tool - Favorites" window is opened.

4. Position the cursor on the entry "Tool from code carrier" and press the "OK" softkey.

   The tool data is read from the code carrier, and is displayed in the "New tool" window with the tool type, tool name and possibly with certain parameters.

5. Press the "OK" softkey.

   The tool is added to the tool list with the specified name. If the cursor is located on an empty magazine location in the tool list, then the tool is loaded to this magazine location.

The tool creation sequence can be defined differently.

Unloading tool on code carrier

1. The tool list is opened.
2. Place the cursor on the tool that you would like to unload from the magazine and press the "Unload" and "On code carrier" softkeys.
   The tool is unloaded and the data of the tool are written to the code carrier.

According to the corresponding setting, the unloaded tool on the code carrier is deleted from the NC memory after reading out the code carrier.

Deleting tool on code carrier

1. The tool list is opened.
2. Position the cursor on the tool on the code carrier that you want to delete.
3. Press the "Delete tool" and "On code carrier" softkeys.
   The tool is unloaded and the data of the tool are written to the code carrier. The tool is then deleted from the NC memory.

The deletion of the tool can be set differently, i.e. the "On code carrier" softkey is not available.

13.5.9 Managing a tool in a file

If the "Enable tool in/out file" option is activated in the settings for the tool list, an additional entry is available in the list of favorites.

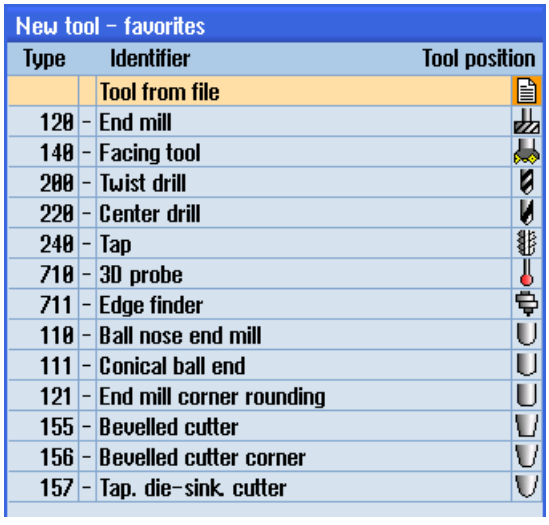


Figure 13-19 New tool from file in the list of favorites

Creating a new tool from a file

1. The tool list is open.

2. Place the cursor in the tool list at the position where the new tool should be created.
   To do this, you can select an empty magazine location or the NC tool memory outside of the magazine.
   You may also position the cursor on an existing tool in the area of the NC tool memory. Data from the displayed tool will not be overwritten.

3. Press the "New tool" softkey.

The "New Tool - Favorites" window is opened.

4. Position the cursor on the entry "Tool from file" and press the "OK" softkey.
   The "Load Tool Data" window opens.

5. Navigate to the required file and press the "OK" softkey.
   The tool data is read from the file, and is displayed in the "New Tool from File" window with the tool type, tool name and possibly with certain parameters.

6. Press the "OK" softkey.
   The tool is added to the tool list with the specified name. If the cursor is located on an empty magazine location in the tool list, then the tool is loaded to this magazine location.

The tool creation sequence can be defined differently.

Unloading a tool in a file

1. The tool list is open.

2. Place the cursor on the tool that you would like to unload from the magazine and press the "Unload" and "In file" softkeys.

3. Navigate to the required directory and press the "OK" softkey.

4. Enter the required file name in the "Name" field and press the "OK" softkey.
   The field is preassigned with tool names.
   The tool is unloaded and the tool data is written to the file.

According to the corresponding setting, the unloaded tool is deleted from the NC memory after it has been read out.

Deleting a tool in a file

1. The tool list is open.
2. Position the cursor on the tool that you wish to delete.

3. Press the "Delete tool" and "In file" softkeys.

3. Navigate to the required directory and press the "OK" softkey.

4. Enter the required file name in the "Name" field and press the "OK" softkey.
   The field is preassigned with tool names.
   The tool is unloaded and the tool data is written to the file. The tool is then deleted from the NC memory.

13.6 Tool wear

All parameters and functions that are required during operation are contained in the tool wear list.

Tools that are in use for long periods are subject to wear. You can measure this wear and enter it in the tool wear list. The controller then takes this information into account when calculating the tool length or radius compensation. This ensures a consistent level of accuracy during workpiece machining.

Monitoring types

You can automatically monitor the tools' working times via the workpiece count, tool life or wear.

Note

Combination of monitoring types

You have the option to activate the monitoring of a tool by type or any combination of monitoring types.

In addition, you can disable tools when you no longer wish to use them.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Tool parameters

Column heading	Meaning
Location BS * If activated in magazine selection	Magazine/location number • The magazine location numbers The magazine number is specified first, followed by the location number in the magazine. If there is only one magazine, only the location number is displayed. • Load position in the load magazine The following icons can also be displayed for other magazine types (e.g. for a chain): • Spindle location as an icon • Locations for gripper 1 and gripper 2 (applies only when a spindle with dual gripper is used) as icons.
Type	Tool type Depending on the tool type (represented by an icon), certain tool offset data is enabled.

Column heading	Meaning
Tool name	The tool is identified by the name and the replacement tool number. You can enter the name as text or number. **Note**: The maximum length of tool names is 31 ASCII characters. The number of characters is reduced for Asian characters or Unicode characters. The following special characters are not permitted: | # ".
ST	Replacement tool number (for replacement tool strategy).
D	Cutting edge number
Δ Length	Length wear
Δ Radius	Radius wear
T C	Selection of tool monitoring - by tool life (T) - by count (C) - by wear (W) The wear monitoring is configured via a machine data item. Please refer to the machine manufacturer's instructions.
Tool life Workpiece count Wear * *Parameter depends on selection in TC	Tool life Number of workpieces Tool wear
Setpoint	Setpoint for tool life, workpiece count, or wear
Prewarning limit	Specification of the tool life, workpiece count or wear at which a warning is displayed.
G	The tool is disabled when the checkbox is selected.

Further parameters

If you have created unique cutting edge numbers, they will be displayed in the first column.

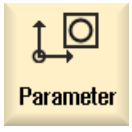
Column heading	Meaning
D no.	Unique cutting edge number
SN	Cutting edge number
SC 	Setting-up offsets Display of the available setup offsets

Icons in the wear list

Icon/ Marking		Meaning
Tool type		
Red "X"		The tool is disabled.
Yellow triangle pointing downward		The prewarning limit has been reached.

Icon/ Marking		Meaning
Yellow triangle pointing upward		The tool is in a special state. Place the cursor on the marked tool. A tooltip provides a short description.
Green border		The tool is preselected.
Magazine/location number		
Green double arrow		The magazine location is positioned at the change position.
Gray double arrow (configurable)		The magazine location is positioned at the loading position.
Red "X"		The magazine location is disabled.

Procedure



1. Select the "Parameter" operating area.

2. Press the "Tool wear" softkey.

See also

Displaying tool details (Page 749)

Changing a tool type (Page 753)

13.6.1 Reactivating a tool

You can replace disabled tools or make them ready for use again.

Requirements

In order to reactivate a tool, the monitoring function must be activated and a setpoint must be stored.

Procedure

1. The tool wear list is opened.
2. Position the cursor on the disabled tool which you would like to reuse.

3. Press the "Reactivate" softkey.

   The value entered as the setpoint is entered as the new tool life or workpiece count.

   The disabling of the tool is canceled.

Reactivating and positioning

When the "Reactivate with positioning" function is configured, the selected tool's magazine location will also be positioned at the loading point. You can exchange the tool.

Reactivation of all monitoring types

When the "Reactivation of all monitoring types" function is configured, all the monitoring types set in the NC for a tool are reset during reactivation.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Multiple load points

If you have configured several loading points for a magazine, then the "Loading Point Selection" window appears after pressing the "Load" softkey.

Select the required load point and confirm with the "OK" softkey.

Further information

Additional information on the configuring options is provided in the Tool Management Function Manual.

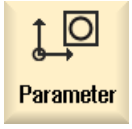
13.7 Tool data OEM

You have the option of configuring the list according to your requirements.

More information

More information on configuring OEM tool data can be found in the Tool Management Function Manual.

Procedure



1. Select the "Parameter" operating area.

2. Press the "OEM tool" softkey.

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 BS * If activated in magazine selection	Magazine/location number • The magazine location numbers The magazine number is specified first, followed by the location number in the magazine. If there is only one magazine, only the location number is displayed. • Load position in the load magazine The following icons can also be displayed for other magazine types (e.g. for a chain): • Spindle location as an icon • Locations for gripper 1 and gripper 2 (applies only when a spindle with dual gripper is used) as icons
Type	Tool type Depending on the tool type (represented by an icon), certain tool offset data is enabled.
Tool name	The tool is identified by the name and the replacement tool number. You can enter the name as text or number. **Note**: The maximum length of tool names is 31 ASCII characters. The number of characters is reduced for Asian characters or Unicode characters. The following special characters are not permitted: | # ".
ST	Replacement tool number (for replacement tool strategy).
D	Cutting edge number
G	Disabling of the magazine location.
Mag.loc. type	Display of magazine location type.
Tool.loc. type	Display of tool location type.
Ü	Marking of a tool as oversized. The tool occupies two half locations left, two half locations right, one half location top and one half location bottom in a magazine.
P	Fixed location coding. The tool is permanently assigned to this magazine location.

Further parameters

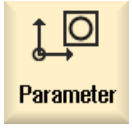
If you have created unique cutting edge numbers, they will be displayed in the first column.

Column heading	Meaning
D no.	Unique cutting edge number
SN	Cutting edge number

Magazine list icons

Icon/ Marking		Meaning
Tool type		
Red "X"		The tool is disabled.
Yellow triangle pointing downward		The prewarning limit has been reached.
Yellow triangle pointing upward		The tool is in a special state. Place the cursor on the marked tool. A tooltip provides a short description.
Green border		The tool is preselected.
Magazine/location number		
Green double arrow		The magazine location is positioned at the change position.
Gray double arrow (configurable)		The magazine location is positioned at the loading position.
Red "X"		The magazine location is disabled.

Procedure



1. Select the "Parameter" operating area.

2. Press the "Magazine" softkey.

See also

Displaying tool details (Page 749)

Changing a tool type (Page 753)

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.

Buffer

You have the option of relocating the tool to buffer locations.

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

Procedure

1. The magazine list is opened.
2. Position the cursor on the tool that you wish to relocate to a different magazine location.

3. Press the "Relocate" softkey.

   The "... relocate from location ... to location ..." window is displayed. The "Location" field is pre-assigned with the number of the first empty magazine location.

4. Press the "OK" softkey to relocate the tool to the recommended magazine location.

   - OR -

   Enter the required magazine, enter the location number and press the "OK" softkey.

   - OR -

   Enter the number "9998" or the number "9999" into the "... magazine" field in order to select the buffer as well as the required buffer location in the "Location" field.

   - OR -

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

   The tool is relocated to the specified magazine location, in the spindle or in the buffer.

Several magazines

If you have set up several magazines, then the "...relocate from magazine... location... to..." window appears after pressing the "Relocate" softkey.

Select the desired magazine and location, and confirm your selection with "OK" to load the tool.

13.8.3 Deleting / unloading / loading / relocating all tools

You have the option of deleting or unloading all tools in the magazine list, loading them into the magazine list or relocating them in the magazine list. With one task, the tools are deleted or unloaded from the list or loaded, relocated in the list.

Requirement

The following requirements must be satisfied so that the "Delete all", "Unload all", "Load all" or "Relocate all" softkey is displayed and available:

- Magazine management is set up
- There is no tool in the buffer / in the spindle

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure

1. The magazine list is open.

2. Press the "Delete all" softkey.

   - OR -

   Press the "Unload all" softkey.

   - OR -

   Press the "Load all" softkey.

   - OR -

   Press the "Relocate all" softkey.

   A prompt is displayed as to whether you really want to delete, unload, load or relocate all tools.

3. Press the "OK" softkey to continue with deleting, unloading, loading or relocation of the tools.

   The tools are deleted, unloaded, loaded or relocated in the magazine in ascending magazine location number order.

4. Press the "Cancel" softkey if you wish to cancel the unloading operation.

Multiple load points

For a magazine, if more than one loading point was set-up, using the "Select loading point" softkey, you have the option of opening a window in which you can assign a loading point to a magazine.

13.9 Tool details

13.9.1 Displaying tool details

The following parameters of the selected tool can be displayed using softkeys in the "Tool Details" window.

- Tool data
- Cutting edge data
- Monitoring data

Procedure

1. The tool list, the wear list, the OEM tool list or the magazine is 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" softkeys.

- OR -

If you are in the wear list or OEM tool list, press the "Details" softkey.

The "Tool Details" window opens.

All the available tool data are displayed.

4. Press the "Cutting edge data" softkey if you want to display the cutting data.

5. Press the "Monitoring data" softkey if you want to display the monitoring data.

13.9.2 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	

13.9.3 Cutting edge data

The "Tool Details" window provides the following data on the selected tool when the "Cutting edge data" softkey is active.

Parameter	Meaning
Magazine location	The magazine number is specified first, followed by the location number in the magazine. If there is only one magazine, only the location number is displayed.
Tool name	The tool is identified by the name and the sister tool number. You can enter the name as text or number.
ST	Replacement tool number (for replacement tool strategy)
D quantity	Number of created cutting edges
D	Cutting edge number
Tool type	Tool symbol with type number

Parameter		Meaning
		Length
Geometry		Tool length
Wear		Tool wear
		Ø (diameter)
Geometry		Tool diameter
Wear		Tool wear, diameter
Type 240 - tap		
Pitch		Height of the developed helix parallel to the screw axis
Type 200 - twist drill, type 220 - centering tool and Type 230 - countersink		
Tip angle		Angle is less than 180°
Type 520 -plunge cutter, type 530 - parting tool, type 540 - threading tool		
Cutting tip length		For displaying the tools during the simulation of the program execution.
Cutting tip width		Width of the plunge cutter
Type 110 - ball end mill for cylindrical die-sinking cutter, type 111 - ball end mill for tapered die-sinking cutter, type 120 - end mill, type 121 - end mill with corner rounding, type 130 - angle head cutter, type 140 - facing tool, type 150 - side mill, type 155 - bevel cutter, type 156 - bevel cutter with corner rounding and type 157 - tapered die-sinking cutter		
N		Number of teeth
For driven tools (drills and milling tools)		
Direction of spindle rotation		Spindle is not switched on
		CW spindle rotation
		CCW spindle rotation
		Coolant 1 and 2 (e.g. internal and external cooling) can be switched on and off. Please refer to the machine manufacturer's specifications
Cutting edge OEM parameters 1 - 2		

Software option

In order to be able to manage the parameter spindle direction of rotation, coolant and tool-specific functions (M1-M4), you require the "ShopMill/ShopTurn" option.

13.9.4 Monitoring data

The "Tool Details" window provides the following data on the selected tool when the "Monitoring data" softkey is active.

Parameter	Meaning
Magazine location	The magazine number is specified first, followed by the location number in the magazine. If there is only one magazine, only the location number is displayed.
Tool name	The tool is identified by the name and the sister tool number. You can enter the name as text or number.
ST	Sister tool number (for sister tool strategy)
D quantity	Number of created cutting edges
D	Cutting edge number
Monitoring type 	T - tool life C - count W - wear The wear monitoring is configured via machine data. Please note the specifications of the machine manufacturer.
	Actual value
Tool life, count and wear	Actual value for tool life, count or wear
	Setpoint
Tool life, count and wear	Setpoint for tool life, count or wear
	Prewarning limit
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	

13.10 Changing a tool type

Procedure

1. The tool list, the wear list, the OEM tool list or the magazine is opened.

...

2. Position the cursor in the column "Type" of the tool that you wish to change.

3. Press the <SELECT> key.

   The "Tool types - Favorites" window opens.

4. Select the desired tool type in the list of favorites or select using the softkeys "Cutters 100-199", "Drill 200-299", or "Spec. tool 700-900".

5. Press the "OK" softkey.

   The new tool type is accepted and the corresponding icon is displayed in the "Type" column.

13.11 Graphic display

In addition to the list of tools, you can also display the tools and magazine locations in a dynamic graphic display.

The tools are displayed in the list in the order with their correct proportions.

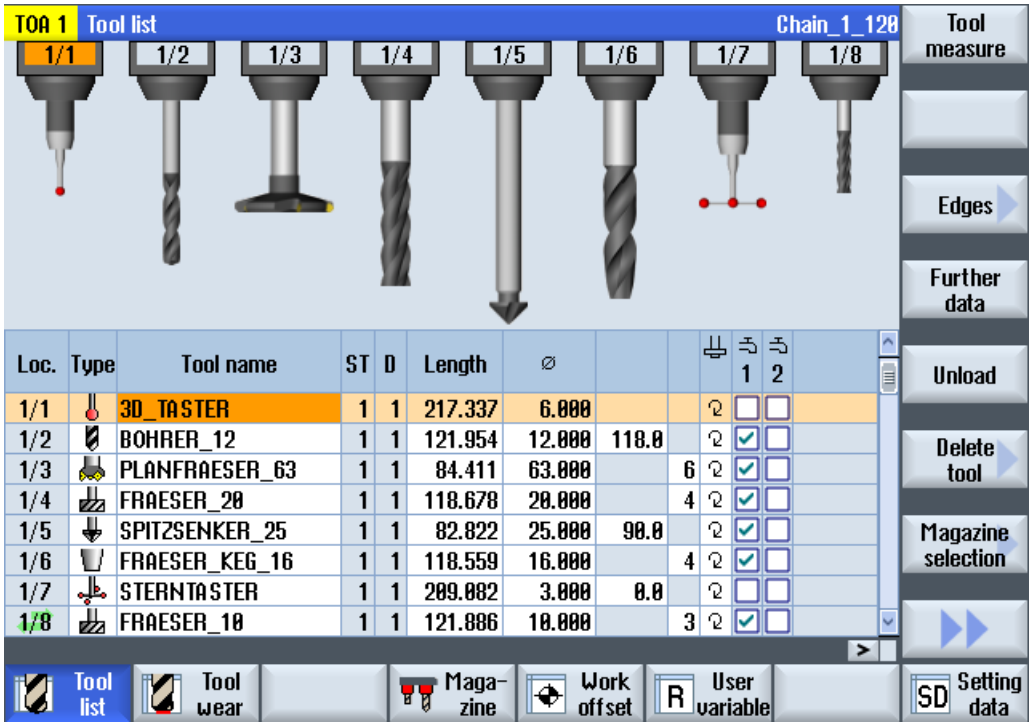
The graphic display must be set up by the machine manufacturer.

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

More information

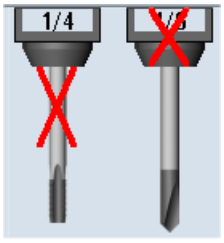
More information on the configuration can be found in the Tool Management Function Manual.

Graphic display of tools and magazine locations



The following applies to the graphic display:

- If a tool is too long for the display, the maximum possible length is shown.
- Oversized tools are trimmed on the left and right sides.
- Tools that are not located in the magazine are displayed without toolholder.
- Disabled tools or magazine locations are marked by means of a red cross:



Note

Measuring tools type 713 / 714

So that the tools "L button" and "star probe" are displayed in the graphic tool display, enter in the "More data" window the additional Parameter "Boom length" or "External diameter".

Switching the graphical magazine display on/off

1. The tool list or wear or magazine list is opened.
2. Press the "Continue" and "Settings" softkeys.

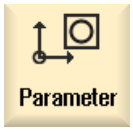
The "Settings" window appears.

3. Activate the "Switch on graphic magazine display" check box in order to change to display of the lists in graphic form.

13.12 Sorting tool management lists

When you are working with many tools, with large magazines or several magazines, it is useful to display the tools sorted according to different criteria. Then you will be able to find a specific tool more easily in the lists.

Procedure



1. Select the "Parameter" operating area.

2. Press the "Tool list", "Tool wear" or "Magazine" softkey.

...

3. Press the ">>" and "Sort" softkeys.

The lists are displayed sorted numerically according to magazine location.

Tool types are used to sort tools with the same magazine location. Identical types (e.g. milling cutters), in turn, are sorted according to their radius value.

4. Press the "Acc. to type" softkey to display the tools arranged by tool type. Identical types (e.g. milling cutters) are sorted according to their radius value.

- OR -

Press the "Acc. to name" softkey to display the tool names in alphabetical order.

The replacement tool numbers are used to sort tools with the same names.

- OR -

Press the "Acc. to T number" softkey to display the tool names sorted numerically.

- OR -

Press the "Acc. to D number" softkey to display the tools sorted by D number.

The list is sorted according to the specified criteria.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

13.13 Filtering the tool management lists

The filter function allows you to filter-out tools with specific properties in the tool management lists.

For instance, you have the option of displaying tools during machining that have already reached the prewarning limit in order to prepare the corresponding tools to be loaded.

Filter criteria

- Only display the first cutting edge
- Only tools that are ready to use
- Only tools with active code
- Only locked tools
- Only tools that have reached the prewarning limit
- Only tools with remaining quantity of ... to ...
- Only tools with residual tool life of ... to ...
- Only tools with unloading marking
- Only tools with loading marking

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

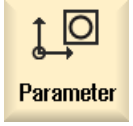
Note

Multiple selection

You have the option of selecting several criteria. You will receive an appropriate message if conflicting filter options are selected.

You can configure OR logic operations for the various filter criteria.

Procedure



1. Select the "Parameter" operating area.

2. Press the "Tool list", "Tool wear" or "Magazine" softkey.

...

3. Press the ">>" and "Filter" softkeys.
   The "Filter" window opens.

4. Activate the required filter criterion and press the "OK" softkey.
   The tools that correspond to the selection criteria are displayed in the list.
   The active filter is displayed in the window header.

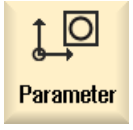
13.14 Specific search in the tool management lists

There is a search function in all tool management lists, where you can search for the following objects:

- **Tools**
  - You enter a tool name. You can narrow down your search by entering a replacement tool number.
    You have the option of only entering a part of the name as search term.
  - Enter the D number and, if necessary, activate the "Active D number" checkbox.
- **Magazine locations or magazines**
  If only one magazine is configured, then the search is only made for the magazine location.
  If several magazines are configured, then it is possible to search a specific magazine location in a specific magazine or just to search in a specific magazine.
- **Empty locations**
  If the lists with the location type are used, then the empty location search is made using the location type and location size.

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

Procedure



1. Select the "Parameter" operating area.

2. Press the "Tool list", "Tool wear" or "Magazine" softkey.

...

3. Press the ">>" and "Search" softkeys.

...

4. Press the "Tool" softkey if you wish to search for a specific tool.

   - OR -

   Press the "Magazine location" softkey if you wish to search for a specific magazine location or a specific magazine.

- OR -

Press the "Empty location" softkey if you wish to search for a specific empty location.

13.15 Settings for tool lists

The "Settings" window provides the following options to set the view in the tool lists:

- Display only one magazine in the magazine sorting
  - You limit the display to one magazine. The magazine is displayed with the assigned buffer locations and the not-loaded tools.
  - You use a configuration to specify whether the "Magazine selection" softkey jumps to the next magazine or whether the "Magazine Selection" dialog for changing switches to any magazine.
- Display only spindles in the buffer
  In order to display during operation only the spindle location, the remaining locations of the buffer will be hidden.
- Enable tool in/out file
  - When creating a new tool, the tool data can be loaded from a file.
  - When deleting or unloading a tool, the tool data can be backed up in a file.
- Activating the adapter-transformed view
  - The geometry lengths and the operation offsets are displayed transformed in the tool list.
  - The wear lengths and the sum offsets are displayed transformed in the tool wear list.

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

Procedure

1. Select the "Parameter" operating area.

2. Press the "Tool list", "Tool wear" or "Magazine" softkey.

...

3. Press the "Continue" and "Settings" softkeys.

4. Activate the checkbox for the desired setting.

13.16 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.

13.16.1 Tool list for multitool

If you work with a multitool, the tool list is supplemented by the column for the multitool location number. As soon as the cursor is at a multitool in the tool list, certain column headings change.

Column header	Meaning
Location	Magazine/location number
MT loc.	Multitool location number
TYPE	Symbol for multitool
Multitool name	Name of the multitool

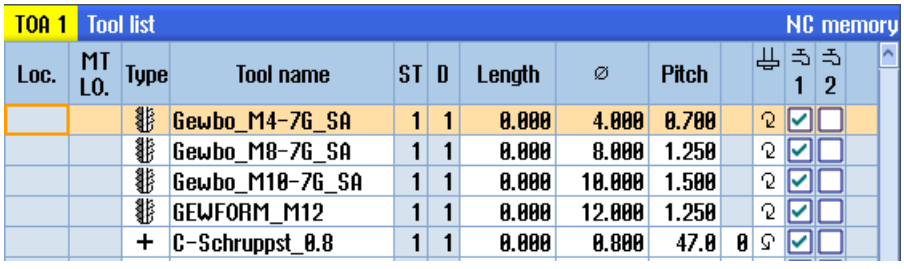


Figure 13-20 Tool list with multitool in the spindle

Procedure

1. Select the "Parameter" operating area.

2. Press the "Tool list" softkey.
   The "Tool list" window is opened.

13.16.2 Create multitool

The multitool can be selected in the list of favorites as well as in the list of special tool types.

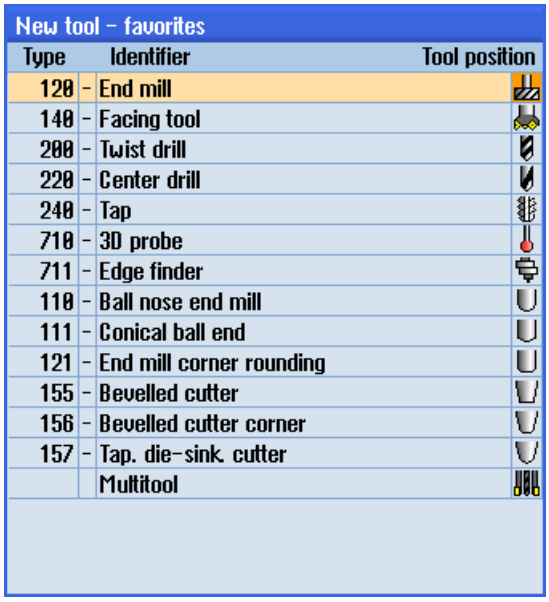


Figure 13-21 List of favorites with multitool

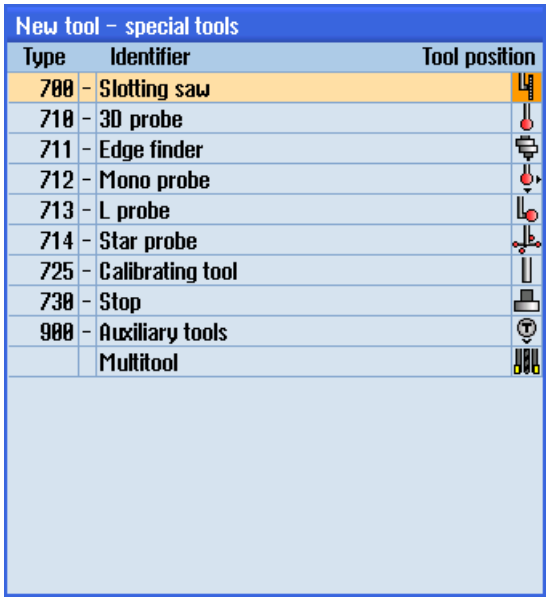


Figure 13-22 Selection list for special tools with multitool

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.

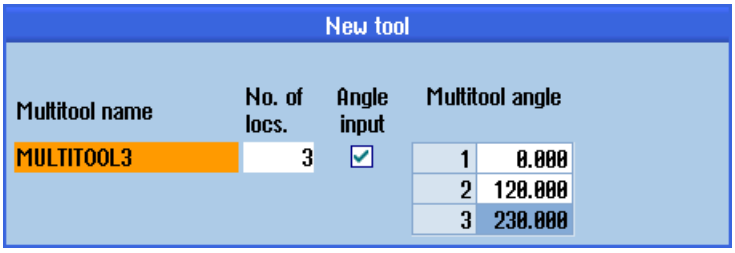
   The "New Tool - Favorites" window opens.

   - OR -

   Press the softkey "Special tool 700-900".

4. Select the multitool and press the "OK" softkey.
   The "New Tool" window appears.

5. Enter the multitool name and define the number of multitool locations.
   If you wish to define the clearance of the tools based on the angle, activate the "Angle input" checkbox, and for each multitool location, enter the clearance to the reference location as angular value.



The multitool is created in the tool list.

Note

The tool creation sequence can be defined differently.

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

13.16.3 Equipping multitool with tools

Precondition

A multitool has been created in the tool list.

Procedure

Tool list

1. The tool list is opened.

Equipping the multitool with a new tool

SELECT

2. Select the required multitool, position the cursor on an empty multitool location.

New tool

3. Press the "New tool" softkey.

4. Using the appropriate selection list, e.g. favorites, select the required tool.

Loading the multitool

SELECT

2. Select the required multitool, position the cursor on an empty multitool location.

Load

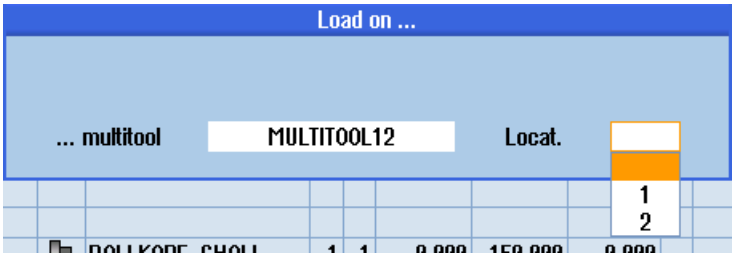
3. Press the "Load" softkey.
The "Load with..." window opens.

4. Select the required tool.

Loading the tool into the multitool

2. Position the cursor on the tool that you want to load into the multitool.

3. Press the "Load" and "Multitool" softkeys.
   The "Load on ..." window opens.



4. Select the required multitool and the multitool location to which you wish to load the tool.

13.16.4 Removing a tool from multitool

If the multitool was mechanically re-equipped, then the old tools in the tool list must be removed from the multitool.

To do this, the cursor is positioned at the line where the tool which is to be removed is located. When unloading, the tool is automatically saved in the tool list outside the magazine in the NC memory.

Procedure

1. The tool list is opened.

2. Position the cursor on the tool that you would like to unload from the multitool and press the "Unload" softkey.

   - OR -

   Position the cursor on the tool that you want to remove and delete from the multitool and press the "Delete tool" softkey.

13.16.5 Deleting multitool

Procedure

1. The tool list is opened.
2. Position the cursor on the multitool that you wish to delete.

3. Press the "Delete multitool" softkey.
   The multitool with all of the tools that are located in it is deleted.

13.16.6 Loading and unloading multitool

Procedure

1. The tool list is opened.

Loading a multitool into the magazine

2. Position the cursor at the multitool that you wish to load into the magazine.

3. Press the "Load" softkey.
   The "Load on ..." window opens.
   The field "... Locat." is initialized with the number of the first empty magazine location.

4. Press the "OK" softkey if you want to load the multitool to the recommended empty location.
   - OR -

   Enter the location number you require and press the "OK" softkey.
   The multitool together with the tools in it is loaded to the specified magazine location.

Loading a multitool into a magazine

2. Position the cursor on the required empty magazine location.

3. Press the "Load" softkey.
   The "Load with" window opens.

4. Select the required multitool.

5. Press the "OK" softkey.

Unloading a multitool

2. Position the cursor on the multitool that you wish to unload from the magazine.

Unload

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.

13.16.7 Reactivating the multitool

Multitool and tools located on the multitool can be disabled independently of one another.

If a multitool is disabled, then the tools of the multitool can no longer be changed in using a tool change.

If only one tool on a multitool has a set monitoring function and the lifetime or the unit quantity has expired, then the tool and the multitool on which the tool is located are disabled. The other tools on the multitool are not disabled.

Machine manufacturer

Please observe the information provided by the machine manufacturer.

If several tools with monitoring are mounted on the multitool and the lifetime or unit quantity has expired for one tool, then only this tool is disabled.

TOA 1 | Tool wear | WZ-Zwischenspeic...

Loc.	MT LO.	Type	Tool name	ST	D	Length	Δ ⌀	T C	Quan-tity	Set val	Prewar limit
			MULTITOOL								
	1		Schlichtfr_10_VHM	1	1	0.000	0.000				
	2		FRAESER_D10	1	1	0.000	0.000	C	0	15	11
C											
1/1			BOHRER_G19	1	1	0.000	0.000				

Reactivating

If a tool with expired lifetime or unit quantity that is mounted on a multitool is reactivated, then for this tool, the lifetime/unit quantity is set to the setpoint and the tool and the multitool are re-enabled (disable status is removed).

If a multitool is reactivated, on which tools with monitoring are mounted, then the lifetime/unit quantity for all tools on the multitool are set to the setpoint no matter whether the tools are disabled or not.

Requirements

In order to reactivate a tool, the monitoring function must be activated and a setpoint must be stored.

Procedure

1. Select the "Parameter" operating area.

2. Press the "Tool wear" softkey.

3. Position the cursor at the multitool that is disabled and which you would like to reactivate.

   - OR -

   Position the cursor on the tool that you want to reactivate again.

4. Press the "Reactivate" softkey.

   The value entered as the setpoint is entered as the new tool life or workpiece count.

   The tool and the multitool are then no longer disabled (the disable is withdrawn).

Reactivating and positioning

When the "Reactivate with positioning" function is configured, then also the magazine location at which the selected multitool is located is positioned at the loading point. You can exchange the multitool.

Reactivation of all monitoring types

When the "Reactivation of all monitoring types" function is configured, all the monitoring types set in the NC for a tool are reset during reactivation.

Machine manufacturer

Please observe the information provided by the machine manufacturer.

13.16.8 Relocating a multitool

Multitools can be directly relocated within magazines to another magazine location, which means that you do not have to unload multitools with the associated tools from the magazine in order to relocate them to a different location.

When you are relocating a multitool, the system automatically recommends an empty location. You may also directly specify an empty magazine location.

Procedure

1. Select the "Parameter" operating area.

2. Press the "Magazine" softkey.
3. Position the cursor at the multitool that you wish to relocate to a different magazine location.

4. Press the "Relocate" softkey.

   The "... relocate from location ... to location ..." window is displayed. The "Location" field is pre-assigned with the number of the first empty magagine location.

5. Press the "OK" softkey to locate the multitool at the recommended magazine location.

   - OR -

   Enter the required magazine number in the "...magazine" field and the required magazine location number in "Location" field.

   Note:

   Please refer to the machine manufacturer's specifications.

   Press the "OK" softkey.

   The multitool with the tools is relocated to the specified magazine location.

13.16.9 Positioning a multitool

You can position a magazine. In this case, a magazine location is positioned to the loading point.

Multitools that are located in a spindle can also be positioned. The multitool is rotated and therefore the multitool location involved is brought into the machining position.

Procedure

1. The magazine list is opened.

   The multitool is in the spindle.
2. Position the cursor on the multitool location that you want to bring into the machining position.

3. Press the "Position multitool" softkey.

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 storage space.

NOTICE
Possible interruption when executing from USB FlashDrive
Direct execution or simulation from a USB flash drive is not recommended.
There is no protection against contact problems, falling out, breakage through knocking or unintentional removal of the USB-FlashDrive during operation.
Disconnecting it during operation will result in the stopping of the machining and thus to the workpiece being damaged.

Multiple clamping with ShopMill

With ShopMill, you can implement multiple clamping of identical or different workpieces with optimization of the tool series.

Software options

Multiple clamping is only possible with ShopMill programs. You require the "ShopTurn/ ShopMill" option for this.

Storage for programs

Possible storage locations are:

- NC
- Local drive
- Network drives
- USB drives
- V24
- FTP drives

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).

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, select the storage location that contains the directories and programs that you want to display. In addition to the "NC" softkey, via which the file system data can be displayed, additional softkeys can be displayed.

The "USB" softkey can only be used when an external storage medium is connected (e.g. USB flash drive on the USB port of the operator panel).

Displaying documents

You can display documents on all drives of the Program Manager (e.g. in the local drive or USB) and via the data tree of the system data. Various data formats are supported:

- PDF
- HTML
  It is not possible to preview HTML documents.
- Various graphic formats (e.g. BMP or JPEG)
- DXF

Software options

You require the "DXF-Reader" option in order to display DXF files.

Note

FTP drive

It is not possible to preview documents on the FTP drive.

Structure of the directories

In the overview, the icons in the left-hand column have the following meaning:

Directory

Program

All directories have a plus sign when the program manager is called for the first time.

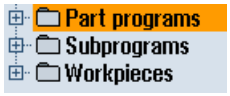


Figure 14-1 Program directory in the program manager

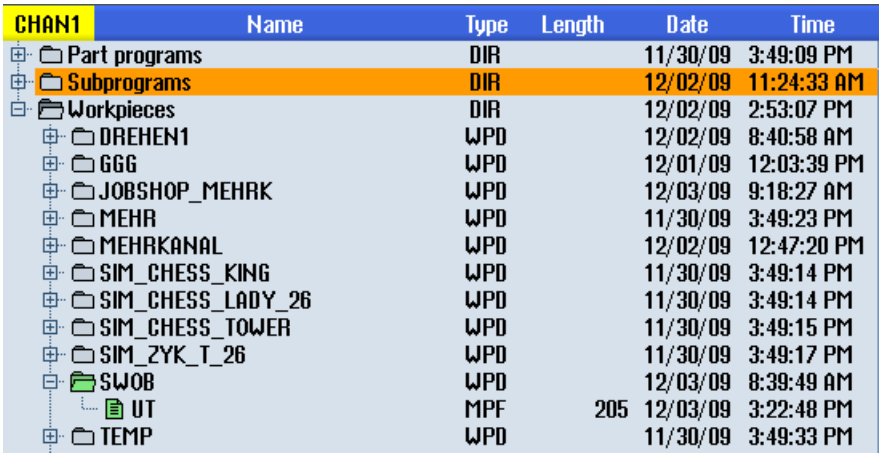
The plus sign in front of empty directories is removed after they have been read for the first time.

The directories and programs are always listed complete with the following information:

- Name
  The maximum length is 24 characters.
  Permissible characters include all upper-case letters (without accents), numbers, and underscores.
- Type
  Directory: WPD
  Program: MPF
  Subprogram: SPF
  Initialization programs: INI
  Job lists: JOB
  Tool data: TOA
  Magazine assignment: TMA
  Zero points: UFR
  R parameters: RPA
  Global user data/definitions: GUD
  Setting data: SEA
  Protection zones: PRO
  Sag: CEC
- Size (in bytes)
- Date/time (of creation or last change)

Active programs

Selected, i.e. active programs are identified using a green icon.



CHAN1	Name	Type	Length	Date	Time
	Part programs	DIR		11/30/09	3:49:09 PM
	Subprograms	DIR		12/02/09	11:24:33 AM
	Workpieces	DIR		12/02/09	2:53:07 PM
	DREHEN1	WPD		12/02/09	8:40:58 AM
	GGG	WPD		12/01/09	12:03:39 PM
	JOBSHOP_MEHRK	WPD		12/03/09	9:18:27 AM
	MEHR	WPD		11/30/09	3:49:23 PM
	MEHRKANAL	WPD		12/02/09	12:47:20 PM
	SIM_CHESS_KING	WPD		11/30/09	3:49:14 PM
	SIM_CHESS_LADY_26	WPD		11/30/09	3:49:14 PM
	SIM_CHESS_TOWER	WPD		11/30/09	3:49:15 PM
	SIM_ZYK_T_26	WPD		11/30/09	3:49:17 PM
	SWOB	WPD		12/03/09	8:39:49 AM
	UT	MPF	205	12/03/09	3:22:48 PM
	TEMP	WPD		11/30/09	3:49:33 PM

Figure 14-2 Active program shown in green

See also

Multiple clamping (Page 829)

14.1.1 NC memory

The complete NC working memory is displayed along with all tools and the main programs and subroutines.

You can create further subdirectories here.

Procedure

1. Select the "Program Manager" operating area.

2. Press the "NC" softkey.

14.1.2 Local drive

Workpieces, main and subprograms 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.

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.

14.1.3 USB drives

USB drives enable you to exchange data. For example, you can copy to the NC and execute programs that were created externally.

NOTICE
Interruption of operation Direct execution and simulation from the USB flash drive is not recommended, because machining can be undesirably interrupted, therefore resulting in workpiece damage.

Partitioned USB flash drive (TCU)

If the USB flash drive has several partitions, these are displayed in a tree structure as a subtree (01,02,...).

For EXTCALL calls, enter the partition (e.g. USB:/02/... or //ACTTCU/FRONT/02/... or //ACTTCU/FRONT,2/... or //TCU/TCU1/FRONT/02/...)

You can also configure any partition (e.g. //ACTTCU/FRONT,3).

Procedure

1. Select the "Program Manager" operating area.

2. Press the "USB" softkey.

Note

The "USB" softkey can only be operated when a USB flash drive is inserted in the front interface of the operator panel.

14.1.4 FTP drive

The FTP drive offers you the following options - to transfer data, e.g. part programs, between your control system and an external FTP server.

You have the option of archiving any files in the FTP server by creating new directories and subdirectories.

Note

Selecting a program / execution

It is not possible to select a program directly on the FTP drive, and change to execution in the "Machine" operating area.

Precondition

User name and password have been set up in the FTP server.

Procedure

1. Select the "Program manager" operating area.

2. Press the "FTP" softkey.

   When selecting the FTP drive for the first time, a login window is displayed.

3. Enter the user name and password and press the "OK" softkey to log into the FTP server.

   The content of the FTP server with its folders is displayed.

4. Press the "Log out" softkey after the required data processing has been completed.

   The connection to the FTP server is disconnected. In order to reselect the FTP drive, you must log on again.

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 / program 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 create 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. Please 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 required storage location.

3. Position the cursor on the folder in which you would like to create a workpiece.

4. Press the "New" softkey.
   The "New Workpiece" window appears.
5. If necessary, select a template if any are available.

6. Enter the desired workpiece name and press the "OK" softkey.

   A new folder with the workpiece name will be created.
   The directory type (WPD) is set by default.
   The "New G Code Program" window opens.

7. Press the "OK" softkey again if you want to create the program.

   The program will open in the editor.

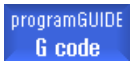
14.4.4 Creating a new G code program

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 opens.

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 either the "Subprograms" or "Part programs" folder, you can only create one subprogram (SPF) or one main program (MPF).

6. Enter the desired program name and press the "OK" softkey.

14.4.5 Creating a new ShopMill program

In the part program and workpiece directories, you can create ShopMill programs and then subsequently generate the machining steps for them.

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.

4. Press the "ShopMill" softkey.
   The "New Step Sequence Program" window opens.
   The "ShopMill" type is specified.

5. Enter the desired program name and press the "OK" softkey.

14.4.6 Storing any new file

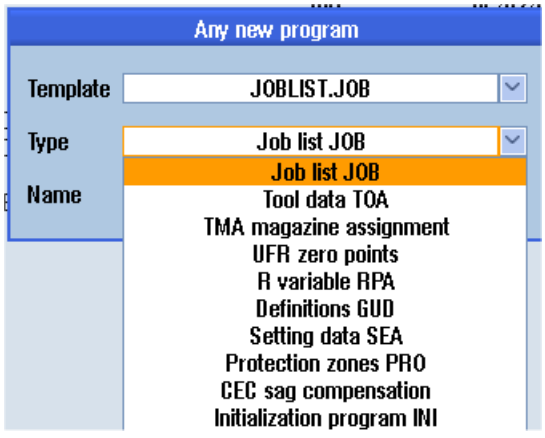
In each directory or subdirectory you can create a file in any format that you specify.

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.

5. Press the "Paste" softkey.

An appropriate note is displayed if a directory/program with the same name exists in this directory. You are requested to assign a new name, otherwise the directory/program is assigned a name by the system.

If the name contains illegal characters or is too long, a prompt will appear for you to enter a permissible name.

6. Press the "OK" or "Overwrite all" softkey if you want to overwrite existing directories/programs.

- OR -

Press the "No overwriting" softkey if you do not want to overwrite already existing directories/programs.

- OR -

Press the "Skip" softkey if the copy operation is to be continued with the next file.

- OR -

Enter another name if you want to paste the directory/program under another name and press the "OK" softkey.

Note

Copying files in the same directory

You cannot copy files to the same directory. You must copy the file under a new name.

14.10 Deleting a program/directory

14.10.1 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.

Temp directory for ShopMill

If you want to free up space in the NCK memory, delete the contents of the "TEMP" directory. ShopMill stores the programs that are created internally for calculating the stock removal processes in this directory.

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 Set up 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		
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.

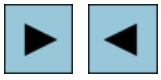
Specifications for the configured softkey

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 memory (EES)

The "Execution from external storage" function allows you to directly execute any size of part program from an appropriately configured drive. The behavior is the same as that for execution from the NC part program memory without the restrictions that apply to "EXTCALL".

Software option

You require the "CNC user memory extended" software option in order to use this function in the user memory (100 MB) of the 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.

See also

Searching directories and files (Page 790)

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

In addition to programs, you also have the option of saving tool data and zero point settings.

In this way you secure the required tools and zero point data for a specific machining step program. If you want to execute this program at a later point in time, then you can quickly access 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.

Backing-up job lists

If you wish to backup a job list, which contains ShopMill and G code programs, you obtain dedicated selection boxes to backup the tool data and zero points.

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	Setting options
Tool data	• No • All used in the program (only for ShopMill program and job list with ShopMill programs) • Complete tool list
Tool data for ShopMill programs -- only available for job list with ShopMill and G code programs	• No • All used in the program • Complete tool list
Tool data for G code programs -- only available for job list with ShopMill and G code programs	• No • Complete tool list
Magazine assignment	• Yes • No
Zero points	• No The selection box "Basis reference" is hidden • All used in the program (only for ShopMill program and job list with ShopMill programs) • All

Data	Setting options
Zero points for ShopMill programs -- only available for job list with ShopMill and G code programs	• No The selection box "Basis zero point" is hidden • All used in the program • Complete tool list
Zero points for G code programs -- only available for job list with ShopMill and G code programs	• No The selection box "Basis reference" is hidden • All
Basis reference	• No • Yes
Directory	The directory is displayed, in which the selected program is located.
File name	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 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

Identical names

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 observe the information provided by the machine manufacturer.

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 Recording tools and determining the demand

14.18.1 Overview

The "Record tools and determine tool demand" function allows you to record all the tools you need when executing and simulating part programs. In this way you can determine the tool demand.

Software options

In order to use the "Record tools and determine demand" function, you need the "Determine tool demand" option.

Machine manufacturer

Please observe the information provided by the machine manufacturer.

The "Record tools and determine demand" function can be helpful in the following work steps on single-channel and multi-channel machines:

- Preparation of a new workpiece while the old workpiece is still being machined:
  - All new tools must be loaded additionally
- Change of the workpiece at the machine:
  - All old tools can be unloaded
  - All new tools must be loaded

You activate the recording of tools in the settings for automatic operation (Page 281). Recording takes place during processing.

You can also activate recording for the simulation (Page 229).

The recorded tool data are stored in a TTD file (Tool Time Data). The TTD file is always located with the associated part program.

Note

The recorded tool data of a tool path is only valid if the program has been run through completely. Otherwise the data will not be saved and the previous TTD file will be retained.

14.18.2 Opening tool data

Introduction

The recorded tool data are stored in a TTD file (Tool Time Data). The TTD file is always located with the associated part program and contains the following information:

- Tool data
- Magazine assignment
- Operating time, idle time, tool number, D number, ID per block

You can open the TTD file with the "Check loading" softkey.

Note

You can only open TTD files that are assigned to the active channel.

Opening a TTD file

1. Select the "Program Manager" operating area.
2. Under part programs, open the corresponding file with the extension *.TTD.

   The "Tools for program" window opens.
   The file contents are displayed in the form of a tool list.

Note

If the TTD file is older than the associated part program, a message informs you of this.

14.18.3 Checking the loading

When you click the "Check loading" softkey, the "Check loading" window opens.

The view in the "Check loading" window is divided into the following sections:

- Missing tools: tool not available
- Tools still to be loaded: tool available but not loaded

- Loaded tools: tools available and loaded
- Tools not required: tools available and loaded, but not required

You can adjust all tool data displayed in the "Check loading" window as required, load and unload tools, enter the tool lengths for newly measured tools directly.

The list is updated after each action.

14.19 Backing up parameters

In addition to the programs, you can also save R-parameters and global user variables.

You can use this option, for example, to back up the required arithmetic parameters and user variables for a specific program. If you want to execute this program at a later point in time, you will then have quick access to the relevant data.

Note

Backing up parameters from part programs

Parameters from part programs can only be backed up if they have been saved in the "Workpieces" directory.

For part programs that are located in the "Part programs" or "Subprograms" directory, "Save parameters" is not listed.

Backing up data

Which data is offered for backup depends on the machine configuration:

Data	
R parameters	• No • Yes - all channel-specific arithmetic parameters
Global R parameters	• No • Yes - all global arithmetic parameters
UGUD parameters	• No • Yes - all channel-specific variables of the user
Global UGUD parameters	• No • Yes - all global variables of the user
MGUD parameters	• No • Yes - all channel-specific variables of the machine manufacturer
Global MGUD parameters	• No • Yes - all global variables of the machine manufacturer
Directory	The directory is displayed, in which the selected program is located.
File name	Here you have the option of changing the suggested file names.

For multi-channel machines, the parameters of the active channel are always backed up.

Job lists

If you select Back up parameters for a job list, the parameters of all programs that it contains are backed up.

The name of the job list does not agree with the names of the programs it contains. To nevertheless permit the unique assignment of the parameter files, they are always assigned the same name as the associated program. You cannot change these file names.

Procedure

1. Select the "Program Manager" operating area.

2. Select the drive on which the program is saved.

...

3. Position the cursor on the program whose parameters you want to back up.

4. Press the ">>" and "Archive" softkeys.

5. Press the "Save parameters" softkey.
   The "Save parameters" window appears.
6. Select the data you want to back up.

7. Press the <CHANNEL> key or click on the channel display if you want to change the active channel.

- OR -

8. Change the specified name of the originally selected program in the "File name" field as required.

9. Press the "OK" softkey.
   The parameters are saved in the same directory in which the selected program is stored.
   The R-parameters (*.RPA) and the user variables (*.GUD) are saved in separate files.

Note

Program selection

If a directory contains a main program as well as an RPA file or a GUD file with the same name, initially these files are automatically started when the main program is selected. Tool data or parameters may accidentally be changed as a result.

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

14.20 RS-232-C

14.20.1 Reading-in and reading-out archives via a serial interface

You have the option of reading out and reading in archives in the "Program manager" operating area as well as in the "Startup" operating area via the V24 serial interface.

Availability of the V24 serial interface

If you want to change the availability of the V24 interface, you can adjust the following parameters in file "slpmconfig.ini":

Parameters	Description	
`[V24]`	Describes the section in which the relevant setting parameters are located.	
`useV24`	Setting for the availability of the V24 serial interface	
	`= true`	Interface and softkeys are available (default)
	`= false`	Interface and softkeys are not available

Storage of file "slpmconfig.ini"

The template of the file "slpmconfig.ini" for SINUMERIK Operate is stored in the following directory:

<Installation path>/siemens/sinumerik/hmi/template/cfg

Copy the file to one of the following directories:

<Installation path>/**user**/sinumerik/hmi/cfg

<Installation path>/**oem**/sinumerik/hmi/cfg

Note

If you want to achieve a better overview of the changes you have made yourself, simply delete the unchanged parameters from the file copy "slpmconfig.ini".

Reading-out archives

The files to be sent (directories, individual files) are zipped in an archive (*.arc). If you send an archive (*.arc), this is sent directly without being additionally zipped. If you have selected an archive (*.arc) together with an additional file (e.g. directory), then these are zipped into a new archive and are then sent.

Reading-in archives

Use interface V24 if you want to read in archives. They are transferred and then subsequently unzipped.

Note

Reading in commissioning archives

When you read in a commissioning archive via the V24 interface, then this is immediately activated.

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.

14.20.2 Setting V24 in the program manager

V24 setting	Meaning
Protocol	The following protocols are supported for transfer via the V24 interface: • RTS/CTS (default setting) • Xon/Xoff
Transfer	Data transfer using a secured protocol (ZMODEM protocol): • Normal (default setting) • secure For the selected interface, secure data transfer is set in conjunction with handshake RTS/CTS.
Baud rate	Transfer rate: up to 115 kbaud data transfer rate. The baud rate that can be used depends on the connected device, the cable length and the general electrical conditions. • 110 • .... • 19200 (default) • ... • 115200
Archive format	• 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.

14.21 Multiple clamping

14.21.1 Multiple clamping

The "Multiple clamping" function optimizes tool changes over several workpiece clampings. This shortens idle times because a tool performs all machining operations in all clampings before the next tool change is initiated.

Software options

Multiple clamping is only possible with ShopMill programs. You require the "ShopTurn/ ShopMill" option for this.

In addition to surface clampings, you also have the option of using the "multiple clamping" function for rotating fixture plates. For this, the machine must have an additional rotary axis (e.g. A-axis) or a dividing unit.

Machine manufacturer

Please observe the information provided by the machine manufacturer.

You can machine not only identical but also different workpieces.

ShopMill automatically generates a single program out of several programs. The tool sequence within a program remains unchanged. Cycles and subprograms are not opened, position patterns are processed as closed units.

Requirements

The individual programs must meet the following requirements:

- Only sequence programs (not G code programs)
- Programs must be executable
- Program for the 1st clamping must have been installed.
- No markers/repetitions, i.e. no jumps in the program
- No inch/metric switchover
- No work offsets (except ShopMill program header)
- No coordinate transformation (translation, scaling, etc.)
- Contours must have unique names, i.e. the same contour name must not be called in several different programs.
- The "Starting point" parameter must not be set to "manual" in the stock removal cycle (contour milling).
- No modal settings, i.e. settings that are effective for all subsequent program blocks (with multiple clamping for different programs)

- 3500 operating steps max. per clamping
- Max. 49 clampings

Note

You can bypass the markers or repetitions, which must not be included in programs for multiple clampings, by using subprograms.

14.21.2 Program header setting, "Clamping"

During the generation of a multiple clamping program, data from the program header of a source program are transferred to a settings step of the multiple clamping program after every clamping change. The setting of the clamping in the program header is also part of this data. The clamping in the program header designates the name of the rotary axis in which the blank is clamped.

A distinction is made between the following multiple clamping programs:

- Program type 1, in which the same program is assigned to several clampings.
- Program type 2, in which a different program is assigned to each clamping.

A distinction is made between the following systems:

- System type A
  All clampings of the multiple clamping are mounted on a table or on a rotary axis.
- System type B
  The clampings are distributed over different rotary axes (rotary tables).

Multiple clamping programs of program type 1 cannot be executed on installations of type B and are therefore not permitted.

Reason:
The program header contains the axis name of one clamping. Since this axis name is entered in each setting step at a clamping change, the assignment to the actual rotary axis of the respective clamping is missing in the generated multiple clamping program.

Example:
The installation has two rotary tables C1 and C2. The same blank is clamped on both tables. Both blanks are to be machined with the same program. The rotary table must be addressed for the machining.

The original program designates clamping C1 in the program header. The resulting multiple clamping program uses the axis name C1 for both rotary tables, because the other axis name is not known to the program.

The other combinations of multiple clamping programs and installation types are permitted and supported.

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

See also

Program header (Page 341)

14.21.3 Creating a multiple clamping program

When assigning ShopMill programs to a multiple clamping program, you can use programs from NC directories and from external storage media (e.g. USB-FlashDrive).

Procedure

1. Select the "Program manager" operating area.

2. Press the ">>" and "Multiple clamping" softkeys.
   The "Multiple Clamping" window opens.

3. Enter the number of clampings and the number of the first work offset to be used.
   The clampings are processed in ascending order from the start work offset.
4. Enter a name for the new, global program.

4. Press the "OK" softkey.
   A list is displayed in which the different programs must be assigned to the work offsets. Not all work offsets, i.e. clampings, must be assigned to ShopMill programs, but at least two.
5. Enter the name of the desired ShopMill program directly in the list with the complete path specification. The file format (*.mpf) is automatically added.
   - OR -

   Press the "Program selection" softkey.
   The program view is displayed.

6. Position the desired ShopMill program in the program overview and press the "OK" softkey.
   - OR -

   If you wish to execute the same program on all clampings, press the "On all clampings" softkey.
   You can assign different programs to individual work offsets first, and then assign one program to the remaining work offsets by selecting the "On all clampings" softkey.

The program is included in the assignment list.

Assigned ShopMill programs that are not in the folder in which you create the multiple clamping program, are displayed with the complete path

7. Place the cursor on the desired program and press the "Delete selection" softkey if you want to remove individual programs from the assignment list.

   - OR -

   Press the "Delete all" softkey if you want to remove all programs from the global program.

8. Press the "OK" softkey when the assignment list is complete.

This optimizes the tool changes.

The global program is then renumbered. The number of the current clamping is specified every time the program switches from one clamping to another.

Apart from the global program (XYZ.MPF), the file XYZ_MCD.INI is also set up in which the assignment between work offsets and programs is stored. Both programs are stored in the directory that was previously selected in the Program Manager.

Note

If you switch from the assignment list (without canceling) to another function and then call the "Multiple clamping" function later, the same assignment list is displayed again.

Alarm, error, and system messages 15

15.1 Displaying alarms

If the machine develops a fault in operation, an alarm is generated and machining is possibly interrupted.

The error text that is displayed together with the alarm number gives you more detailed information on the error cause.

You have the possibility of saving all of the relevant diagnostic data to a ZIP file, which you can subsequently send to the hotline for analysis.

⚠ CAUTION
Dangers for persons and machines Carefully check the system, based on the description of the active alarm(s). Resolve the cause of the alarms. Then acknowledge the alarms in the specified way. Failure to observe this warning will place your machine, workpiece, saved settings and possibly even your own safety at risk.

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.

Save diag- nostics data

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 -

Delete Cancel alarm

- OR -

Delete HMI alarm

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 -

Acknowl. alarm

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

Symbol	Meaning
	RESET alarm
	Cancel alarm
	HMI alarm
	Dialog alarms of the HMI
	PLC alarm
	PLC alarm of the SQ type (alarm number from 800000)
	Safety alarms

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

15.2 Displaying an alarm log

A list of all the alarms and messages that have occurred so far are listed in the "Alarm Log" window.

Up to 500 administered, incoming and outgoing events are displayed in chronological order.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure

1. Select the "Diagnostics" operating area.

2. Press the "Alarm log" softkey.

   The "Alarm Log" window opens.

   All of the coming and going events - that have occurred since the HMI was started - are listed.

3. Press the "Display new" softkey to update the list of displayed alarms/ messages.

4. Press the "Save Log" softkey.

   The log that is currently displayed is stored as text file alarmlog.txt in the system data in directory `/user/sinumerik/hmi/log/alarm_log`.

15.3 Displaying messages

PLC and part program messages may be issued during machining.

These message will not interrupt the program execution. Messages provide information with regard to a certain behavior of the cycles and with regard to the progress of machining and are usually kept beyond a machining step or until the end of the cycle.

Overview of messages

You can display all issued messages.

The message overview contains the following information:

- Date
- Message number
  is only displayed for PLC messages
- Message text

Procedure

1. Select the "Diagnostics" operating area.

2. Press the "Messages" softkey.
   The "Messages" window appears.

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

Operand +

DB97.DBX2.5
Result: DB97.DBX2.6
$AA_IM[1]
Result: $AA_IM[2]

Operand –

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

The logbook provides you with the machine history in an electronic form.

If service is carried out on the machine, this can be electronically saved. This means that it is possible to obtain a picture about the "History" of the control and to optimize service.

Editing the logbook

You can edit the following information:

- Editing information on the machine identity
  - Machine name/No.
  - Machine type
  - Address data
- Make logbook entries (e.g. "filter replaced")
- Deleting logbook entries

Note

Deleting logbook entries

Up to the 2nd commissioning, you have the option to delete all of the entered data up to the time of the first commissioning.

Output of the logbook

You have the possibility of exporting the logbook by generating a file using the "Save version" function in which the logbook is contained as section.

See also

Save information (Page 846)

15.8.1 Displaying and editing the logbook

Procedure

1. Select the "Diagnostics" operating area.

2. Press the "Version" softkey.

3. Press the "Logbook" softkey.
   The "Machine logbook" window opens.

Editing end customer data

4. You have the option of changing the address data of the end customer using the "Change" softkey.

   - OR -

   Using the "Clear" softkey, you can delete all logbook entries.

   All entries, except the date of the first commissioning, are deleted. Softkey "Clear" is deactivated.

Note

Deleting logbook entries

As soon as the 2nd commissioning has been completed, the "Clear" softkey to delete the logbook data is no longer available.

15.8.2 Making a logbook entry

Using the "New logbook entry" window to make a new entry into the logbook.

Enter your name, company and department and a brief description of the measure taken or a description of the fault.

Note

Setting line breaks

If you wish to make line breaks in the "Fault diagnostics/measure" field, use the key combination <ALT> + <INPUT>.

The date and entry number are automatically added.

Sorting the entries

The logbook entries are displayed numbered in the "machine logbook" window.

More recent entries are always added at the top in the display.

Procedure

1. The logbook is opened.

2. Press the "New entry" softkey.

   The "New logbook entry" window opens.

3. Enter the required data and press the "OK" softkey.

   You return to the "Machine logbook" window and the entry is displayed below the machine identity data.

Note

Deleting logbook entries

Up to the completion of the 2nd commissioning, you have the option to delete the logbook entries up to the time of the first commissioning using the "Clear" softkey.

Searching for a logbook entry

You have the option for searching for specific entries using the search function.

1. The "Machine logbook" window is opened.
2.

Press the "Find" softkey.
3. Enter the desired term in the search form. You can make a search according to date/time, company name/department or according to fault diagnostics/measure.
   The cursor is positioned on the first entry that corresponds to the search term.
4.

Press the "Continue search" softkey if the entry found is not the one that you are looking for.

Additional search option

Press the "Go to Beginning" softkey to start the search at the latest entry.

Press the "Go to End" softkey to start the search at the oldest entry.

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".

Working with Manual Machine 16

16.1 Manual Machine

"Manual Machine" offers a modified comprehensive spectrum of functions for manual mode. You can carry out all the important machining processes without writing a program.

Software options

You require the "ShopTurn/ShopMill" option for working with "Manual Machine".

Machine manufacturer

Please observe the information provided by the machine manufacturer.

Main screen

After the controller has powered up, the following main screen is displayed:

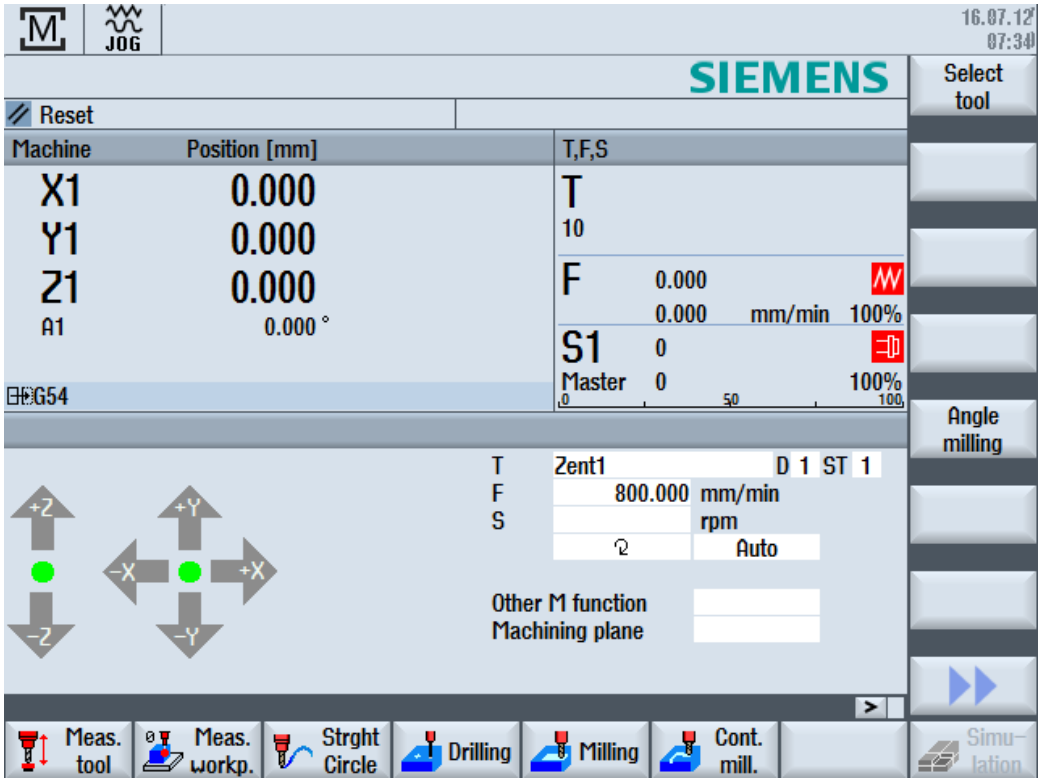


Figure 16-1 Main screen for milling machine

Main screen for milling/turning machine

The same scope of turning cycles is available for a milling/turning machine as in automatic mode.

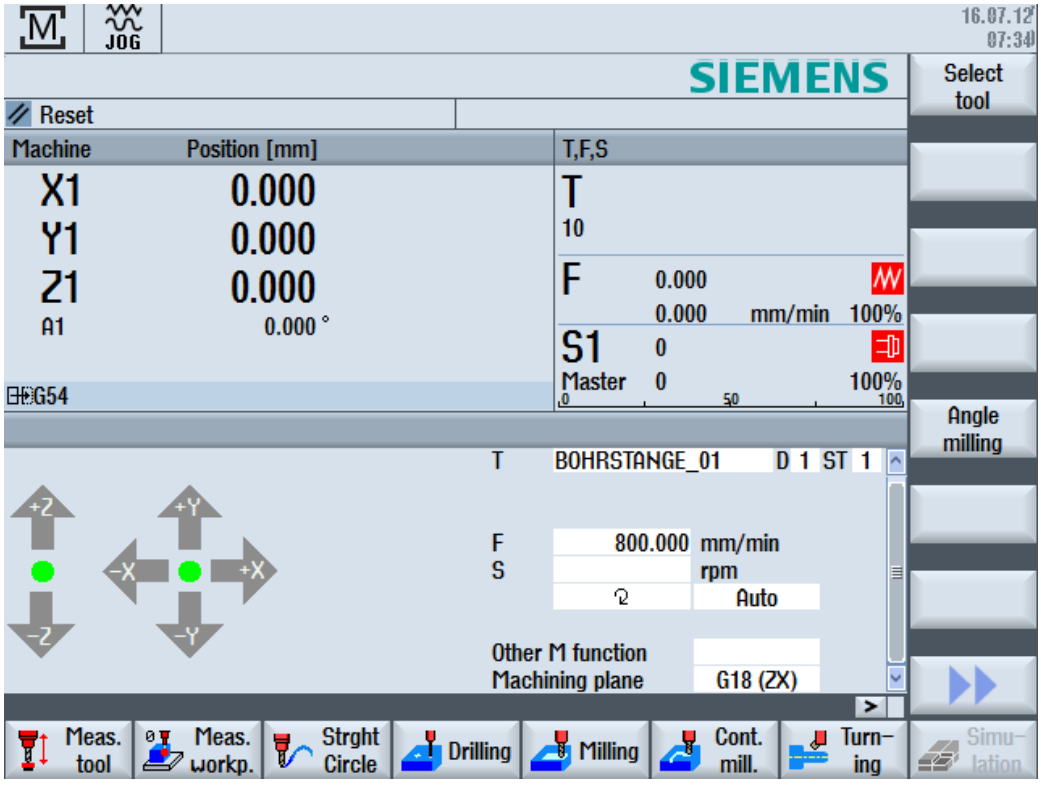


Figure 16-2 Main screen for milling/turning machine

Machining options

You have the following options for machining the workpieces:

- Manual mode
- Single-cycle machining

16.2 Measuring the tool

All manual and automatic measurement options are available to determine the tool offset data. Further information can be found in the "Measure tool (Page 107)" section.

Procedure

1. "Manual Machine" is active.
2.

Press the "Meas. tool" softkey.
3. Select the the required measuring function in the vertical softkey bar and press the appropriate softkey.

16.3 Measuring the workpiece zero

You can use the following workpiece elements to determine the workpiece zero:

- Edge
- Corner
- Pocket/hole
- Spigot
- Plane

You can measure the workpiece zero either manually or automatically. Further information can be found in the "Measuring the workpiece zero (Page 120)" section.

Procedure

1. "Manual Machine" is active.
2.

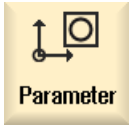
Press the "Workpiece zero" softkey.
3. Select the the required measuring variant in the vertical softkey bar and press the appropriate softkey.

16.4 Setting the zero offset

Directly select the work offset in the "Parameter" operating area in the work offset list.

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

Procedure

1. "Manual Machine" is active.
2. 

Select the "Parameter" operating area.
3.

Press the "Work offset" and "G54...599" softkeys.
The "Work offset G54...599" window opens.
4.

Position the cursor on the desired work offset and press the "Workpiece zero" softkey.
The "Measure: Edge" is opened with the selected work offset.
5.

Press the "Set WO" softkey and confirm with "OK".

16.5 Set limit stop

You can limit the traversing range of the axes.

To do this, enter the values for the respective axes. The values refer to the workpiece coordinate system. The limits can be switched on and off individually.

Activated, i.e. active set limits, are indicated by a bar next to the wind rose in the direction graphic.

When a limit is reached, an alarm appears which disappears again when the axis is moved away from the limit.

Note

Mode change

Entered and activated limit stops remain active after switching from JOG mode to the MDA or AUTO mode.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure

1. "Manual Machine" is active
2. Press the "Limit stops" softkey.

   The "Limit Stops" window opens.
3. Enter the desired position of the limit stop for each axis.

   - OR -

   Press the "Set limit stop" softkey to enter the current position of an axis.

4. In the field next to the position specification select the entry "On" to activate the desired limit stop.

   The bar is displayed next to the wind rose.
5. Press the "Back" softkey to return to the basic screen.

   The active limit stops are also displayed here with bars.

16.6 Simple workpiece machining

In "Manual Machine", you machine workpieces directly in the "JOG" mode without creating a program.

Functions

The following functions are available to you for machining in manual mode:

- Axis movements
- Angular milling
- Straight (face and longitudinal milling) and circle

Note

Tool, spindle speed and direction of spindle rotation are activated with <CYCLE START>.

A change in feedrate immediately becomes active.

16.6.1 Traversing axes

For preparatory actions and simple traversing movements, input the parameters directly into the "Manual Machine" input fields of the basic screen.

Tool selection

1. "Manual Machine" is active.

Tool selection

2. Select the desired tool in "T".
3. Enter the feedrate (F) and the spindle speed (S).

4. Select the direction of spindle rotation (e.g. clockwise direction of rotation):

   - OR -

   Set the direction of rotation via the machine control panel.

5. Press the <CYCLE START> key.

   The spindle starts immediately after the tool is selected.

   Note:

   Please refer to the machine manufacturer's specifications.

Machining

6. Select the axis to be traversed on the machine control panel.

...

7. Press the <+> or <-> key on the machine control panel.

... - OR -

Select the direction with the aid of the cross-switching lever.

The axes are moved at the set machining feedrate.

Note:

Please refer to the machine manufacturer's specifications.

The active direction is graphically displayed in the basic screen by means of the wind rose.

16.6.2 Angular milling

The basic effective direction can be selected via the axis direction keys or via the cross-switching lever. In addition, an angle (α1) can also be entered.

Procedure

1. "Manual Machine" is active.

2. Press the "Angular milling" softkey.

3. Select the tool, spindle, and spindle direction and specify the machining feedrate.
4. Enter the desired angle α1.

Note

Selecting/deselecting the angular milling and changing the angle α1 is only possible in the reset state.

Parameter	Description	Unit
T	Tool name	
D	Cutting edge number	
F	Feedrate	mm/min mm/rev
S / V	Spindle speed or constant cutting rate	rev/min m/min

Parameter	Description	Unit
α1	Rotation of the coordinate system	Degrees
Other M function	Input of machine functions Refer to the machine manufacturer's table for the correlation between the meaning and number of the function.	
Zero offset	Selecting the zero offset.	

16.6.3 Straight and circular machining

16.6.3.1 Straight milling

Use this function for simple, straight machining (e.g. face or longitudinal turning).

Procedure

1. "Manual Machine" is active.
2. Press the "Straight Circle" softkey.

3. Select the desired straight machining and press the "Straight all axes" softkey

- OR -

Press the "Straight X α" softkey.

- OR -

Press the "Straight X α" softkey.

4. Specify the desired value for the feedrate F.

- OR -

Press the "Rapid traverse" softkey.

The rapid traverse is displayed in field "F".

5. Enter the target position and, if required, the angle (α) for the axis or axes to be traversed.

Using the "Graphic view" softkey, you can toggle between the help screen and the graphic view in the screen.

Parameter	Description	Unit
F ○	Feedrate	mm/min mm/rev
	Straight all axes	
X○	Target position in the X direction (abs or inc)	mm
Z○	Target position in the Z direction (abs or inc)	mm
Y○	Target position in the Y direction (abs or inc)	mm
B○	Target position of the B axis (abs or inc)	mm
	Straight X α	
X	Target position in the X direction (abs or inc)	mm
α	Angle of the straight line to the X axis	Degrees
	Straight Z α	
Z	Target position in the Z direction (abs or inc)	mm
α	Angle of the straight line to the Z axis	Degrees

16.6.3.2 Circular milling

You can use this function for a simple circular machining.

Procedure

1. "Manual Machine" is active.
2.

Press the "Straight circle" softkey.
3.

Press the "Circle" softkey.
4. Specify the desired value for the feedrate F.
5.

Select the desired circle input (e.g. "End point + radius") and the direction of rotation.
6. Enter the target position as well as the circle center point or radius.

Using the "Graphic view" softkey, you can toggle between the help screen and the graphic view in the screen.

Parameter

Parameter	Description	Unit
F	Feedrate	mm/min mm/rev
Circle input	• End point + center point • End point + radius	mm
Direction of rotation		
Z	Target position in the Z direction (abs and inc)	mm
X	Target position in the X direction (abs and inc)	mm
K	Center of the circle K (inc) - only if circle input via end point and center point **Note**: Incremental dimensions: The sign is also evaluated.	mm
I	Center of circle I (inc) - only if circle input via end point and center point	mm
PL	Circle plane • XY IJ • YZ JK • ZX KI	

16.7 More complex machining

The following functions are available to you for more extensive and complicated machining in manual mode:

- Drilling (centering, drilling, reaming, deep-hole drilling, threads, positions)
- Milling (face milling, pocket, spigot, multiple edge, groove, thread milling, engraving)
- Contour milling (contour, path milling, predrilling, pocket)
- Turning (stock removal, groove, undercut, threads, tapping) – only milling/turning machines

General sequence

For more complex machining processes, proceed in the following order:

- Select the desired function via the corresponding softkey.
- Input the desired values in the parameter screen.
- Press the "OK" softkey in order to accept the values.
  The input screen closes.
  A line with the specified parameters is displayed on the basic screen.
- Press the <CYCLE START> key.
  The selected cycle is started.

Note

You can return to the parameter screen at any time to check and correct the inputs.

Press the "Cursor right" key to jump back to the input screen.

Drilling a position pattern

You can drill a position pattern:

- First select the desired function (e.g. "Centering") via the softkey in "Drilling".
- Select the appropriate tool and enter the desired values in the parameter screen. Press the "Accept" softkey to confirm the technology block.
  The input screen closes. The main screen shows the line with the technology data
- Press the "Positions" softkey and select the desired position pattern (e.g. Any position) via softkey. Enter the desired values in the parameter screen and press the "Accept" softkey.
  The input screen is closed and the technology and positioning blocks are displayed in brackets.

Swiveling

Manual swivel is available for the setup, measuring and machining of workpieces with inclined, swiveled surfaces (see Section "Swiveling (Page 180)").

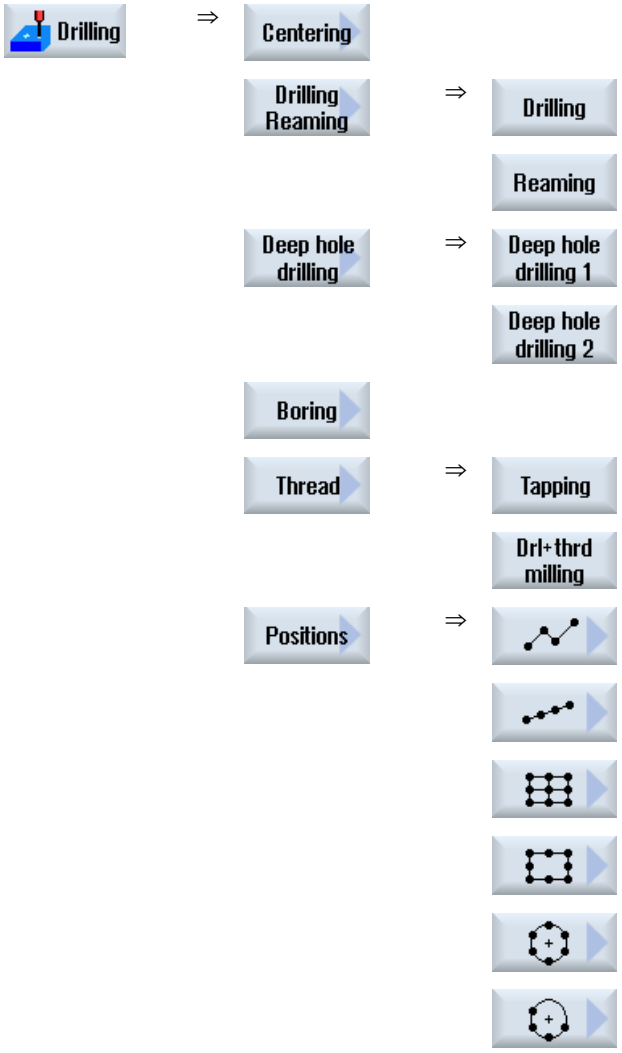
Approach and retraction

When machining the workpiece, you traverse from the current position to the machining start point. After the machining process, the tool is returned along a direct path to the starting point.

16.7.1 Drilling with Manual Machine

Functions (cycles)

The same range of technological functions (cycles) is available as in automatic mode for drilling on the face or peripheral surface of a workpiece:

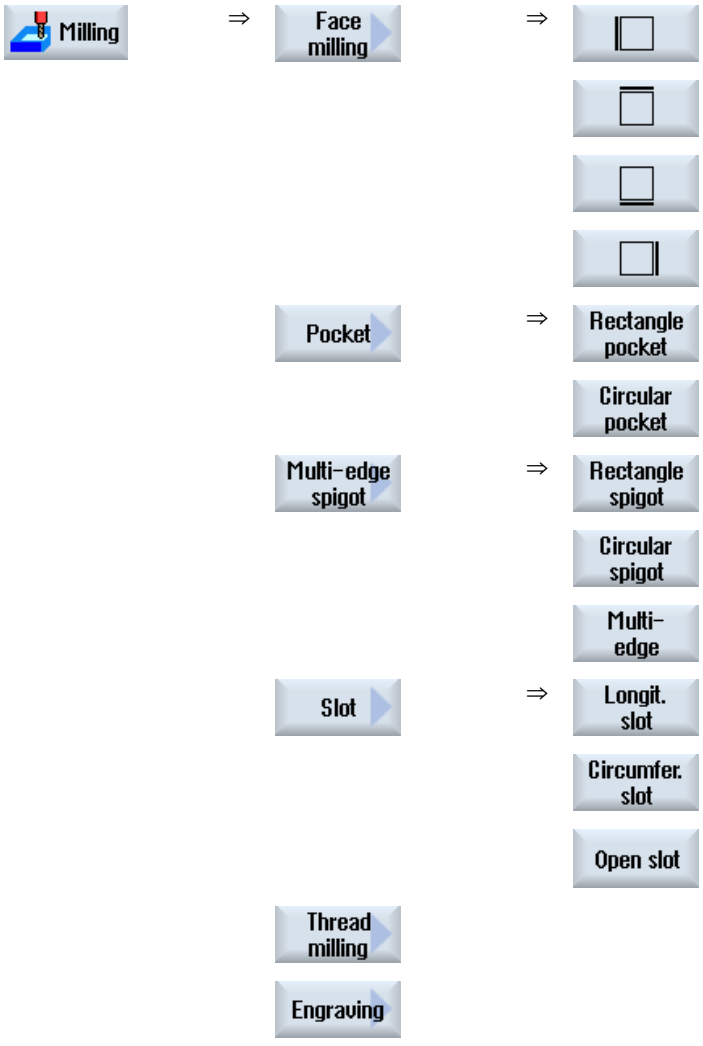


Parameter

The parameters of the input screen forms correspond to the parameters under Automatic (see Section "Drilling (Page 386)").

16.7.2 Milling with Manual Machine

The same range of technological functions (cycles) is available as in automatic mode for the milling of simple geometric shapes:

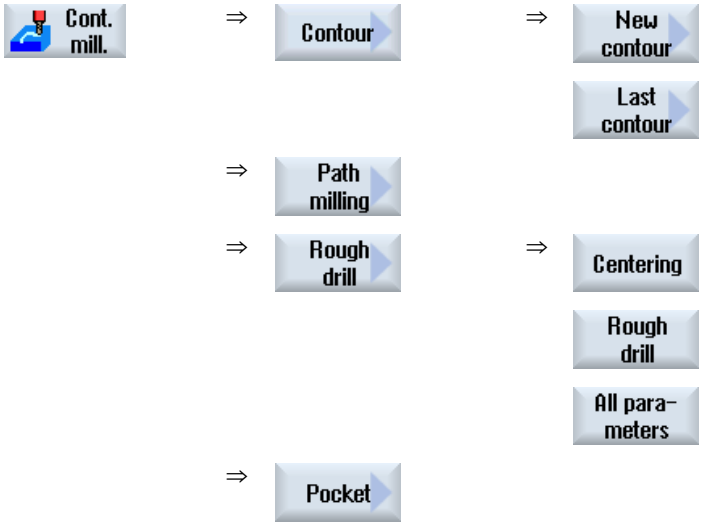


Parameter

The parameters of the input screen forms correspond to the parameters under Automatic (see Section "Milling (Page 434)").

16.7.3 Contour milling with manual machine

The same range of technological functions (cycles) is available as in automatic mode for contour milling of simple geometric shapes:



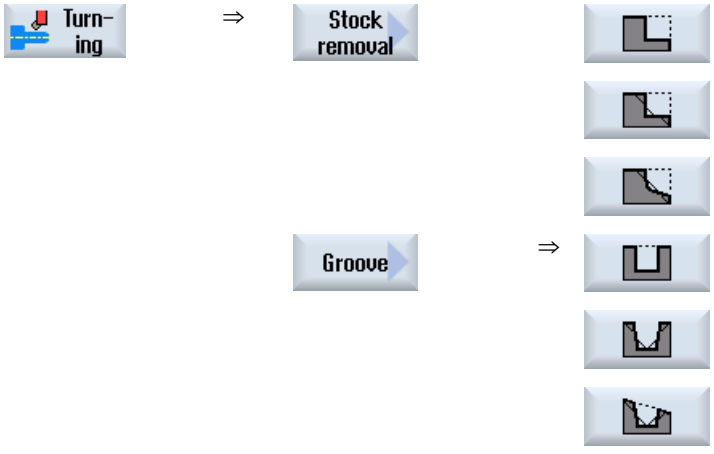
Parameter

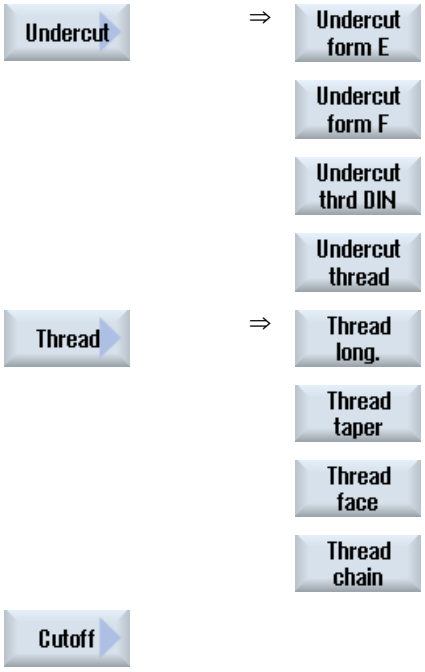
The parameters of the input screen forms correspond to the parameters under Automatic (see Section Contour milling (Page 497)).

16.7.4 Turning with manual machine - milling/turning machine

Functions (cycles)

The same range of technological functions (cycles) is available as in automatic mode for the turning of simple geometric shapes:




Undercut
⇒
Undercut form E
Undercut form F
Undercut thrd DIN
Undercut thread
Thread
⇒
Thread long.
Thread taper
Thread face
Thread chain
Cutoff

16.8 Simulation and simultaneous recording

For more complex processing, you can check the result of your inputs with the aid of the simulation, without having to traverse the axes (see Section "Simulating machining (Page 285)"). The execution of the work steps is graphically displayed on the screen during this.

Software option

You require the option "Simultaneous recording (real-time simulation)" for the simultaneous recording of the work steps.

Note

In "Manual Machine", you can simulate a work step with an already opened and filled out parameter screen form.

Teaching in a program 17

17.1 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.

17.2 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.

17.3 Processing a program

17.3.1 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.

17.3.2 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.

17.3.3 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.

17.3.4 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.

17.4 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.

17.4.1 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

17.5 Settings for teach-in

In the "Settings" window, you define which axes are to be included in the teach-in block and whether motion-type and continuous-path mode parameters are to be provided.

Procedure

1. Teach-in mode is active.

2. Press the ">>" and "Settings" softkeys.
   The "Settings" window opens.

3. Under "Axes to be taught" and "Parameters to be taught", select the check boxes for the desired settings.

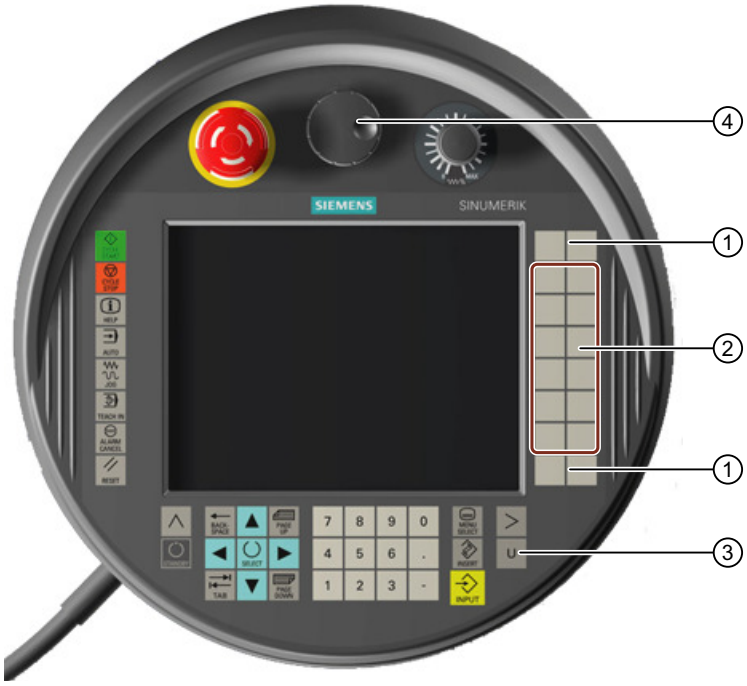
4. Press the "Accept" softkey to confirm the settings.

Handheld terminals for multi-touch operation 18

18.1 HT 8

18.1.1 HT 8 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.

18.1.2 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.

18.1.3 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.

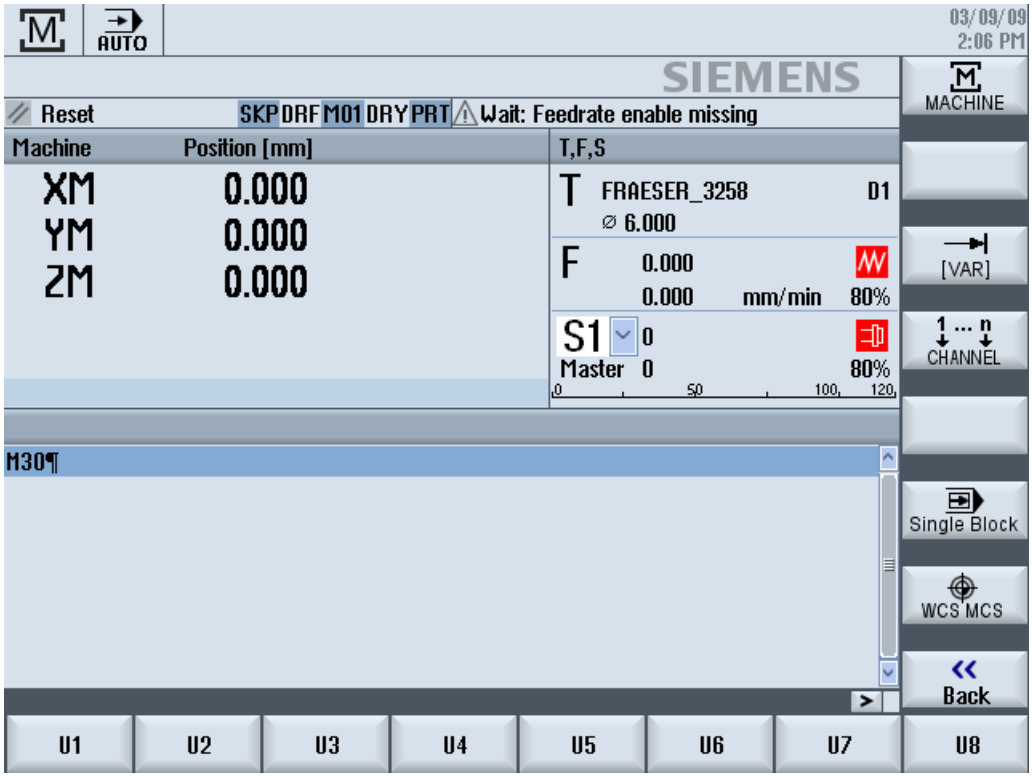
See Section "Controls on the machine control panel" for a description of the individual keys.

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.

18.1.4 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

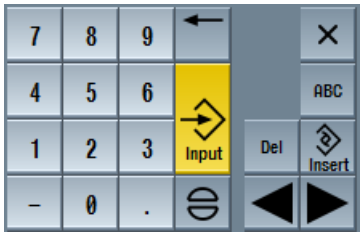
Toggles the keyboard assignment between the English keyboard assignment and the keyboard assignment for the current language setting.

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.

18.2 HT 10

18.2.1 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.

18.2.2 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.

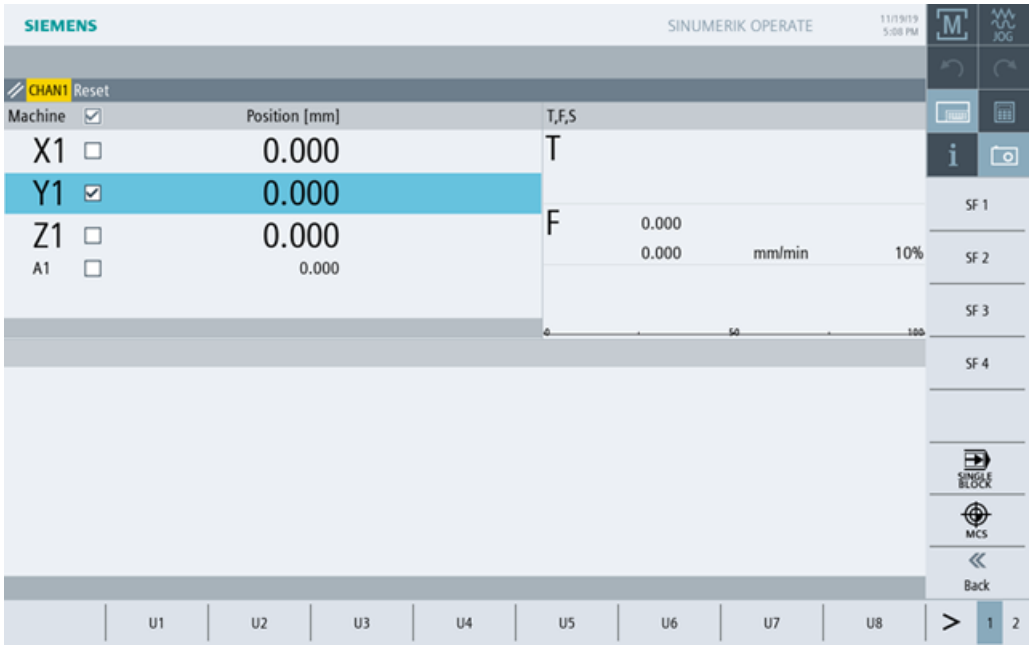
See Section "Controls on the machine control panel" for a description of the individual keys.

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.

18.2.3 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.

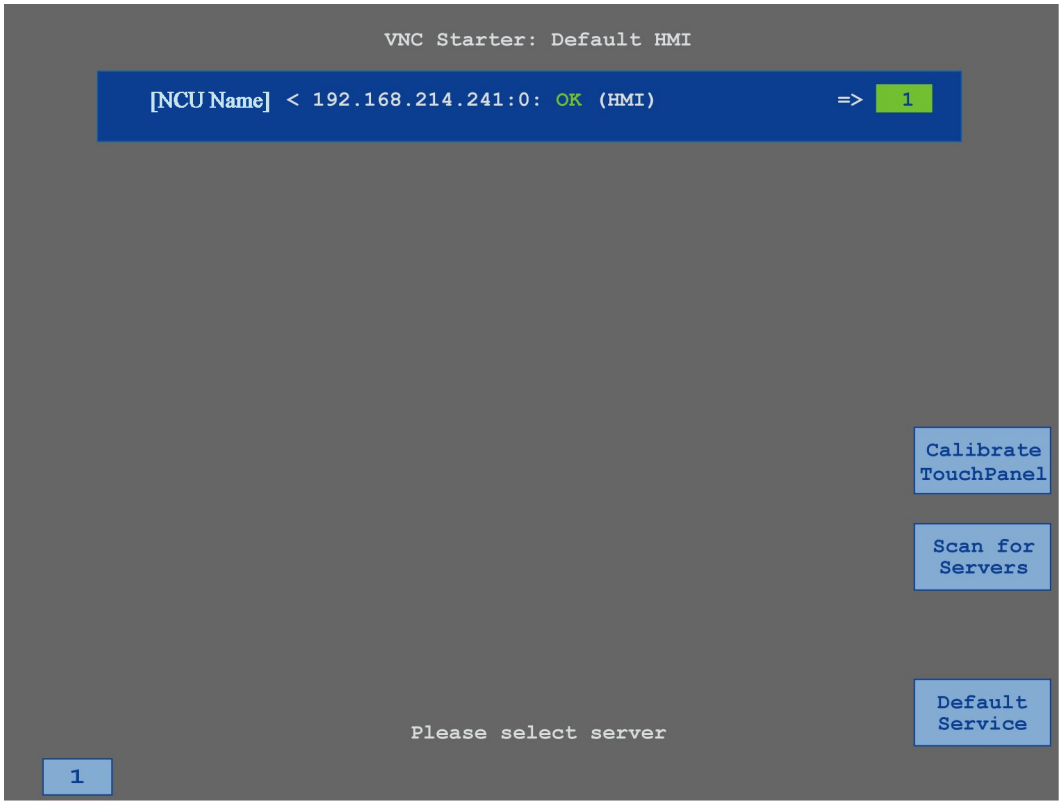
18.3 Calibrating the touch panel

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.

Ctrl-Energy 19

19.1 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.

19.2 Ctrl-E analysis

19.2.1 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

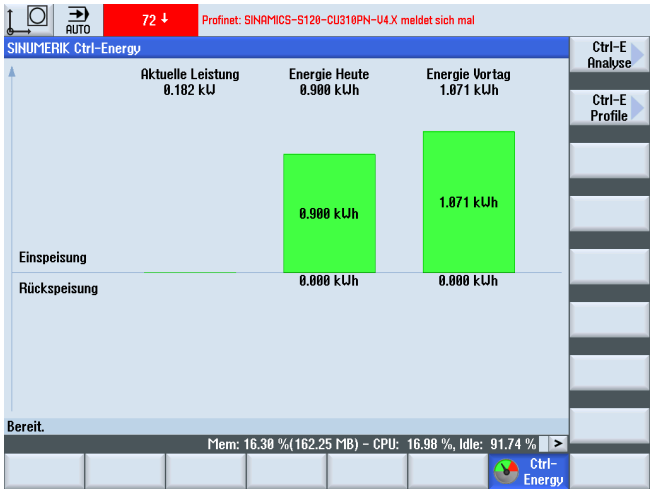


Figure 19-1 Ctrl-Energy entry screen with display of the current energy consumption

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.

19.2.2 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.

19.2.3 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.

19.2.4 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.

19.2.5 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.

19.2.6 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.

19.2.7 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.

19.3 Ctrl-E profiles

19.3.1 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

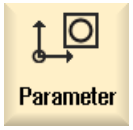
Energy-saving profile	Meaning
Simple energy-saving mode (machine standby)	Machine units that are not required are either throttled or shut down. When required, the machine is immediately ready to operate again.
Full energy-saving mode (NC standby)	Machine units that are not required are either throttled 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.

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: Functions
- Manufacturer/Service: Hardware
- Manufacturer/Service: Configuration/Setup
- Manufacturer/Service: Safety Integrated
- Manufacturer/Service: SINUMERIK Integrate/MindApp
- Information and training
- Manufacturer/Service: SINAMICS

Index

"

"SINUMERIK Operate Gen. 2"
Multitouch panel, 71
"SINUMERIK Operate Generation 2" user interface, 71
Function key block, 77
Touch operator controls, 77
Virtual keyboard, 78

A

ABC keyboard, 87
Acknowledgment button
HT 10, 891
HT 8, 884
Actual-value display, 43
Adapt to load - CYCLE782
Function, 674
Parameter, 675
Adapter-transformed view, 761
Advanced Surface, 265
Alarm log
display, 836
Sorting, 838
Alarms
Backing up log data, 833
cancel, 834
Displaying, 833
Sorting, 838
Align turning tool - CYCLE800
Function, 664
Parameters, 668
Angular milling
Manual Machine, 862
Any file, 785
Arbitrary positions - CYCLE802
Function, 424
Parameter, 425
Archive
Generate in the program manager, 810
generating in the system data, 811
Punched tape format, 810
read in archive from system data, 814
reading-in in the Program Manager, 813
Auxiliary functions
H functions, 266
M functions, 266
Axes
Defined increment, 177
Direct positioning, 179
Referencing, 96
Repositioning, 206
Traversing, 177
Variable increment, 178

B

Backing up
Data - in the Program Manager, 810
Data - via the system data, 811
Parameter, 822
Setup data, 815
Base offset, 153
Basic blocks, 202
Binary format, 810
Blank
Change, 356
Blank input
Function, 318
Parameter, 319
Block
Search, 207
Searching - interruption point, 210
Searching - search pointer, 211
Block search
Coolant, 726
Mode, 212
Position pattern, 214
Program interruption, 210
Search pointer, 211
Search target parameters, 212
Search target specification, 210
Using, 207
Boring - CYCLE86
Function, 408
Parameter, 410

C

Calibration of length
Probe, 147
Calibration of radius / diameter
Probe, 147
Call
Interpolation turning, 678

Camera
Streaming, 65
Centering - CYCLE81
Function, 387
Parameter, 388
Change
Blank, 356
Channel switchover, 101
Checking the loading, 820
Circle
Manual Machine, 864
Polar, 693
With known center point - function, 688
With known center point - parameter, 689
With known radius - function, 689
With known radius - parameter, 690
Circle position pattern - HOLES2
Parameter, 430
Circle position pattern – HOLES2
Function, 429
Circular pocket - POCKET4
Function, 443
Input simple, 444
Parameter - input complete, 448
Parameter - input simple, 448
Circular spigot - CYCLE77
Function, 455
Input simple, 455
Parameter - input complete, 456
Parameter - input simple, 457
Circumferential slot - SLOT2
Function, 469
Input simple, 470
Parameter - input complete, 473
Parameter - input simple, 473
Cleaning mode, 65
Coarse and fine offsets, 153
Code carrier connection, 734
Collision avoidance, 705
Displaying the machine model (kinematic chains), 707
Machine operating area, 708
Settings, 708
Compare to sphere
Probe, 149
Configured stop, 217
Context-sensitive online help, 67
Contour call - CYCLE62
Function, 506, 603
Parameter, 507, 604
Contour milling
Manual Machine, 869
Contour pocket residual material - CYCLE63
Function, 522
Parameters, 523
Contour spigot residual material - CYCLE63
Function, 527
Parameters, 529
Contour turning
Overview, 592
Retraction planes, 592
Coolant
Block search, 726
Tool change, 726
Coordinate system
Switching over, 103
transforming, 679
Coordinate transformation
Calling the functions, 679
Functions, 679
Mirroring - function, 682
Mirroring - parameter, 682
Offset - function, 680
Offset - parameter, 680
Rotation - function, 680
Rotation - parameter, 681
Scaling - function, 681
Creating
Multitool, 763
Program block, 227
Ctrl-Energy
Comparing usage values, 902
Displaying measured curves, 901
Displaying usage values, 901
Energy analysis, 898, 899
Energy-saving profiles, 904
Functions, 897
Measuring the energy consumption, 900
Saved measuring curves, 901, 902
Cut-off - CYCLE92
Function, 588
Parameter, 591
Cutting edge width, 725
Cutting tip length, 725
Cutting tip width, 725
CYCLE60 - engraving
Function, 490
CYCLE60 - Engraving
Parameters, 496
CYCLE61 - Face milling
Function, 434
Parameters, 436

CYCLE62- contour call
Function, 506, 603
Parameter, 507, 604
CYCLE63 - Contour pocket residual material
Function, 522
Parameters, 523
CYCLE63 - Contour spigot residual material
Function, 527
Parameters, 529
CYCLE63 - Milling contour pocket
Function, 517
Input simple, 517
Parameter - input complete, 520
Parameter - input simple, 520
CYCLE63 - Milling contour spigot
Function, 524
Input simple, 524
Parameter - input complete, 526
Parameter - input simple, 526
CYCLE64 - Predrilling contour pocket
Function - Centering, 514
Function - Predrilling, 514
Parameter - Centering, 515
Parameter - Predrilling, 516
CYCLE70 - thread milling
Function, 486
Parameter, 490
CYCLE72 - path milling
Function, 507
CYCLE72 - Path milling
Parameter, 512
CYCLE76 - rectangular spigot
Function, 450
Input simple, 451
Parameter - input complete, 453
Parameter - input simple, 453
CYCLE77 - circular spigot
Function, 455
Input simple, 455
Parameter - input complete, 456
Parameter - input simple, 457
CYCLE78 - Drill thread milling
Function, 417
Parameter, 420
CYCLE782 - Adapt to load
Function, 674
Parameter, 675
CYCLE79 - multi-edge
Function, 459
Input simple, 459
Parameter - input complete, 461
Parameter - input simple, 461
CYCLE800 - align turning tool
Function, 664
Parameters, 668
CYCLE800 - Setting milling tool
Function, 662
Parameter, 664
CYCLE800 - Swiveling
Function, 652
Parameter, 660
CYCLE801 - frame position pattern
Function, 427
Parameter, 428
CYCLE801 - grid position pattern
Function, 427
Parameter, 428
CYCLE802 - arbitrary positions
Function, 424
Parameter, 425
CYCLE81 - centering
Function, 387
Parameter, 388
CYCLE82 - drilling
Function, 388
Input simple, 389
Parameter - input simple, 391
Parameter input complete, 390
CYCLE83 - deep-hole drilling 1
Function, 393
Input simple, 393
Parameter - input simple, 396
CYCLE83 - Deep-hole drilling 1
Parameter - input complete, 396
CYCLE830 - deep-hole drilling 2
Parameter - input complete, 405
Parameter - input simple, 405
Through drilling, 399
CYCLE830 - Deep-hole drilling 2
Function, 398
Hole entry, 399
Input simple, 398
With pilot hole, 399
With/without predrilling, 399
CYCLE832 - High Speed Settings
Function, 668
Parameters, 672
CYCLE84 - tapping without compensating chuck
Function, 410
Input simple, 410
Parameter - input complete, 415
Parameter - input simple, 415
CYCLE840 - tapping with compensating chuck
Function, 410

Input simple, 410
Parameter - input complete, 415
Parameter - input simple, 415
CYCLE85 - reaming
Function, 392
Parameter, 393
CYCLE86 - boring
Function, 408
Parameter, 410
CYCLE899 - open slot
Parameter - input complete, 482
Parameter - input simple, 482
CYCLE899 – open slot
Function, 475
Input simple, 476
CYCLE92 - cut-off
Function, 588
Parameter, 591
CYCLE930 - groove
Function, 534
Parameter, 538
CYCLE940 - Undercut
Function - DIN thread, 544
Function - Form E, 538
Function - Form F, 538
Function - Thread, 544
Parameter - DIN thread, 547
Parameter - Form E, 542
Parameter - Form F, 544
Parameter - Thread, 550
CYCLE951 - stock removal
Function, 530
Parameter, 534
CYCLE952 - Grooving
Function, 622
Input simple, 625
Parameter - input complete, 629
Parameter - input simple, 629
CYCLE952 - Grooving residual material
Function, 633
Parameter, 637
CYCLE952 - Plunge turning
Function, 637
Input simple, 639
Parameter - input complete, 643
Parameter - input simple, 644
CYCLE952 - Plunge turning residual material
Function, 647
Parameter, 651
CYCLE952 - Stock removal
Function, 604
Input simple, 609
Parameter - input complete, 614
Parameter - input simple, 614
CYCLE952 - Stock removal residual material
Function, 619
Parameter, 622
CYCLE98 - thread chain
Function, 578
Input simple, 579
Parameter - input complete, 583
Parameter - input simple, 584
CYCLE99 - thread turning
Parameter - face thread, input complete, 565
Parameter - face thread, input simple, 565
Parameter - longitudinal thread, complete input, 557
Parameter - longitudinal thread, simple input, 557
Parameter - tapered thread, input complete, 573
Parameter - tapered thread, input simple, 573
CYCLE99 - Thread turning
Function - Face thread, 550
Function - Longitudinal thread, 550
Function - Tapered thread, 550
Input simple, 551
Cycles
Current levels, 315
Hiding cycle parameters, 325
Screen forms, 315
Cylinder surface transformation
Function, 682
General programming, 684
Parameters, 685

D

Data block (SB2), 200
Deep-hole drilling 1 - CYCLE83
Function, 393
Input simple, 393
Parameter - input complete, 396
Parameter - input simple, 396
Deep-hole drilling 2 - CYCLE830
Function, 398
Hole entry, 399
Hole exit, 400
Input simple, 398
Parameter - input complete, 405
Parameter - input simple, 405
Through drilling, 399
With pilot hole, 399
With/without predrilling, 399
Delete
Multitool, 767

Deleting
Directory, 797
Program, 797
Determine demand, 819
Dictionary
Importing, 59
Directory
Copying, 795
Creating, 782
Deleting, 797
Highlight, 793
Inserting, 795
Naming convention, 782
Properties, 798
Selecting, 793
Display Manager
Operator controls, 90
DRF (handwheel offset), 216
Drill thread milling - CYCLE78
Function, 417
Parameter, 420
Drilling
Manual Machine, 867
Drilling - CYCLE82
Function, 388
Input simple, 389
Parameter - input simple, 391
Parameter input complete, 390
Drilling radius, 725
Drilling tool
Automatic measurement, 111
Manual measurement, 107
Measuring radius/diameter, 110
Measuring the length using the workpiece reference point, 108
Measuring the length with a fixed reference point, 109
Drives
Logical drive, 799
Setting up, 799
DRY (dry run feed), 216
Duplo number, (See sister tool number)
DXF file
Changing the section, 236
Cleaning, 234
close, 234
Delete Element, 240
Delete range, 240
Increasing / decreasing the section, 235
Machining plane, 239
Open, 234
Rotating the drawing, 236
Saving, 241
Select contour and accept, 244
Select the machining range, 240
Snap radius, 239
Specifying a reference point, 238
Transferring the drilling position, 242
Zero point, 238

E

Edges
Managing, 730
Editor
Calling, 221
Settings, 229
EES (execution from external storage), 802
Elongated hole - LONGHOLE
Function, 484
Parameter, 486
Energy analysis
Details, 899
Displaying, 899
Long time measurement, 903
Energy consumption
Displaying, 898
Measuring, 900
Energy-saving profiles, 904
Engraving - CYCLE60
Function, 490
Parameter, 496
EXTCALL call, 807

F

Face milling - CYCLE61
Function, 434
Parameter, 436
Face milling in JOG
Function, 187, 190
Parameter, 189, 192
Feed data
Actual value window, 45
Finger gestures, 73
Frame position pattern - CYCLE801
Function, 427
Parameter, 428
FTP drive, 776
Function
REF POINT, 99
REPOS, 99

Single block, 100
TEACH IN, 100
Function key block
"SINUMERIK Operate Generation 2" user interface, 77

G

G code program
Blank input, 318
Creating, 784
Programming variables, 326
G functions
Display all G groups, 265
Displaying selected G groups, 263
Mold making, 265
Global R parameters, 254
Global user variables, 257
Gloves, 72
Graphic tool display, 754
Graphic view
ShopMill, 330
Grid position pattern - CYCLE801
Function, 427
Parameter, 428
Groove - CYCLE930
Function, 534
Parameter, 538
Grooving - CYCLE952
Function, 622
Input simple, 625
Parameter - input complete, 629
Parameter - input simple, 629
Grooving residual material - CYCLE952
Function, 633
Parameter, 637

H

Handheld terminal 10, 890
Handheld terminal 8, 883
Handwheel
Assigning, 165
Headstock
Turning tool fixed, 664
Helix, 690
High Speed Settings - CYCLE832
Function, 668
Parameters, 672
Highlight
Directory, 793
Program, 793
HOLES1 - position pattern row
Parameter, 426
HOLES1 – row position pattern
Function, 426
HOLES2 - circle position pattern
Parameter, 430
HOLES2 – circle position pattern
Function, 429
HOLES2 - pitch circle position pattern
Parameter, 431
HOLES2 – pitch circle position pattern
Function, 429
HT 10
Acknowledgment button, 891
Overview, 890
Touch Panel, 895
User menu, 892
Virtual keyboard, 894
HT 8
Acknowledgment button, 884
Overview, 883
Touch Panel, 895
Traversing keys, 885
User menu, 886
Virtual keyboard, 888

I

IME
Chinese characters, 56
Korean characters, 60
Input simple
Circular spigot - CYCLE77, 455
Circumferential slot - SLOT2, 470
CYCLE82 - drilling, 389
CYCLE83 - deep-hole drilling 1, 393
CYCLE830 - Deep-hole drilling 2, 398
CYCLE99 - Thread turning, 551
Grooving - CYCLE952, 625
Longitudinal slot - SLOT1, 463
Milling contour pocket - CYCLE63, 517
Milling contour spigots - CYCLE63, 524
Multi-edge - CYCLE79, 459
Open slot – CYCLE899, 476
Plunge turning - CYCLE952, 639
Rectangular pocket - POCKET3, 437
Rectangular spigot - CYCLE76, 451
Stock removal - CYCLE952, 609

Tapping with compensating chuck - CYCLE840, 410
Tapping without compensating chuck - CYCLE84, 410
Thread chain - CYCLE98, 579
Interpolation turning
calling, 678
Interruption point
Approach, 210

J

Job list, 786

K

Key combinations - Simulation
Changing the section, 305
Enlarging/reducing a graphic, 303
Feedrate, 301
Moving the graphic, 304
Override, 301
Rotating the view, 304
Single block mode, 302

L

Layer selection, 234
Limit
Grooving - CYCLE952, 624
Plunge turning - CYCLE952, 638
Stock removal - CYCLE952, 607
Limit stops, 860
List of machining steps
Accepting the machining step, 252
Open, 249
Reference points, 250
Workpiece zero, 251
Zero point plane, 251
Loading
Multitool, 767
Local drive, 774
Creating an NC directory, 775
Logbook
Delete entries, 848
Displaying, 847
Edit the address data, 847
Entry search, 849
Making an entry, 848
Output, 846
Overview, 847
Long time measurement
Energy analysis, 903
LONGHOLE - elongated hole
Function, 484
Parameter, 486
Longitudinal slot - SLOT1
Function, 463
Input simple, 463
Parameter - input complete, 467
Parameter "Input simple", 467

M

Machine control panel
in the sidescreen, 86
Operator controls, 36
Machine functions, 349
Parameter, 349, 350
Machine model, 705
Machine-specific information, 846
Machining step program, 329
Machining times
Delete, 232
Display, 311, 331
Display in block display, 47, 201
Magazine
Deleting tools, 748
Loading tools, 748
Open, 744
Positioning, 746
Relocating tools, 748
Selecting, 733
Unloading tools, 748
Magazine management, 713
Manual Machine, 855
Angular milling, 862
Circle, 864
Contour milling, 869
Drilling, 867
Manual mode, 861
Measuring the tool, 857
Milling, 868
Set limit stop, 860
Simulating, 871
Single-cycle machining, 866
Straight line, 863
Traversing axes, 861
Work offset, 859
Workpiece zero, 858
Manual mode, 171
Circle, 864
Positioning axes, 179

Settings, 196
Spindle, 174
Straight line, 863
T, S, M windows, 172
Tool, 174
Traversing axes, 177
Unit of measurement, 172
Manual retraction, 185
Manually
Swiveling, 180
MDA
Deleting a program, 170
MDI
Executing a program, 169
Loading a program, 167
Saving a program, 168
Measurement result log
Settings, 151
Tool, 118
Workpiece zero, 146
Measuring
Drilling tool automatically, 111
Drilling tool manually, 107
Manual milling tool, 107
Milling tool automatically, 111
Turning tool automatically, 116
Turning tool manually, 115
Workpiece zero, 120
Measuring the tool
Manual Machine, 857
Messages
Displaying, 837
Sorting, 838
Milling
Manual Machine, 868
Milling contour pocket - CYCLE63
Function, 517
Input simple, 517
Parameter - input complete, 520
Parameter - input simple, 520
Milling contour spigots - CYCLE63
Function, 524
Input simple, 524
Parameter - input complete, 526
Parameter - input simple, 526
Milling tool
Automatic measurement, 111
Manual measurement, 107
Measuring radius/diameter, 110
Measuring the length using the workpiece reference point, 108
Measuring the length with a fixed reference point, 109
Milling/turning machine
ShopMill program header, 343
Mode groups, 101
Mold making view
adapting, 273
Changing a graphic, 276
Changing the section, 277
Editing a program block, 274
Programs, 271
Searching for program blocks, 275
Starting, 273
MRD (Measuring Result Display), 217
Multi-channel view, 697
"Machine" operating area, 698
OP015, OP019, 701
Settings, 703
Multi-edge - CYCLE79
Function, 459
Input simple, 459
Parameter - input complete, 461
Parameter - input simple, 461
Multiple clamping, 829
Program creation, 831
Settings in the program header, 830
Multitool, 762
Creating, 763
Delete, 767
Equipping tools, 765
Loading, 767
Parameters in the tool list, 762
Positioning, 770
Reactivating, 768
Relocating, 769
Removing tools, 766
Unloading, 768
Multitouch panel
SINUMERIK Operate Gen. 2, 71
Widescreen format, 80

N

Navigation bar
Sidescreen, 80
NC memory, 774
NC/PLC variables
Changing, 842
displaying, 840

New contour
Function - Milling, 499
Function - Turning, 594
Parameter - Milling, 500
Parameter - Turning, 596
Number of teeth, 725

O

Obstacle
Function, 694
Parameter, 695
Online help
Context-sensitive, 67
Open slot - CYCLE899
Parameter - input complete, 482
Parameter - input simple, 482
Open slot – CYCLE899
Function, 475
Input simple, 476
Opening tool data, 820
Operating areas, 25
Changing, 48
Selecting, 48
Operating mode
AUTO, 100
Changing, 48
JOG, 99, 171
MDI, 100
REPOS, 99
Operator controls
Display Manager, 90
Machine control panel, 36
Operator panel fronts, 26
Keys, 28

P

Pages, 80
Panel
cleaning, 65
Parameter
Backing up, 822
Calculating, 50
Changing, 50
Entering, 50
Face milling in JOG, 189, 192
Path milling - CYCLE72
Function, 507
Parameter, 512
Pitch circle position pattern - HOLES2
Parameter, 431
Pitch circle position pattern – HOLES2
Function, 429
Plunge turning - CYCLE952
Function, 637
Input simple, 639
Parameter - input complete, 643
Parameter - input simple, 644
Plunge turning residual material - CYCLE952
Function, 647
Parameter, 651
POCKET3 - rectangular pocket
Function, 436
Input simple, 437
Parameter - input complete, 441
Parameter - input simple, 441
POCKET4 - circular pocket
Function, 443
Input simple, 444
Parameter - input complete, 448
Parameter - input simple, 448
Polar coordinates, 691
Position pattern
Block search, 214
Positioning
Multitool, 770
Positions
display/hide, 431
Predrilling contour pocket - CYCLE64
Function - Centering, 514
Function - Predrilling, 514
Parameter - Centering, 515
Parameter - Predrilling, 516
Preview
Program, 792
Probe, 114
Calibration of length, 147
Calibration of radius / diameter, 147
Compare to sphere, 149
Electronic, 147, 149
Program
Closing, 778
Copying, 795
Correcting, 205
Creating with cycle support, 317
Deleting, 797
Editing, 221
Executing, 780
Highlight, 793
Inserting, 795
Multiple clamping, 831

Naming convention, 782
Open, 778
opening additional programs, 228
Preview, 792
Properties, 798
Renumbering blocks, 226
Replacing text, 223
Running-in, 200
Searching for a program position, 221
Selecting, 199, 793
Teach-in, 873
Program block
Changing, 355
Configuration, 335
Copying and inserting, 224
Creating, 346
Current, 47, 201
Delete, 224
insert, 224
linked, 335
Numbering, 225, 226
Repeat, 352
Search, 221
Selecting, 224
Program blocks, 227
Program control
activate, 217
Modes of operation, 216
Program correction, 205
Program header, 341
Important parameters, 347
Milling/turning machine, 343
Multiple clamping, 830
Program level, 203
Program list, 788
Program Manager, 771
Displaying HTML documents, 805
Displaying PDF documents, 805
Searching for directories and files, 790
Program runtime, 279
Program settings
Change, 356
Parameters, 356, 357
Program views
G code, 310
ShopMill, 330
Programmed stop 1, 216
Programmed stop 2, 216
Programming variables
G code program, 326
ShopTurn program, 364
Programs
Managing, 771
Properties
Directory, 798
Program, 798
Protection levels
Softkeys, 63
PRT (no axis motion), 216

Q

Quantity, 740

R

R parameters, 255
Backing up, 822
Reactivating
Multitool, 768
Reaming - CYCLE85
Function, 392
Parameter, 393
Record tools, 819
Rectangular pocket - POCKET3
Function, 436
Input simple, 437
Parameter - input complete, 441
Parameter - input simple, 441
Rectangular spigot - CYCLE76
Function, 450
Input simple, 451
Parameter - input complete, 453
Parameter - input simple, 453
Reference, 96
Relocating
Multitool, 769
Remote access
Allowing, 851
Setting, 850
Remote diagnostics, 850
Exit, 853
Requesting, 852
Repeat positions
Function, 433
Parameters, 433
Repositioning, 206
Retract, 185
RG0 (reduced rapid traverse), 216
Row position pattern - HOLES1
Parameter, 426

Row position pattern – HOLES1
Function, 426

S

Save
Parameter, 822
Setup data, 815
SB (single blocks), 216
SB1, 200
SB2, 200
SB3, 200
Screen layout, 76
Screenshots
Copying, 839
Creating, 839
Open, 839
Search
in the Program Manager, 790
Search mode, 212
Search pointer, 211
Searching
Logbook entry, 849
Selecting
Directory, 793
Program, 793
Setting actual values, (See setting zero offsets)
Setting milling tool - CYCLE800
Function, 662
Parameter, 664
Settings
Collision avoidance, 708
Editor, 229
For automatic operation, 281
For manual operation, 196
Measurement result log, 151
Multi-channel view, 703
Teach-in, 882
Tool lists, 761
Setup data
Backing up, 815
reading in, 817
ShopMill program
Creating, 340
Cutting edge, 347
Feedrate, 348
Machine functions, 349
Program blocks, 346
Program header, 341, 343
Program settings, 356
Program structure, 335
Radius compensation, 347
Spindle speed, 348
Straight line/Circle, 685
Tool, 347
ShopTurn program
Programming variables, 364
Shortcuts
Operator panel fronts, 28
Sidescreen
ABC keyboard, 86
MCP, 86
Navigation bar, 80
Operator controls, 80
Overview, 80
Pages, 86
Requirements, 80
Show, 82
Standard widgets, 82
Widgets, 80
Simulation, 285
Alarm display, 307
Blank, 300
Canceling, 293
Changing the feedrate, 301
Changing the graphic section, 305
Manual Machine, 871
Moving the graphic, 304
Non-modal, 302
Program control, 301
Rotating the graphic, 304
Showing and hiding the path display, 300
Start, 293
stopping, 293
Views, 296
Simultaneous recording, 287
Before machining, 294
Changing the graphic section, 305
Moving the graphic, 304
Rotating the graphic, 304
Single block
Coarse (SB1), 200
Fine (SB3), 200
SINUMERIK Operate Gen. 2
Screen layout, 76
Sister tool number, 724
Skip blocks, 217
SKP (skip blocks), 216
SLOT1 - longitudinal slot
Function, 463
Input simple, 463
Parameter - input complete, 467
Parameter "Input simple", 467

SLOT2 - circumferential slot
Function, 469
Input simple, 470
Parameter - input complete, 473
Parameter - input simple, 473
Special characters, 27
Spindle
positioning by CYCLE806, 677
Spindle data
Actual value window, 46
Spindle speed limitation, 163
Standard widget
Actual values, 82
Alarms, 83
Axle load, 84
Camera, 86
Service life, 85
Tool, 84
Variables, 84
Zero point, 83
Starting hole, 214
Startup, 95
Status display, 41
Stock removal - CYCLE951
Function, 530
Parameter, 534
Stock removal - CYCLE952
Function, 604
Input simple, 609
Parameter - input complete, 614
Parameter - input simple, 614
Stock removal in JOG
Function, 192
Parameter, 194, 195
Stock removal residual material - CYCLE952
Function, 619
Parameter, 622
Straight line, 687
Manual Machine, 863
Polar, 692
Straight line/Circle, 685
Subprogram
Function, 672
Parameters, 674
Switching over
Coordinate system, 103
Swiveling
in JOG - Function, 180
in JOG - Parameter, 184
Manually, 180
Swiveling - CYCLE800
Function, 652
Parameter, 660
Synchronization commands
Display, 312, 332
Synchronized actions
Displaying status, 268
System data
Displaying HTML documents, 805
Displaying PDF documents, 805

T

Tapping with compensating chuck - CYCLE840
Function, 410
Input simple, 410
Parameter - input complete, 415
Parameter - input simple, 415
Tapping without compensating chuck - CYCLE84
Function, 410
Input simple, 410
Parameter - input complete, 415
Parameter - input simple, 415
Teach-in, 873
Changing blocks, 876
Circle intermediate position CIP, 879
Continuous-path mode, 881
Deleting blocks, 877
General sequence, 873
Inserting a position, 879
Inserting blocks, 876
Motion type, 880
Parameter, 880
Rapid traverse G0, 879
Selecting, 875
Selecting a block, 877
Settings, 882
Traversing block G1, 879
Templates
Creating, 789
Storage locations, 789
Thread chain - CYCLE98
Function, 578
Input simple, 579
Parameter - input complete, 583
Parameter - input simple, 584
Thread milling - CYCLE70
Function, 486
Parameter, 490
Thread turning - CYCLE99
Function - Face thread, 550
Function - Longitudinal thread, 550

Function - Tapered thread, 550
Input simple, 551
Parameter - face thread, input complete, 565
Parameter - face thread, input simple, 565
Parameter - longitudinal thread, complete input, 557
Parameter - longitudinal thread, simple input, 557
Parameter - tapered thread, input complete, 573
Parameter - tapered thread, input simple, 573
Tip angle, 725
Tool
Change type, 753
Create, 728
Deleting, 731
Details, 749
Dimensioning, 717
Fixed point calibration, 111
Loading, 731
Measurement result log, 118
Measuring, 107
Multiple edges, 730
Reactivating, 741
Relocating, 746
Unloading, 731
Tool change
Coolant, 726
Tool data
Actual value window, 45
Backing up, 815
reading in, 817
Tool life, 740
Tool list, 724
Tool lists
Settings, 761
Tool management, 711
List filtering, 757
Sorting lists, 756
Tool parameters, 717
Tool probe, 114
Tool types, 714
Tool wear, 739
Tool wear list
Open, 739
Tools
Graphic display, 754
Touch operator controls
Cancel alarms, 78
Changing the channel, 78
Touch Panel
calibration, 895
Transformed view, 761
TTD file
Open, 820
Turning tool
Automatic measurement, 116
Fixed on the headstock, 664
Measure manually, 115

U

Undercut - CYCLE940
Function - DIN thread, 544
Function - Form E, 538
Function - Form F, 538
Function - Thread, 544
Parameter - DIN thread, 547
Parameter - Form E, 542
Parameter - Form F, 544
Parameter - Thread, 550
Unit of measurement
switching, 103
Unloading
Multitool, 768
USB drive, 776
User agreement, 97
User variables, 253
Activating, 261
Backing up, 822
Channel GUD, 258
Defining, 261
Global GUD, 257, 261
Global R parameters, 254
Local LUD, 259
Program PUD, 260
R parameters, 255
Searching, 260

V

Variable screen forms, 844
Virtual keyboard
"SINUMERIK Operate Generation 2" user interface, 78
HT 10, 894
HT 8, 888
Virtual keys
ABC keyboard, 80
MCP keys, 80

W

Wear, 740

Widgets, 80
Work offset
    Manual Machine, 859
Work offsets
    Calling, 351
    Delete, 159
    Overview, 153, 155
Work plan
    ShopMill, 330
Working area limitation, 162
Workpiece
    Creating, 783
    Starting machining, 197
    Stopping machining, 197, 198
Workpiece counter, 279
Workpiece zero
    Align edge, 127
    Aligning the plane, 143
    Automatic measurement, 120
    Changing the user interface, 145
    Manual Machine, 858
    Manual measurement, 120
    Measurement result log, 146
    Measuring, 160
    Measuring a circular spigot, 138
    Measuring a hole, 135
    Measuring a rectangular pocket, 135
    Measuring a rectangular spigot, 138
    Measuring a right-angled corner, 132
    Measuring any corner, 132
    Measuring the distance between two edges, 127

Z

Zero offsets
    Active ZO, 154
    Displaying details, 158
    Settable ZO, 157
    Setting, 105
Zero point
    DXF file, 238
Zero point settings
    Backing up, 815
    reading in, 817