

SIEMENS

SINUMERIK

SINUMERIK 840D sl/828D Turning

Operating Manual

Valid for:

SINUMERIK 840D sl / 840DE sl / 828D controllers

Software

CNC software for 840D sl / 840DE sl V4.5

SINUMERIK Operate for PCU/PC V4.5

02/2012

6FC5398-8CP40-3BA0

Preface

Introduction

1

Setting up the machine

2

Working in manual mode

3

Machining the workpiece

4

Simulating machining

5

Creating a G code program

6

Creating a ShopTurn
program

7

Programming technology
functions (cycles)

8

Multi-channel view

9

Multi-channel support

10

User variables

11

Teaching in a program

12

Working with Manual
Machine

13

Working with a B axis

14

Working with two tool
carriers

15

Tool management

16

Continued on next page

SINUMERIK 840D sl/828D Turning

Operating Manual




Continuation

<u>Managing programs</u>	17
<u>Setting up drives</u>	18
<u>HT 8</u>	19
<u>Ctrl-Energy</u>	20
<u>Easy Message (828D only)</u>	21
<u>Easy Extend (828D only)</u>	22
<u>Service Planner (828D only)</u>	23
<u>Ladder Viewer and Ladder add-on (828D only)</u>	24
<u>Alarm, error and system messages</u>	25
<u>Appendix</u>	A

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
with a safety alert symbol, indicates that minor personal injury can result if proper precautions are not taken.
CAUTION
without a safety alert symbol, indicates that property damage can result if proper precautions are not taken.
NOTICE
indicates that an unintended result or situation can occur if the relevant information is not taken into account.


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.

Preface

SINUMERIK documentation

The SINUMERIK documentation is organized in the following categories:

- General documentation
- User documentation
- Manufacturer/service documentation

Additional information

You can find information on the following topics at www.siemens.com/motioncontrol/docu:

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

Please send any questions about the technical documentation (e.g. suggestions for improvement, corrections) to the following address:

docu.motioncontrol@siemens.com

My Documentation Manager (MDM)

Under the following link you will find information to individually compile OEM-specific machine documentation based on the Siemens content:

www.siemens.com/mdm

Training

For information about the range of training courses, refer under:

- www.siemens.com/sitrain
SITRAIN - Siemens training for products, systems and solutions in automation technology
- www.siemens.com/sinustrain
SinuTrain - training software for SINUMERIK

FAQs

You can find Frequently Asked Questions in the Service&Support pages under Product Support. <http://support.automation.siemens.com>

SINUMERIK

You can find information on SINUMERIK under the following link:

www.siemens.com/sinumerik

Target group

This documentation is intended for users of turning 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, such as the tapping cycle, is a subprogram defined in SINUMERIK Operate for executing a frequently repeated machining operation.

Technical Support

You will find telephone numbers for other countries for technical support in the Internet under <http://www.siemens.com/automation/service&support>

Table of contents

	Preface	5
1	Introduction.....	23
1.1	Product overview	23
1.2	Operator panel fronts	25
1.2.1	Overview	25
1.2.2	Keys of the operator panel.....	28
1.3	Machine control panels	37
1.3.1	Overview	37
1.3.2	Controls on the machine control panel	37
1.4	User interface.....	40
1.4.1	Screen layout	40
1.4.2	Status display.....	41
1.4.3	Actual value window	44
1.4.4	T,F,S window	45
1.4.5	Current block display	47
1.4.6	Operation via softkeys and buttons	47
1.4.7	Entering or selecting parameters	49
1.4.8	Pocket calculator.....	51
1.4.9	Context menu.....	53
1.4.10	Touch operation	53
1.4.11	Changing the user interface language.....	54
1.4.12	Entering Asian characters.....	54
1.4.13	Protection levels.....	57
1.4.14	Online help in SINUMERIK Operate.....	59
2	Setting up the machine	63
2.1	Switching on and switching off.....	63
2.2	Approaching a reference point.....	64
2.2.1	Referencing axes	64
2.2.2	User agreement	65
2.3	Modes and mode groups	67
2.3.1	General	67
2.3.2	Modes groups and channels.....	69
2.3.3	Channel switchover.....	69
2.4	Settings for the machine	71
2.4.1	Switching over the coordinate system (MCS/WCS)	71
2.4.2	Switching the unit of measurement.....	72
2.4.3	Setting the zero offset.....	73
2.5	Measuring the tool	75
2.5.1	Measuring a tool manually	75
2.5.2	Measuring a tool with a tool probe	77
2.5.3	Calibrating the tool probe.....	79

2.5.4	Measuring a tool with a magnifying glass	81
2.6	Measuring the workpiece zero	82
2.7	Zero offsets	84
2.7.1	Display active zero offset	86
2.7.2	Displaying the zero offset "overview"	87
2.7.3	Displaying and editing base zero offset	88
2.7.4	Displaying and editing settable zero offset	89
2.7.5	Displaying and editing details of the zero offsets	90
2.7.6	Deleting a zero offset	92
2.7.7	Measuring the workpiece zero	93
2.8	Monitoring axis and spindle data	94
2.8.1	Specify working area limitations	94
2.8.2	Editing spindle data	95
2.8.3	Spindle chuck data	96
2.9	Displaying setting data lists	99
2.10	Handwheel assignment	100
2.11	MDA	102
2.11.1	Loading an MDA program from the Program Manager	102
2.11.2	Saving an MDA program	103
2.11.3	Executing an MDA program	104
2.11.4	Deleting an MDA program	104
3	Working in manual mode	105
3.1	General	105
3.2	Selecting a tool and spindle	105
3.2.1	T,S,M window	105
3.2.2	Selecting a tool	107
3.2.3	Starting and stopping the spindle manually	108
3.2.4	Positioning the spindle	109
3.3	Traversing axes	110
3.3.1	Traverse axes by a defined increment	111
3.3.2	Traversing axes by a variable increment	112
3.4	Positioning axes	113
3.5	Manual retraction	114
3.6	Simple stock removal of workpiece	116
3.7	Thread synchronizing	119
3.8	Default settings for manual mode	121
4	Machining the workpiece	123
4.1	Starting and stopping machining	123
4.2	Selecting a program	125
4.3	Executing a trail program run	126
4.4	Displaying the current program block	128
4.4.1	Current block display	128

4.4.2	Displaying a basic block.....	128
4.4.3	Display program level	129
4.5	Correcting a program	131
4.6	Repositioning axes.....	133
4.7	Starting machining at a specific point	135
4.7.1	Use block search	135
4.7.2	Continuing program from search target	137
4.7.3	Simple search target definition.....	137
4.7.4	Defining an interruption point as search target.....	138
4.7.5	Entering the search target via search pointer.....	139
4.7.6	Parameters for block search in the search pointer	140
4.7.7	Block search mode	140
4.8	Controlling the program run	143
4.8.1	Program control.....	143
4.8.2	Skip blocks.....	145
4.9	Overstore	147
4.10	Editing a program.....	149
4.10.1	Searching in programs.....	150
4.10.2	Replacing program text.....	151
4.10.3	Copying/pasting/deleting a program block.....	153
4.10.4	Renumbering a program.....	154
4.10.5	Creating a program block.....	155
4.10.6	Opening additional programs.....	156
4.10.7	Editor settings	157
4.11	Mold making view	159
4.11.1	Mold making view	159
4.11.2	Starting the mold making view.....	161
4.11.3	Specifically jump to the program block	162
4.11.4	Searching for program blocks	163
4.11.5	Changing the view	164
4.11.5.1	Enlarging or reducing the graphical representation	164
4.11.5.2	Modifying the viewport	165
4.12	Displaying G functions and auxiliary functions	166
4.12.1	Selected G functions.....	166
4.12.2	All G functions.....	168
4.12.3	Auxiliary functions	169
4.13	Displaying the program runtime and counting workpieces	172
4.14	Setting for automatic mode.....	174
5	Simulating machining.....	177
5.1	Overview	177
5.2	Simulation before machining of the workpiece	183
5.3	Simultaneous recording before machining of the workpiece	185
5.4	Simultaneous recording during machining of the workpiece	186
5.5	Different views of the workpiece	187

5.5.1	Side view	187
5.5.2	Half section	187
5.5.3	Face view	188
5.5.4	3D view.....	188
5.5.5	2-window	189
5.6	Graphical display.....	190
5.7	Editing the simulation display	191
5.7.1	Blank display	191
5.7.2	Showing and hiding the tool path	191
5.8	Program control during the simulation	192
5.8.1	Changing the feedrate.....	192
5.8.2	Simulating the program block by block	193
5.9	Editing and adapting a simulation graphic	194
5.9.1	Enlarging or reducing the graphical representation	194
5.9.2	Panning a graphical representation	195
5.9.3	Rotating the graphical representation	195
5.9.4	Modifying the viewport	196
5.9.5	Defining cutting planes.....	197
5.10	Displaying simulation alarms.....	198
6	Creating a G code program	199
6.1	Graphical programming.....	199
6.2	Program views	200
6.3	Program structure	203
6.4	Fundamentals	204
6.4.1	Machining planes	204
6.4.2	Current planes in cycles and input screens	204
6.4.3	Programming a tool (T)	205
6.5	Generating a G code program	206
6.6	Blank input	208
6.6.1	Function.....	208
6.7	Machining plane, milling direction, retraction plane, safe clearance and feedrate (PL, RP, SC, F).....	210
6.8	Selection of the cycles via softkey	211
6.9	Calling technology cycles.....	216
6.9.1	Hiding cycle parameters.....	216
6.9.2	Setting data for cycles.....	216
6.9.3	Checking cycle parameters.....	217
6.9.4	Programming variables	217
6.9.5	Changing a cycle call	218
6.9.6	Compatibility for cycle support	218
6.9.7	Additional functions in the input screens.....	219
6.10	Measuring cycle support	220

7	Creating a ShopTurn program	221
7.1	Graphic program control, ShopTurn programs	221
7.2	Program views	222
7.3	Program structure	226
7.4	Fundamentals	227
7.4.1	Machining planes	227
7.4.2	Machining cycle, approach/retraction	229
7.4.3	Absolute and incremental dimensions	231
7.4.4	Polar coordinates	233
7.4.5	Clamping the spindle	234
7.5	Creating a ShopTurn program	235
7.6	Program header	237
7.7	Generating program blocks.....	239
7.8	Tool, offset value, feedrate and spindle speed (T, D, F, S, V).....	240
7.9	Call work offsets.....	243
7.10	Repeating program blocks	244
7.11	Entering the number of workpieces	246
7.12	Changing program blocks	247
7.13	Changing program settings.....	248
7.14	Selection of the cycles via softkey	250
7.15	Calling technology functions	255
7.15.1	Additional functions in the input screens	255
7.15.2	Checking cycle parameters.....	255
7.15.3	Programming variables	256
7.15.4	Setting data for technological functions	256
7.15.5	Changing a cycle call.....	257
7.15.6	Compatibility for cycle support.....	257
7.16	Programming the approach/retraction cycle	258
7.17	Measuring cycle support.....	260
7.18	Example: Standard machining	261
7.18.1	Workpiece drawing	262
7.18.2	Programming.....	263
7.18.3	Results/simulation test	280
7.18.4	G code machining program.....	282
8	Programming technology functions (cycles)	285
8.1	Drilling	285
8.1.1	General	285
8.1.2	Centering (CYCLE81)	286
8.1.2.1	Function	286
8.1.3	Drilling (CYCLE82).....	289
8.1.4	Reaming (CYCLE85)	291
8.1.5	Boring (CYCLE86)	293

8.1.6	Deep-hole drilling (CYCLE83).....	296
8.1.7	Tapping (CYCLE84, 840).....	300
8.1.8	Drill and thread milling (CYCLE78).....	306
8.1.9	Positions and position patterns	310
8.1.9.1	Approach/retraction.....	310
8.1.10	Arbitrary positions (CYCLE802).....	311
8.1.10.1	Function.....	311
8.1.11	Position pattern line (HOLES1), grid or frame (CYCLE801).....	314
8.1.12	Circle position pattern (HOLES2).....	317
8.1.13	Displaying and hiding positions.....	320
8.1.14	Repeating positions.....	322
8.1.14.1	Function.....	322
8.2	Rotate.....	323
8.2.1	General.....	323
8.2.2	Stock removal (CYCLE951).....	323
8.2.3	Groove (CYCLE930).....	327
8.2.4	Undercut form E and F (CYCLE940).....	330
8.2.5	Thread undercuts (CYCLE940).....	333
8.2.6	Thread turning (CYCLE99).....	336
8.2.7	Thread chain (CYCLE98).....	346
8.2.8	Cut-off (CYCLE92).....	349
8.3	Contour turning	351
8.3.1	General information.....	351
8.3.2	Representation of the contour.....	352
8.3.3	Creating a new contour.....	354
8.3.4	Creating contour elements.....	356
8.3.5	Entering the master dimension	363
8.3.6	Changing the contour.....	364
8.3.7	Contour call (CYCLE62) - only for G code program	365
8.3.8	Stock removal (CYCLE952).....	366
8.3.9	Stock removal rest (CYCLE952).....	372
8.3.10	Plunge-cutting (CYCLE952).....	374
8.3.11	Plunge-cutting rest (CYCLE952).....	378
8.3.12	Plunge-turning (CYCLE952).....	380
8.3.13	Plunge-turning rest (CYCLE952).....	384
8.4	Milling	386
8.4.1	Face milling (CYCLE61).....	386
8.4.2	Rectangular pocket (POCKET3).....	389
8.4.3	Circular pocket (POCKET4).....	394
8.4.4	Rectangular spigot (CYCLE76).....	400
8.4.5	Circular spigot (CYCLE77).....	404
8.4.6	Multi-edge (CYCLE79).....	408
8.4.7	Longitudinal groove (SLOT1).....	411
8.4.8	Circumferential groove (SLOT2).....	416
8.4.9	Open groove (CYCLE899).....	420
8.4.10	Long hole (LONGHOLE) - only for G code program.....	428
8.4.11	Thread milling (CYCLE70).....	430
8.4.12	Engraving (CYCLE60).....	434
8.5	Contour milling	442
8.5.1	General information.....	442

8.5.2	Representation of the contour.....	442
8.5.3	Creating a new contour.....	444
8.5.4	Creating contour elements.....	446
8.5.5	Changing the contour.....	451
8.5.6	Contour call (CYCLE62) - only for G code program.....	452
8.5.7	Path milling (CYCLE72).....	453
8.5.8	Contour pocket/contour spigot (CYCLE63/64).....	459
8.5.9	Predrilling contour pocket (CYCLE64).....	460
8.5.10	Milling contour pocket (CYCLE63).....	465
8.5.11	Contour pocket residual material (CYCLE63, option).....	468
8.5.12	Milling contour spigot (CYCLE63).....	471
8.5.13	Contour spigot residual material (CYCLE63, option).....	474
8.6	Further cycles and functions.....	477
8.6.1	Swiveling plane / aligning tool (CYCLE800).....	477
8.6.2	Swiveling tool (CYCLE800).....	485
8.6.2.1	Aligning turning tools - only for G code program (CYCLE800).....	485
8.6.2.2	Aligning milling tools - only for G code program (CYCLE800).....	487
8.6.2.3	Preloading milling tools - only for G code program (CYCLE800).....	488
8.6.3	High-speed settings (CYCLE832).....	490
8.6.4	Subroutines.....	493
8.7	Additional cycles and functions in ShopTurn.....	495
8.7.1	Drilling centric.....	495
8.7.2	Thread centered.....	497
8.7.3	Transformations.....	500
8.7.4	Translation.....	501
8.7.5	Rotation.....	502
8.7.6	Scaling.....	503
8.7.7	Mirroring.....	504
8.7.8	Rotation C.....	505
8.7.9	Straight and circular machining.....	506
8.7.10	Selecting a tool and machining plane.....	507
8.7.11	Programming a straight line.....	508
8.7.12	Programming a circle with known center point.....	511
8.7.13	Programming a circle with known radius.....	513
8.7.14	Polar coordinates.....	515
8.7.15	Straight line polar.....	517
8.7.16	Circle polar.....	519
8.7.17	Machining with the counterspindle.....	521
8.7.17.1	Programming example: Machining main spindle – Transfer workpiece – Machining counterspindle.....	522
8.7.17.2	Programming example: Machining, counterspindle - without previous transfer.....	523
8.7.17.3	Programming example: Machining bar material.....	523
8.7.17.4	Parameters, counterspindle.....	524
9	Multi-channel view.....	529
9.1	Multi-channel view.....	529
9.2	Multi-channel view in the "Machine" operating area.....	529
9.3	Multi-channel view for large operator panels.....	533
9.4	Setting the multi-channel view.....	535

10	Multi-channel support	537
10.1	Working with several channels	537
10.2	Multi-channel editor	538
10.2.1	Creating a multi-channel program.....	538
10.2.2	Entering multi-channel data	539
10.2.3	Editing the multi-channel program	542
10.2.3.1	Changing the job list.....	542
10.2.3.2	Editing a G code multi-channel program.....	543
10.2.3.3	Editing a ShopTurn multi-channel program	545
10.2.3.4	Creating a program block.....	553
10.2.4	Synchronizing programs	556
10.2.5	Optimizing the machining time	559
10.3	Simulating machining	561
10.3.1	Simulation.....	561
10.3.2	Different workpiece views for multi-channel support	562
10.4	Display/edit the multi-channel functionality in the "Machine" operating area	564
10.4.1	Running-in a program	564
10.4.2	Block search and program control	565
10.5	Multi-channel functionality for large operator panels	567
10.6	Setting the multi-channel function.....	570
10.7	Synchronizing a counterspindle	572
11	User variables.....	577
11.1	Overview	577
11.2	R parameters	578
11.3	Displaying global user data (GUD)	579
11.4	Displaying channel GUDs	581
11.5	Displaying local user data (LUD)	583
11.6	Displaying program user data (PUD).....	584
11.7	Searching for user variables	585
11.8	Defining and activating user variables	586
12	Teaching in a program.....	589
12.1	Overview	589
12.2	General sequence.....	590
12.3	Inserting a block	591
12.3.1	Input parameters for teach-in blocks.....	591
12.4	Teach-in via Windows	594
12.4.1	General.....	594
12.4.2	Teach in rapid traverse G0.....	595
12.4.3	Teach in straight G1.....	595
12.4.4	Teaching in circle intermediate and circle end point CIP	596
12.4.5	Teach-in A spline	596

12.5	Editing a block.....	598
12.6	Selecting a block.....	599
12.7	Deleting a block	600
12.8	Settings for teach-in	601
13	Working with Manual Machine	603
13.1	Manual Machine.....	603
13.2	Measuring the tool	605
13.3	Setting the zero offset.....	605
13.4	Set limit stop.....	606
13.5	Simple workpiece machining	607
13.5.1	Traversing axes	607
13.5.2	Taper turning.....	608
13.5.3	Straight and circular machining.....	609
13.5.3.1	Straight turning.....	609
13.5.3.2	Circular turning.....	610
13.6	More complex machining	612
13.6.1	Drilling with Manual Machine	613
13.6.2	Turning with manual machine	614
13.6.3	Milling with Manual Machine	615
13.7	Simulation and simultaneous recording.....	617
14	Working with a B axis	619
14.1	Lathes with B axis	619
14.2	Tool alignment for turning	622
14.3	Milling with a B axis.....	623
14.4	Swiveling	624
14.5	Approach/retraction.....	626
14.6	Position pattern	628
14.7	Tool selection for the manual mode.....	630
14.8	Measuring a tool with the B axis	632
15	Working with two tool carriers	633
15.1	Programming with two tool carriers	633
15.2	Measure tool	635
16	Tool management.....	637
16.1	Lists for the tool management.....	637
16.2	Magazine management	639
16.3	Tool types.....	640
16.4	Tool dimensioning.....	643
16.5	Tool list.....	648

16.5.1	Additional data	652
16.5.2	Creating a new tool	653
16.5.3	Measuring the tool.....	655
16.5.4	Managing several cutting edges	656
16.5.5	Delete tool	657
16.5.6	Loading and unloading tools	657
16.5.7	Selecting a magazine.....	659
16.6	Tool wear.....	661
16.6.1	Reactivate tool	664
16.7	Tool data OEM	665
16.8	Magazine.....	667
16.8.1	Positioning a magazine	669
16.8.2	Relocating a tool	670
16.8.3	Unload all tools.....	671
16.9	Sorting tool management lists.....	672
16.10	Filtering the tool management lists	673
16.11	Specific search in the tool management lists.....	674
16.12	Displaying tool details	676
16.13	Changing the cutting edge position or tool type.....	678
16.14	Settings for tool lists	679
16.15	Working with multitool	681
16.15.1	Tool list for multitool	682
16.15.2	Create multitool	683
16.15.3	Equipping multitool with tools.....	685
16.15.4	Removing a tool from multitool	686
16.15.5	Delete multitool	686
16.15.6	Loading and unloading multitool	687
16.15.7	Reactivating the multitool.....	688
16.15.8	Relocating a multitool.....	690
16.15.9	Positioning multitool	691
17	Managing programs.....	693
17.1	Overview	693
17.1.1	NC memory	696
17.1.2	Local drive	697
17.1.3	USB drives	698
17.2	Opening and closing the program.....	699
17.3	Executing a program	701
17.4	Creating a directory / program / job list / program list	703
17.4.1	Creating a new directory	703
17.4.2	Creating a new workpiece.....	704
17.4.3	Creating a new G code program.....	705
17.4.4	New ShopTurn program.....	706
17.4.5	Storing any new file.....	707
17.4.6	Creating a job list	708
17.4.7	Creating a program list.....	710

17.5	Creating templates	711
17.6	Searching directories and files	712
17.7	Displaying the program in the Preview.	714
17.8	Selecting several directories/programs	715
17.9	Copying and pasting a directory/program	717
17.10	Deleting a directory/program	719
17.11	Changing file and directory properties	720
17.12	Viewing PDF documents	722
17.13	EXTCALL	724
17.14	Backing up data	727
17.14.1	Generating an archive in the Program Manager	727
17.14.2	Generating an archive via the system data	728
17.14.3	Reading in an archive in the Program Manager	731
17.14.4	Read in archive from system data	732
17.15	Setup data	734
17.15.1	Backing up setup data	734
17.15.2	Reading-in set-up data	737
17.16	RS-232-C	739
17.16.1	Reading-in and reading-out archives	739
17.16.2	Setting V24 in the program manager	741
18	Setting up drives	743
18.1	Overview	743
18.2	Setting up drives	744
19	HT 8	747
19.1	HT 8 overview	747
19.2	Traversing keys	750
19.3	Machine control panel menu	752
19.4	Virtual keyboard	754
19.5	Calibrating the touch panel	756
20	Ctrl-Energy	759
20.1	Overview	759
20.2	Displaying energy consumption	760
20.3	Measuring and saving the energy consumption	761
20.4	Long-term measurement of the energy consumption	763
20.5	Displaying measured curves	764
20.6	Using the energy-saving profile	765

21	Easy Message (828D only)	767
21.1	Overview	767
21.2	Activating Easy Message	769
21.3	Creating/editing a user profile	770
21.4	Setting-up events	772
21.5	Logging an active user on and off.....	774
21.6	Displaying SMS logs	775
21.7	Making settings for Easy Message	776
22	Easy Extend (828D only)	777
22.1	Overview	777
22.2	Enabling a device.....	778
22.3	Activating and deactivating a device.....	779
22.4	Commissioning Easy Extend	780
23	Service Planner (828D only)	781
23.1	Performing and monitoring maintenance tasks.....	781
23.2	Set maintenance tasks.....	783
24	Ladder Viewer and Ladder add-on (828D only)	785
24.1	PLC diagnostics	785
24.2	Structure of the user interface.....	786
24.3	Control options	788
24.4	Displaying PLC properties.....	790
24.5	Displaying information on the program blocks	791
24.6	Displaying and editing NC/PLC variables	794
24.7	Downloading a PLC user program.....	795
24.8	Editing the local variable table	796
24.9	Creating a new block.....	798
24.10	Editing block properties.....	799
24.11	Inserting and editing networks	800
24.12	Editing network properties.....	802
24.13	Displaying/canceling the access protection	803
24.14	Displaying and editing symbol tables.....	804
24.15	Searching for operands.....	805
24.16	Inserting/deleting a symbol table	807
24.17	Displaying the network symbol information table.....	808

24.18	Displaying and editing PLC signals	809
24.19	Displaying cross references.....	810
25	Alarm, error and system messages	813
25.1	Displaying alarms.....	813
25.2	Displaying an alarm log.....	815
25.3	Displaying messages	816
25.4	Sorting, alarms, faults and messages.....	817
25.5	PLC and NC variables	818
25.5.1	Displaying and editing PLC and NC variables.....	818
25.5.2	Saving and loading screen forms	822
25.5.3	Load symbols.....	823
25.6	Version	824
25.6.1	Displaying version data.....	824
25.6.2	Save information	825
25.7	Logbook	827
25.7.1	Displaying and editing the logbook	827
25.7.2	Making a logbook entry.....	828
25.8	Creating screenshots	830
25.9	Remote diagnostics	831
25.9.1	Setting remote access	831
25.9.2	Permit modem.....	833
25.9.3	Request remote diagnostics	833
25.9.4	Exit remote diagnostics.....	834
A	Appendix.....	835
A.1	Overview	835
	Index.....	837

Introduction

1.1 Product overview

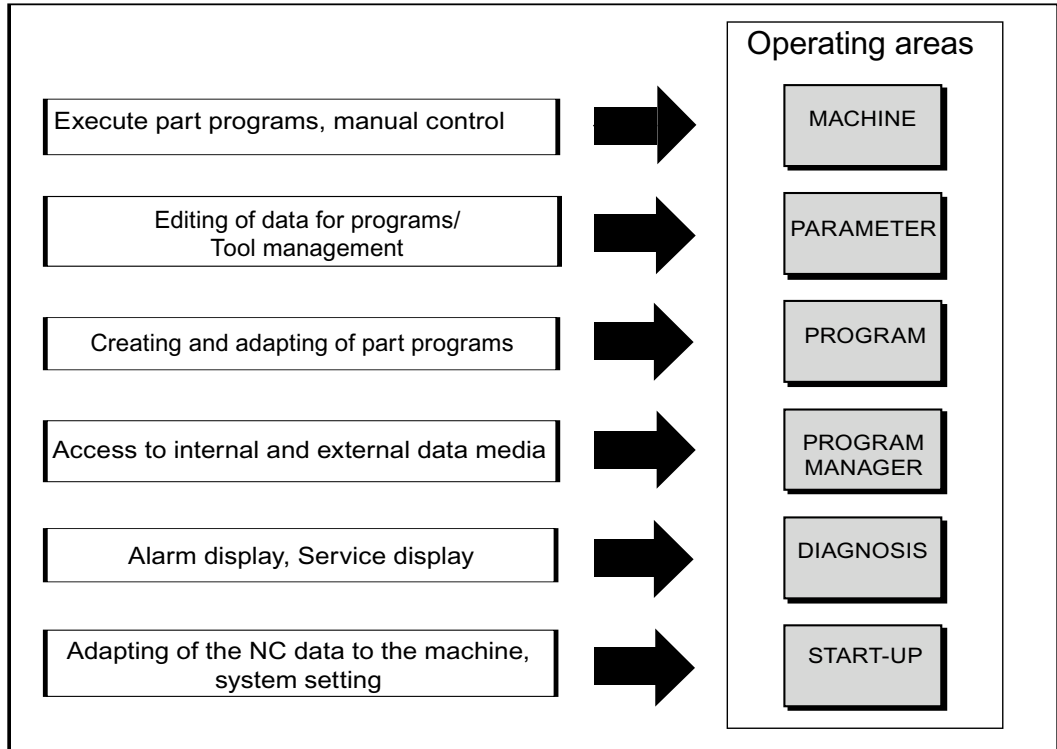
The SINUMERIK controller is a CNC (Computerized Numerical Controller) for machine tools.

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

- Creation and adaptation of part programs
- Execution of part programs
- Manual control
- Access to internal and external data media
- Editing of data for programs
- Management of tools, zero points and further user data required in programs
- Diagnostics of controller and machine

Operating areas

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



1.2 Operator panel fronts

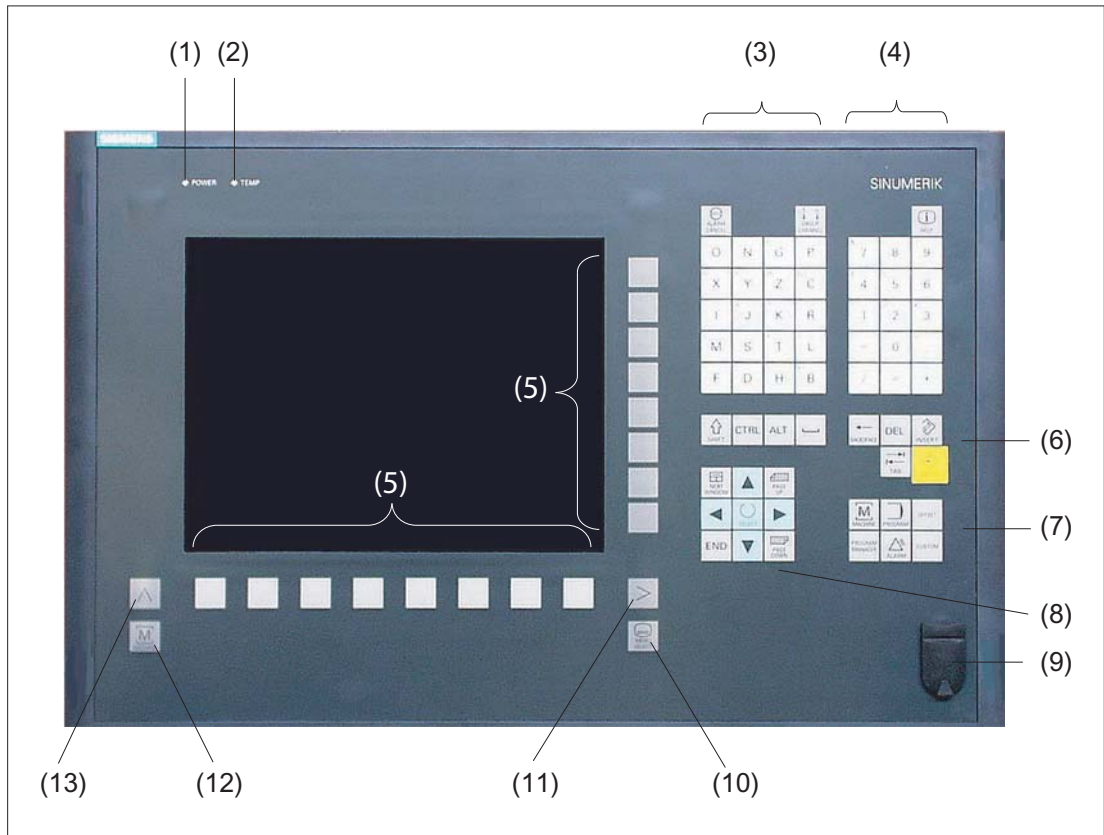
1.2.1 Overview

Introduction

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

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.

Operator controls and indicators



- 1 Status LED: POWER
- 2 Status LED: TEMP
(illuminated LEDs indicate increased wear)
- 3 Alphabetic key group
- 4 Numerical key group
- 5 Softkeys
- 6 Control key group
- 7 Hotkey group
- 8 Cursor key group
- 9 USB interface
- 10 Menu select key
- 11 Menu forward button
- 12 Machine area button
- 13 Menu back key

Figure 1-1 View of OP 010 operator panel front

References









A more precise description as well as a view of the other operator panel fronts that can be used may be found in the following reference:

















Operator Components and Networking Manual; SINUMERIK 840D sl/840Di sl











1.2.2 Keys of the operator panel














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

















Keys and key combinations


Key	Function
	<p><ALARM CANCEL> Cancels alarms and messages that are marked with this symbol.</p>
	<p><CHANNEL> Advances for several channels.</p>
	<p><HELP> Calls the context-sensitive online help for the selected window.</p>
	<p><NEXT WINDOW> *</p> <ul style="list-style-type: none"> • 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.
 + 	<p>* on USB keyboards use the <Home> or <Pos 1> key</p> <p><NEXT WINDOW> + <SHIFT></p> <ul style="list-style-type: none"> • 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.
 + 	<p><NEXT WINDOW> + <ALT></p> <ul style="list-style-type: none"> • Moves the cursor to the first object. • Moves the cursor in the first column of a table row. • Moves the cursor to the beginning of a program block.

Key	Function
 + 	<NEXT WINDOW> + <CTRL> <ul style="list-style-type: none"> Moves the cursor to the beginning of a program. Moves the cursor in the first row of the current column.
 +  + 	<NEXT WINDOW> + <CTRL> + <SHIFT> <ul style="list-style-type: none"> Moves the cursor to the beginning of a program. Moves the cursor in 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> <ul style="list-style-type: none"> Editing box <ul style="list-style-type: none"> Opens a directory or program (e.g. cycle) in the editor. Navigation <ul style="list-style-type: none"> Moves the cursor further to the right by one character.










Key	Function
 + 	<p><Cursor right> + <CTRL></p> <ul style="list-style-type: none"> • 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.
	<p><Cursor left></p> <ul style="list-style-type: none"> • 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.
 + 	<p><Cursor left> + <CTRL></p> <ul style="list-style-type: none"> • 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.
	<p><Cursor up></p> <ul style="list-style-type: none"> • Editing box Moves the cursor into the next upper field. • Navigation <ul style="list-style-type: none"> – Moves the cursor in a table to the next cell upwards. – Moves the cursor upwards in a menu screen.
 + 	<p><Cursor up> + <CTRL></p> <ul style="list-style-type: none"> • Moves the cursor in a table to the beginning of the table. • Moves the cursor to the beginning of a window.
 + 	<p><Cursor up> + <SHIFT></p> <p>In the program manager and in the program editor, selects a contiguous selection of directories and program blocks.</p>










Key	Function
	<p><Cursor down></p> <ul style="list-style-type: none"> Editing box <ul style="list-style-type: none"> Moves the cursor downwards. Navigation <ul style="list-style-type: none"> Moves the cursor in a table to the next cell downwards. Moves the cursor in a window downwards.
 + 	<p><Cursor down> + <CTRL></p> <ul style="list-style-type: none"> Navigation <ul style="list-style-type: none"> Moves the cursor in a table to the end of the table. Moves the cursor to the end of a window. Simulation <ul style="list-style-type: none"> Reduces the override.
 + 	<p><Cursor down> + <SHIFT></p> <p>In the program manager and in the program editor, selects a contiguous selection of directories and program blocks.</p>
	<p><SELECT></p> <p>Switches between several specified options in selection drop-down list boxes and in selection boxes.</p> <p>Activates checkboxes.</p> <p>In the program editor and in the program manager, selects a program block or a program.</p>
 + 	<p><SELECT> + <CTRL></p> <p>When selecting table rows, switches between selected and not selected.</p>
 + 	<p><SELECT> + <SHIFT></p> <p>Selects in selection lists and in selection fields the previous entry or the last entry.</p>
	<p><END></p> <p>Moves the cursor to the last entry field in a window, to the end of a table or a program block.</p> <p>Selects the last entry in selection lists and in selection fields.</p>
 + 	<p><END> + <SHIFT></p> <p>Moves the cursor to the last entry.</p> <p>Selects a contiguous selection from the cursor position up to the end of a program block.</p>

Key	Function
 + 	<p><END> + <CTRL></p> <p>Moves the cursor to the last entry in the last line of the actual column or to the end of a program.</p>
 +  + 	<p><END> + <CTRL> + <SHIFT></p> <p>Moves the cursor to the last entry in the last line of the actual column or to the end of a program.</p> <p>Selects a contiguous selection from the cursor position up to the end of a program block.</p>
	<p><BACKSPACE></p> <ul style="list-style-type: none"> Editing box <ul style="list-style-type: none"> Deletes a character selected to the left of the cursor. Navigation <ul style="list-style-type: none"> Deletes all of the selected characters to the left of the cursor.
 + 	<p><BACKSPACE> + <CTRL></p> <ul style="list-style-type: none"> Editing window <ul style="list-style-type: none"> Deletes a word selected to the left of the cursor. Navigation <ul style="list-style-type: none"> Deletes all of the selected characters to the left of the cursor.
	<p><TAB></p> <ul style="list-style-type: none"> In the program editor, indents the cursor by one character. In the program manager, moves the cursor to the next entry to the right.
 + 	<p><TAB> + <SHIFT></p> <ul style="list-style-type: none"> In the program editor, indents the cursor by one character. In the program manager, moves the cursor to the next entry to the left.
 + 	<p><TAB> + <CTRL></p> <ul style="list-style-type: none"> In the program editor, indents the cursor by one character. In the program manager, moves the cursor to the next entry to the right.
 +  + 	<p><TAB> + <SHIFT> + <CTRL></p> <ul style="list-style-type: none"> In the program editor, indents the cursor by one character. In the program manager, moves the cursor to the next entry to the left.

Key	Function
CTRL + A	<CTRL> + <A> In the actual window, selects all entries (only in the program editor and program manager).
CTRL + C	<CTRL> + <C> Copies the selected content.
CTRL + E	<CTRL> + <E> Calls the "Ctrl Energy" function.
CTRL + F	<CTRL> + <F> Opens the search dialog in the machine data and setting data lists, when loading and saving in the MDA editor as well as in the program manager and in the system data.
CTRL + L	<CTRL> + <L> Scrolls the actual user interface through all installed languages one after the other.
CTRL +  + L	<CTRL> + <SHIFT> + <L> Scrolls the actual operator interface through all installed languages in the inverse sequence.
CTRL + P	<CTRL> + <P> Generates a screenshot from the actual operator interface and saves it as file.
CTRL + S	<CTRL> + <S> Switches the single block in or out in the simulation.
CTRL + V	<CTRL> + <V> <ul style="list-style-type: none"> • Pastes the 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).

Key	Function
CTRL + ALT + C	<p><CTRL> + <ALT> + <C> Creates a complete standard archive (.ARC) on an external data carrier (USB-FlashDrive) (for 840D sl / 828D).</p> <p>Note: Please refer to the machine manufacturer's specifications.</p>
CTRL + ALT + S	<p><CTRL> + <ALT> + <S> Creates a complete standard archive (.ARC) on an external data carrier (USB-FlashDrive) (for 840D sl). Creates a complete Easy Archive (.ARD) on an external data carrier (USB-FlashDrive) (for 828D).</p> <p>Note: Please refer to the machine manufacturer's specifications.</p>
CTRL + ALT + D	<p><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.</p>
SHIFT + ALT + D	<p><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.</p>
SHIFT + ALT + T	<p><SHIFT> + <ALT> + <T> Starts "HMI Trace".</p>
SHIFT + CTRL + T	<p><SHIFT> + <ALT> + <T> Exits "HMI Trace".</p>
ALT + S	<p><ALT> + <S> Opens the editor to enter Asian characters.</p>
DEL	<p></p> <ul style="list-style-type: none"> Editing box Deletes the first character right of the cursor. Navigation Deletes all characters.
DEL + CTRL	<p> + <CTRL></p> <ul style="list-style-type: none"> Editing box Deletes the first word to the right of the cursor. Navigation Deletes all characters.

Key	Function
	<p><Spacebar></p> <ul style="list-style-type: none"> • Editing window Inserts a space • Switches between several specified options in selection drop-down list boxes and in selection boxes.
	<p><Plus></p> <ul style="list-style-type: none"> • Opens a directory which contains the element. • Increases the size of the graphic view for simulation and traces.
	<p><Minus></p> <ul style="list-style-type: none"> • Closes a directory which contains the element. • Reduces the size of the graphic view for simulation and traces.
	<p><Equals></p> <p>Opens the calculator in the entry fields.</p>
	<p><Asterisk></p> <p>Opens a directory with all of the subdirectories.</p>
	<p><Tilde></p> <p>Changes the sign of a number between plus and minus.</p>
	<p><INSERT></p> <ul style="list-style-type: none"> • 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.
 + 	<p><INSERT> + <SHIFT></p> <p>For G code programming, for a cycle call activates or deactivates the edit mode.</p>

Key	Function
	<p><INPUT></p> <ul style="list-style-type: none"> • 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.
	<p><ALARM> - only OP 010 and OP 010C</p> <p>Select the "Diagnosis" operating area.</p>
	<p><PROGRAM> - only OP 010 and OP 010C</p> <p>Calls the "Program Manager" operating area.</p>
	<p><OFFSET> - only OP 010 and OP 010C</p> <p>Calls the "Parameter" operating area.</p>
	<p><PROGRAM MANAGER> - only OP 010 and OP 010C</p> <p>Calls the "Program Manager" operating area.</p>
	<p>Menu forward key</p> <p>Advances in the extended horizontal softkey bar.</p>
	<p>Menu back key</p> <p>Returns to the higher-level menu.</p>
	<p><MACHINE></p> <p>Calls the "Machine" operating area.</p>
	<p><MENU SELECT></p> <p>Calls the main menu to select the operating area.</p>

1.3 Machine control panels

1.3.1 Overview

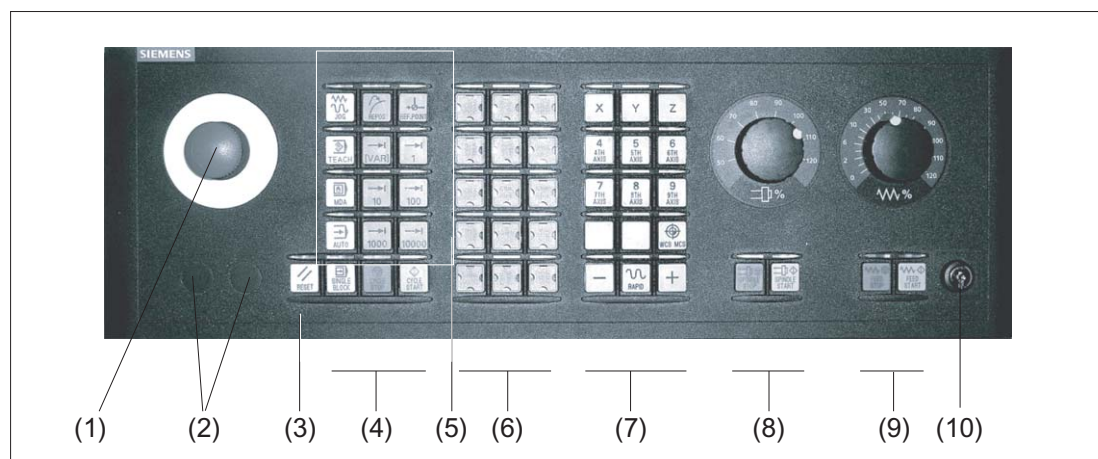
The machine tool can be equipped with a machine control panel by Siemens or with a specific machine control panel from the machine manufacturer.

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.

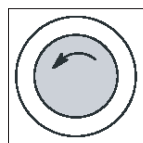
1.3.2 Controls on the machine control panel

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

Activate 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
(2)		For additional responses to pressing the Emergency Stop button, please refer to the machine manufacturer's instructions.
(3)		Installation locations for control devices (d = 16 mm) RESET
		<ul style="list-style-type: none"> • 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.
(4)		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.
(5)		Operating modes, machine functions
		<JOG> Select "JOG" mode.
		<TEACH IN> Select "Teach In" submode.
		<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.
	...	
		











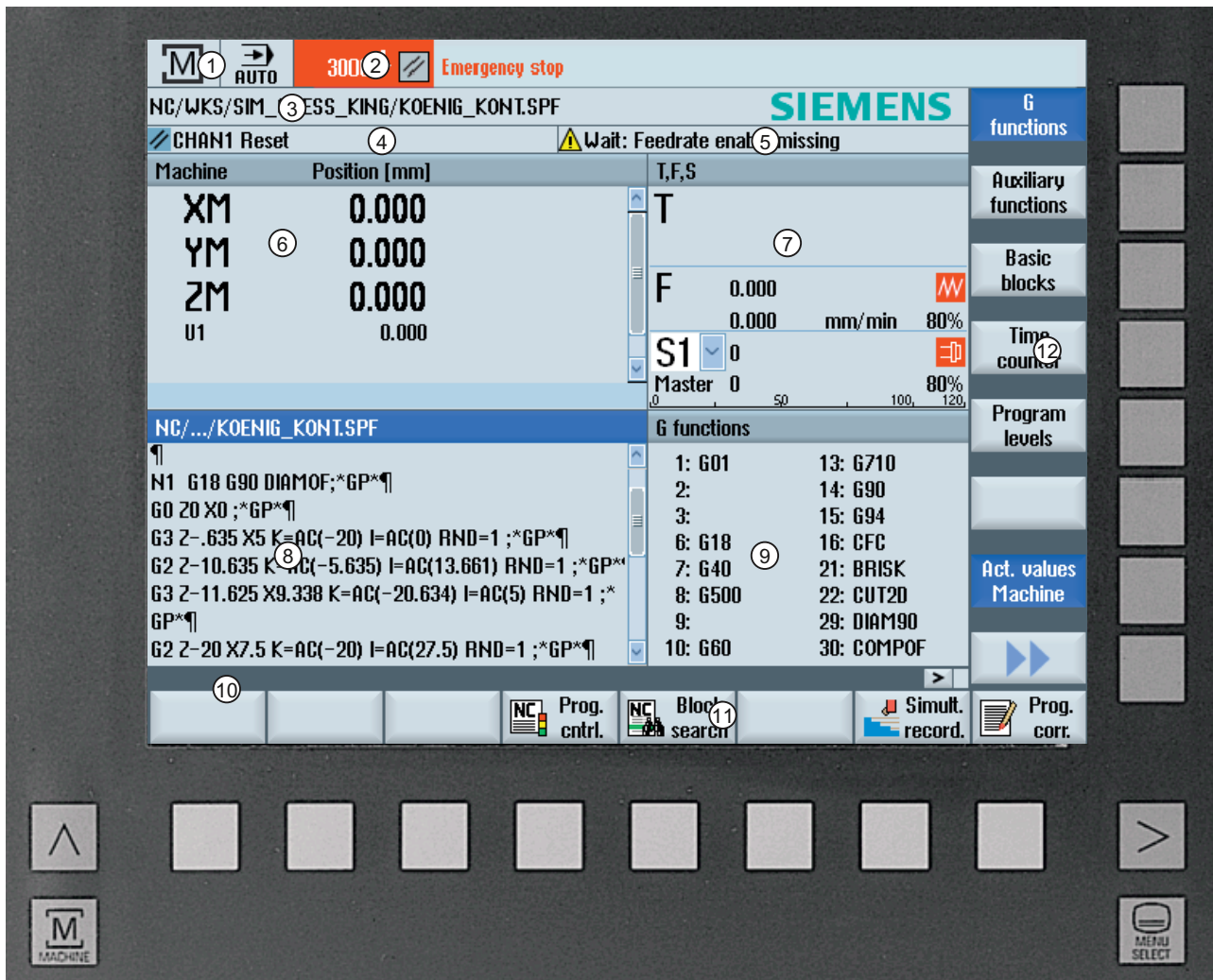
-  **Machine manufacturer**
A machine data code defines how the increment value is interpreted.
- (6) **Customer keys**
T1 to T15
- (7) **Traversal axes with rapid traverse superposition and coordinate exchange**
Axis keys
Selects an axis.
- 
- ...
- 
- Direction keys**
Select the traversing direction.
- 
- ...
- 
- <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).
- (8) **Spindle control with override switch**
<SPINDLE STOP>
Stop spindle.
- 
- <SPINDLE START>**
Spindle is enabled.
- 
- (9) **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.
- 
- (10) **Keyswitch (four positions)**

Figure 1-2 Front view of machine control panel (milling version)

1.4 User interface

1.4.1 Screen layout

Overview



- 1 Active operating area and mode
- 2 Alarm/message line
- 3 Program name
- 4 Channel state and program control
- 5 Channel operational messages
- 6 Axis position display in actual value window

- 7 Display for
 - active tool T
 - current feedrate F
 - active spindle with current status (S)
 - Spindle utilization rate in percent
- 8 Operating window with program block display
- 9 Display of active G functions, all G functions, H functions and input window for different functions (for example, skip blocks, program control)
- 10 Dialog line to provide additional user notes
- 11 Horizontal softkey bar
- 12 Vertical softkey bar

Figure 1-3 User interface

1.4.2 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.

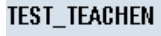
Status display of "Machine" operating area

First line

Display	Description
Active operating area	
	"Machine" operating area With touch operation, you can change the operating area here.
	"Parameter" operating area
	"Program" operating area

Display	Description
	"Program manager" operating area
	"Diagnosis" operating area
	"Start-up" operating area
Active mode or submode	
	"Jog" mode
	"MDA" mode
	"Auto" mode
	"Teach In" submode
	"Repos" submode
	"Ref Point" submode
Alarms and messages	
	<p>Alarm display</p> <p>The alarm numbers are displayed in white lettering on a red background. The associated alarm text is shown in red lettering.</p> <p>An arrow indicates that several alarms are active.</p> <p>An acknowledgment symbol indicates that the alarm can be acknowledged or canceled.</p>
	<p>NC or PLC message</p> <p>Message numbers and texts are shown in black lettering.</p> <p>An arrow indicates that several messages are active.</p>
	<p>Messages from NC programs do not have numbers and appear in green lettering.</p>

Second line

Display	Description
	Program path and program name








The displays in the second line can be configured.



Machine manufacturer

Please also refer to the machine manufacturer's instructions.

Third line

Display	Description
	<p>Display of channel status.</p> <p>If several channels are present on the machine, the channel name is also displayed.</p> <p>If only one channel is available, only the "Reset" channel status is displayed.</p> <p>With touch operation, you can change the channel here.</p>
  	<p>Display of channel status:</p> <p>The program was aborted with "Reset".</p> <p>The program is started.</p> <p>The program has been interrupted with "Stop".</p>
	<p>Display of active program controls:</p> <p>PRT: no axis motion</p> <p>DRY: Dry run feedrate</p> <p>RG0: reduced rapid traverse</p> <p>M01: programmed stop 1</p> <p>M101: programmed stop 2 (name varies)</p> <p>SB1: Single block, coarse (program stops only after blocks which perform a machine function)</p> <p>SB2: Data block (program stops after each block)</p> <p>SB3: Single block, fine (program also only stops after blocks which perform a machine function in cycles)</p>
 Faulty NC block / user alarm  Remaining dwell time:15 Sec.	<p>Channel operational messages:</p> <p>Stop: An operator action is usually required.</p> <p>Wait: No operator action is required.</p>

The machine manufacturer settings determine which program controls are displayed.



Machine manufacturer

Please also refer to the machine manufacturer's instructions.

1.4.3 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 ENS 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 ENS coordinate system, jumps into the actual value display are avoided, that would otherwise be caused by the additional components.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Maximize display



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

Overview of display

Display	Meaning
Header columns	
Work/Machine	Display of axes in selected coordinate system.
Item	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" submode.
Footer	Display of active work offsets and transformations. The T, F, S values are also displayed in the full-screen version.

See also

Zero offsets (Page 84)

1.4.4 T,F,S window


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

Tool data





Display	Meaning
T	
Tool name	Name of current tool
Location	Location number of 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.
\emptyset	Diameter of current tool
R	Radius of the actual tool

Display	Meaning
Z	Z value of the actual tool
X	X value of the actual tool

Feed data

Display	Meaning
F	
	Feed disable
	Actual feed value If several axes traverse, is displayed for: <ul style="list-style-type: none"> "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 <ul style="list-style-type: none">  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.

1.4.5 Current block display

The window of the current block display shows you 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 title row.
- The program block which is just being processed appears colored.

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.

1.4.6 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) and five operating modes or submodes (JOG, MDA, AUTO, TEACH IN, REF POINT, REPOS).

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 submode directly using the keys on the machine control panel or using 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.


1.4.7 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. You can execute the four basic calculation types, work with expressions in brackets as well as generate square roots and squares.

Note

Generating square roots and squares

The extract roots and generate square functions is 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 key to delete individual characters.



+ <*>

Enter the multiplication characters using the <SHIFT> + <*> keys.



+ </>

Enter the division character using the <SHIFT> + </> keys.



Enter bracket expressions using the <SHIFT> + <()> and <SHIFT> + <(> <)> keys.



+ <(>



+ <number> Enter "r" or "R" as well as the number x from which you would like to extract the root.



+ <number> Enter "s" or "S" as well as the number x for which you would like to generate the square.



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.

1.4.8

Pocket calculator

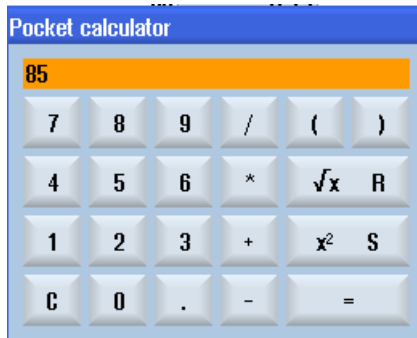
You can use the calculator to quickly calculate parameter values during programming. If, for example, the diameter of a workpiece is only dimensioned indirectly in the workpiece drawing, i.e., the diameter must be derived from the sum of several other dimension specifications, you can calculate the diameter directly in the input field of this parameter.

Calculation methods


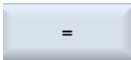



The following arithmetic operations are available:

- Addition
- Subtraction
- Multiplication
- Division
- Calculation with parentheses
- Square root of x
- x squared

You can input a maximum of 256 characters in a field.



Procedure

1. Position the cursor on the desired entry field.
2. Press the <=> key.

- The calculator is displayed.
3. Input the arithmetic statement.
You can use arithmetic symbols, numbers, and commas.
4. 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.
5. Press the "Accept" softkey.

The calculated value is accepted and displayed in the entry field of the window.

Note

Input order for functions

When using the square root or squaring functions, make sure to press the "R" or "S" function keys, respectively, before entering a number.

1.4.9 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 10 changes can be undone.

1.4.10 Touch operation

If you have an operator panel with a touch screen, you can perform the following functions with touch operation:

Operating area switchover



You can display the operating area menu by touching the display symbol for the active operating area in the status display.



Channel switchover



You can switch over to the next channel by touching the channel display in the status display.

1.4.11 Changing the user interface language

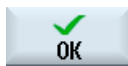
Procedure



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>.

1.4.12 Entering Asian characters

You have the possibility of entering Asian characters.

Note

Call the input editor with <Alt + S>

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

You can select a character by using the Pinyin phonetic notation, which enables Chinese characters to be expressed by combining Latin letters.

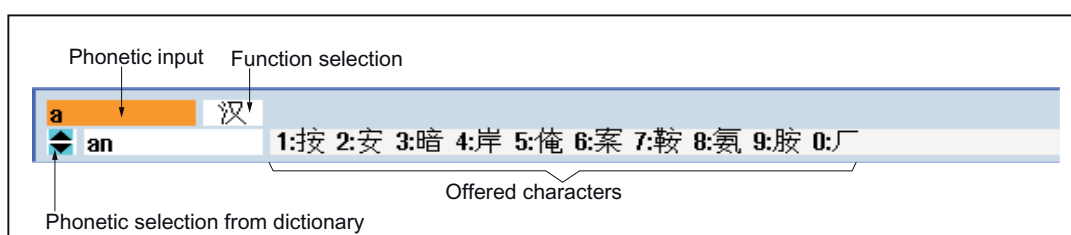
The editor is available for the following Asian languages:

- Simplified Chinese
- Traditional Chinese
- Korean




Note

You require a special keyboard to enter Korean characters.

Structure of editor



Functions

-  Pinyin input
-  Editing of the dictionary
-  Input of Latin letters

Precondition

The control has been set to Chinese or Korean.

Procedure

Editing characters



+



1. Open the screen form and position the cursor on the entry field and press the <Alt +S> keys.
The editor is displayed.
2. Enter the desired phonetic notation.
3. Click the <Cursor down> key to access the dictionary.
4. By 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 the dictionary



1. Select the dictionary editing function in the selection box.
The editor provides a further line in which the combined characters and phonetic notations are displayed.
2. Enter the desired phonetic notation in the phonetic input field.
Various characters are displayed for this phonetic notation, from which you can select a character by entering either of the appropriate number (1 to 9).
You can toggle the input cursor between the compound phonetic notations field and the phonetic input field by pressing the <TAB> key.
Combined characters are cancelled via the <BACKSPACE> key.
3. Press the <SELECT> key to take a combined phonetic notation into the dictionary.

1.4.13 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
- Zero 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.


References







For additional information, please refer to the following documentation:






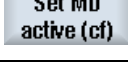

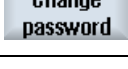
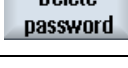
Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

Softkeys

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

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

Diagnostics operating area	Protection level
	Keyswitch 3 (protection level 4)
	User (protection level 3)
	User (protection level 3)
	Manufacturer (protection level 1)
	User (protection level 3)
	Service (protection level 2)

Start-up operating area	Protection levels
 System data	End user (protection level 3)
 Setup archive	Keyswitch 3 (protection level 4)
 General MD	Keyswitch 3 (protection level 4)
 Control unit MD	
 Licenses	Keyswitch 3 (protection level 4)
 Set MD active (cf)	Keyswitch 3 (protection level 4)
 NCK	Service (protection level 2)
 Change password	End user (protection level 3)
 Delete password	End user (protection level 3)

1.4.14 Online help in SINUMERIK Operate

A comprehensive context-sensitive online 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 online 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.



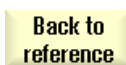
3. Press the "Full screen" softkey to use the entire user interface for the display of the online help.



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



4. If further helps are 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 you are using, the Operating Manuals "Operator control Milling", "Operator control Turning" or "Operator control Universal" as well as the "Programming" Programming Manual are displayed.



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



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



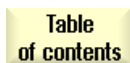
1. Press the "Search" softkey.
The "Search in Help for:" window is opened.

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 if you only want to display the index of the operating and programming manual.

Displaying alarm descriptions and machine data



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

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> or the <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.



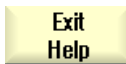
2. Press the "Display all G functions" softkey.



3. With the aid of 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.

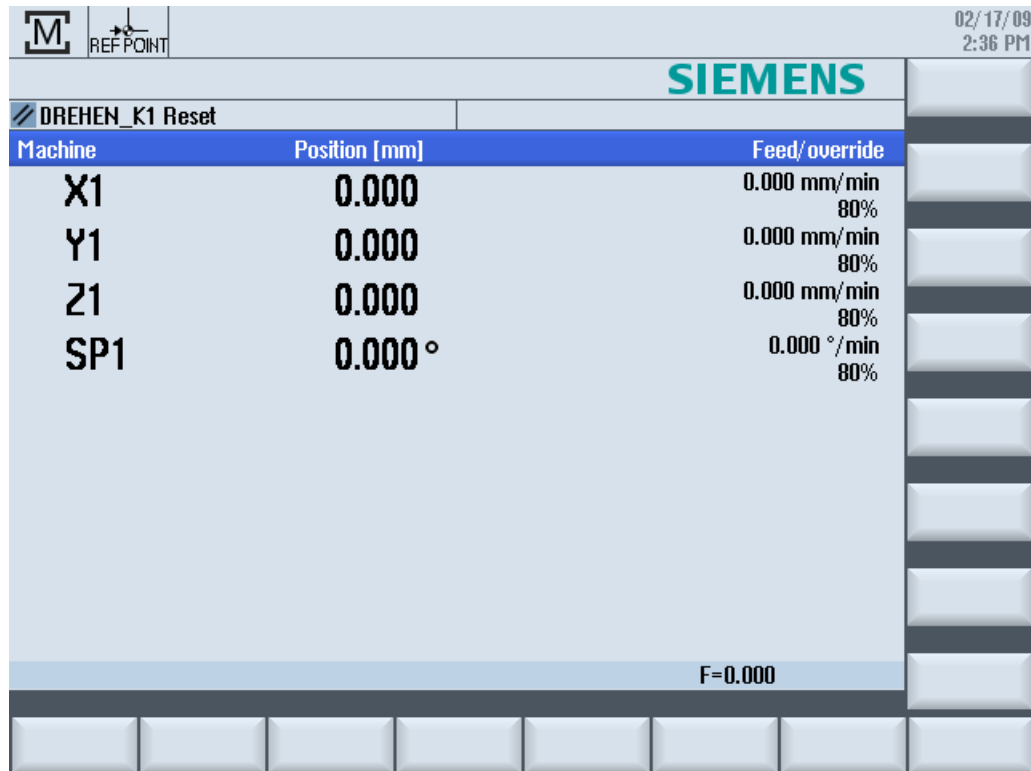


5. Press the "Exit help" softkey again to close the help.

Setting up the machine

2.1 Switching on and switching off

Start-up



When the control starts up, the main screen opens according to the operating mode specified by the machine manufacturer. In general, this is the main screen for the "REF POINT" submode.



Machine manufacturer

Please also refer to the machine manufacturer's instructions.

2.2 Approaching a reference point

2.2.1 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

If the axes are not in a collision-free position, you must first traverse them to safe positions in "JOG" or "MDA" 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".

2.2.2 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.

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



6. 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.



2.3 Modes and mode groups

2.3.1 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
- Traversing axes, e.g. during a program interruption
- Positioning axes

Select "JOG"



Press the <JOG> key.

"Ref Point" submode

The "REF POINT" submode 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" submode

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

The distances 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).

Selecting "Repos"



Press the <REPOS> key.

"MDA" mode (Manual Data Automatic)

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

Selecting "MDA"



Press the <MDA> key.

"AUTO" mode

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

Select "AUTO"



Press the <AUTO> key.

"TEACH IN" submode

The "TEACH IN" submode is available in the "AUTO" and "MDA" modes.

There you may create, edit and execute part programs (main programs or subroutines) for motional sequences or simple workpieces by approaching and saving positions.

Selecting "Teach In"



Press the <TEACH IN> key.

2.3.2 Modes groups and channels

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

- Control with 1 channel
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.

2.3.3 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.

References

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

Channel switchover via touch operation

On the HT 8 and when using a touch screen operator panel, you can switch to the next channel or display the channel menu via touch operation in the status display.

2.4 Settings for the machine

2.4.1 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.

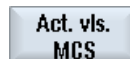
Procedure



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.

2.4.2 Switching the unit of measurement

You can set millimeters or inches as the unit of measurement. 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



Machine manufacturer

Please also refer to the machine manufacturer's instructions.

Proceed as follows



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



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



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



4. Press the "OK" softkey.

The softkey label changes to "Switch to metric".

The unit of measurement applies to the entire machine.



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

See also

Default settings for manual mode (Page 121)

2.4.3 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.

Requirement

The controller is in the workpiece coordinate system.

The actual value can be set in both the Reset and Stop 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 -



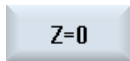
Press the ">>", "REL act. vals" and "Set REL" softkeys to set position values in the relative coordinate system.



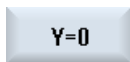
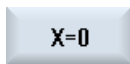
3. Enter the new required position value for Z, X or Y directly in the actual value display (you can toggle between the axes with the cursor keys) and press the <INPUT> key to confirm the entries.



- OR -



Press softkey "Z=0", "X=0" or "Y=0" (if there is a Y axis), to set the required position to zero.



Resetting the actual value



Press the "Delete active ZO" softkey.
The offset is deleted permanently.

NOTICE

Irreversible active zero offset

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

2.5 Measuring the tool

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.

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

See also

Tool dimensioning (Page 643)

Measuring the tool (Page 655)

2.5.1 Measuring a tool manually

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

Lathes with B axis

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

Procedure



1. Select "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 is opened.



5. Select the tool that you wish to measure.
The cutting edge position and the radius or diameter of the tool must already be entered in the tool list.



6. Press the "In manual" softkey.
The tool is accepted into the "Length Manual" window.



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.

2.5.2 Measuring a tool with a tool probe

During automatic measuring, you determine the tool dimensions in the directions X and Z with the aid of a probe.

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

The function "Measure with tool carrier that can be orientated" is implemented for lathes with a swivel axis around Y and associated tool spindle. The swivel axis can be used to align the tool on 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

Lathes with B axis

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

References

For further information about lathes with B axis, please refer to the following reference:

Commissioning Manual SINUMERIK Operate / SINUMERIK 840D sl

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

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)

Preconditions

- 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 the tool should be aligned in the same way that it will be subsequently measured.



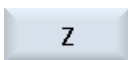
2. Select the "JOG" mode in the "Machine" operating area.



3. Press the "Meas. tool" and "Automatic" softkeys.



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



4. 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.
5. Press the <CYCLE START> key.



The automatic measuring process is started, i.e. the tool is traversed at 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.

If turning tools with tool carrier that can be oriented are measured 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.

2.5.3 Calibrating the tool probe

To be able to measure your tools automatically, you must first determine the position of the tool probe in the machine area in relation to the machine zero.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Sequence

The calibrating tool must be a turning tool type (roughing or finishing tool). Cutting edge positions 1 - 4 can be used for the tool probe calibration. You must enter the length and the radius or diameter of the calibrating tool in the tool list.

Calibrate the probe in all directions in which you wish to subsequently perform measurements.

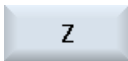
Procedure



1. Change the calibrating tool.
2. Select the "JOG" mode in the "Machine" operating area.



3. Press the "Meas. tool" and "Calibrate probe" softkeys.



4. Press the "X" or "Z" softkey, depending on which point of the tool probe you wish to determine first.



5. Select the direction (+ or -), in which you would like to approach the tool probe.

6. Position the calibrating tool in the vicinity of the tool probe in such a way that any collisions can be avoided when the first point of the tool probe is being approached.



7. Press the <CYCLE START> key.
The calibration process is started, i.e. the calibrating tool is automatically traversed at the measurement feedrate to the probe and back again. The position of the tool probe is determined and saved in an internal data area.
8. Repeat the process for the other other points of the tool probe.

2.5.4 Measuring a tool with a magnifying glass

You can also use a magnifying glass to determine the tool dimensions, if this is available on the machine.

In this case, SINUMERIK Operate calculates the tool offset data from the known positions of the tool carrier reference point and the cross-hairs of the magnifying glass.

Note

Lathes with B axis

For lathes 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 "Zoom" softkey.



4. Press the "Select tool" softkey.
The "Tool selection" window is opened.

5. Select the tool that you wish to measure.
The cutting edge position and the radius or diameter of the tool must already be entered in the tool list.



6. Press the "In manual" softkey.
The tool is accepted in the "Zoom" window.

7. Traverse the tool towards the magnifying glass and align the tool tip P with the magnifying glass cross-hairs.



- Press the "Set length" softkey.

2.6 Measuring the workpiece zero

The reference point for programming a workpiece is always the workpiece zero. To determine this zero point, measure the length of the workpiece and save the position of the cylinder's face surface in the direction Z in a zero offset. This means that the position is stored in the coarse offset and existing values in the fine offset are deleted.

Calculation

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

Measuring only

If you wish to measure the workpiece zero in "Measuring Only" mode, the measured values are merely displayed without any changes being made to the coordinate system.

Adapting the user interface to the measurement functions

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

- Calibration plane, measuring plane (only 840D sl)
- Zero offset as basis for the measuring process (only 840D sl)
- Number of the probe calibration data record (only 840D sl)
- Offset target, settable zero offset
- Offset target, basis reference
- Offset target, global basic zero offset (only 840D sl)
- Offset target, channel-specific basic zero offset (only 840D sl)



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Requirement

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

Procedure



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



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



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

- OR -



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

- OR -



Press the "Zero offset" softkey and select the zero offset in which the zero point is to be saved in the "Zero Offset – G54 ... G599" window that opens and press the "In Manual" softkey.



You return to the "Set Edge" window.



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

Note

Settable zero offsets

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

Please refer to the machine manufacturer's specifications.

2.7 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.

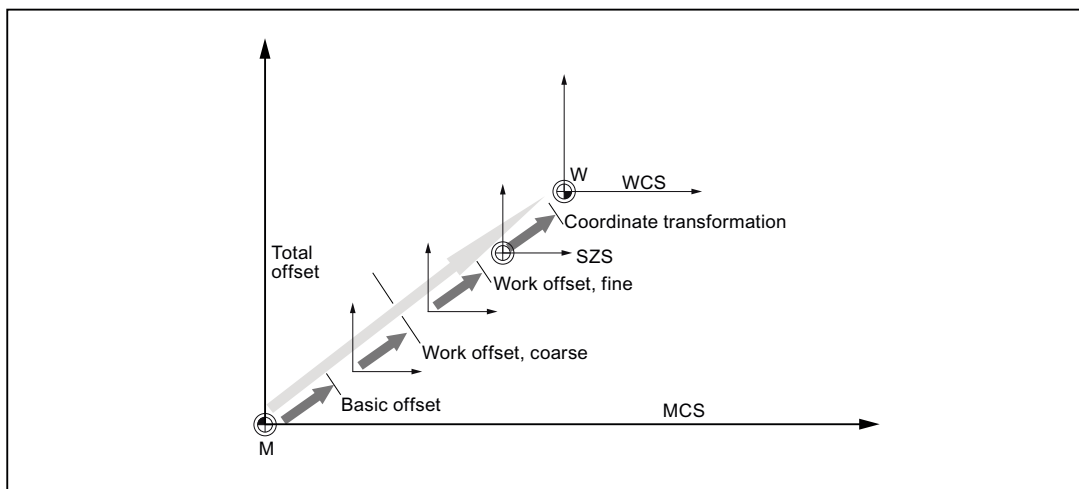


Figure 2-1 Zero offsets

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

Base offset

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

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 via machine data
\$MN_MM_FRAM_FINE_TRANS

See also

Actual value window (Page 44)

2.7.1 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.

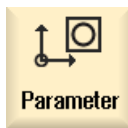
The availability of the offsets depends on the setting.



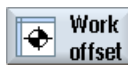
Machine manufacturer

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.

2.7.2 Displaying the zero offset "overview"

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

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

This window is generally used only for monitoring.

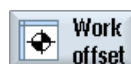
Display of active zero offsets

Zero offsets	
DRF	Displays the handwheel axis offset.
Rotary table reference	Displays the additional zero offsets programmed with \$ P_PARTFRAME.
Basic reference	Displays the additional zero offsets programmed with \$P_SETFRAME. Access to the system offsets is protected via a keyswitch.
External ZO frame	Displays the additional zero offsets programmed with \$P_EXTFRAME.
Total base ZO	Displays all effective basis offsets.
G500	Displays the zero offsets activated with G54 - G599. Under certain circumstances, you can change the data using "Set ZO", i.e. you can correct a zero point that has been set.
Tool reference	Displays the additional zero offsets programmed with \$P_TOOLFRAME.
Workpiece reference	Displays the additional zero offsets programmed with \$P_WPFRAME.
Programmed ZO	Displays the additional zero offsets programmed with \$P_PFRAME.
Cycle reference	Displays the additional zero offsets programmed with \$P_CYCFRAME.
Total ZO	Displays the active zero offset, resulting from the total of all zero offsets.

Procedure



1. Select the "Parameter" operating area.



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



2.7.3 Displaying and editing base zero offset

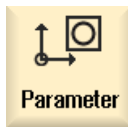
The defined channel-specific and global base offsets, divided into coarse and fine offsets, are displayed for all set-up axes in the "Zero offset - Base" window.



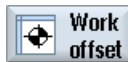
Machine manufacturer

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

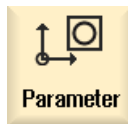
The offsets specified here are immediately active.

2.7.4 Displaying and editing settable zero offset

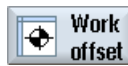
All settable offsets, divided into coarse and fine offsets, are displayed in the "Zero Offset - G54..G599" window.

Rotation, scaling and mirroring are displayed.

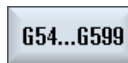
Procedure



1. Select the "Parameter" operating area.



2. Press the "Zero offset" softkey.



3. Press the "G54...G599" softkey.
The "Zero Offset - G54..G599" window is opened.

Note

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 observe the machine manufacturer's specifications.

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.

2.7.5 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



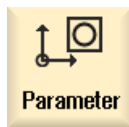
Machine manufacturer

Please refer to the machine manufacturer's specifications.

Note

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

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 appears.



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.

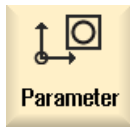


Press the "Back" softkey to close the window.

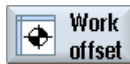
2.7.6 Deleting a zero offset

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

Procedure



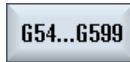
1. Select the "Parameter" operating area.



2. Press the "Zero offset" softkey.



3. Press the "Active", "Base" or "G54...G599" softkey.



4. Press the "Details" softkey.



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

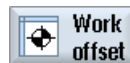
6. Press the "Clear offset" softkey.

2.7.7 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 "Workpiece zero" softkey.



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



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

2.8 Monitoring axis and spindle data

2.8.1 Specify working area limitations

The "Working area limitation" function can be used to limit the range within which a tool can traverse in all channel axes. These commands allow you to set up protection zones in the working area which are out of bounds for tool movements.

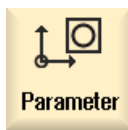
In this way, you are able to restrict the traversing range of the axes in addition to the limit switches.

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.

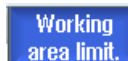
Procedure



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.

2.8.2 Editing spindle data

The speed limits set for the spindles that must not be under- or overshoot 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

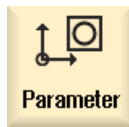
In field "Spindle speed limitation at G96", the programmed spindle speed limitation at constant cutting speed is displayed together with the permanently active limitations.

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.

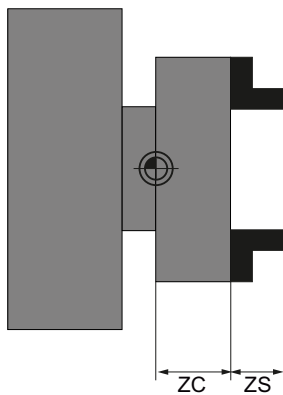
2.8.3 Spindle chuck data

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

Manually measuring a tool

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

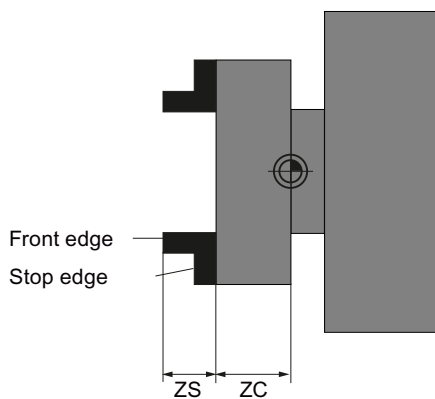
Main spindle



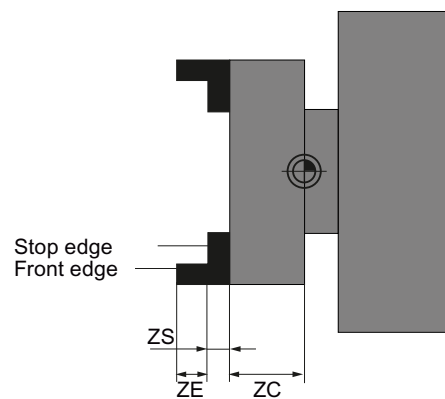
Dimensioning the main spindle

Counterspindle

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

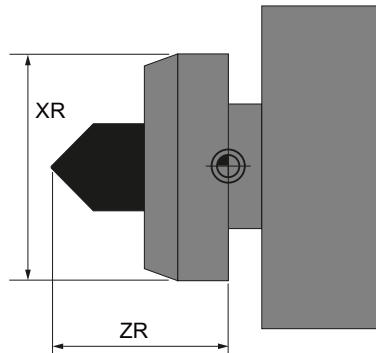


Dimensions, counterspindle jaw type 1



Dimensions, counterspindle jaw type 2

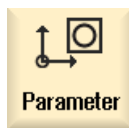
Tailstock



Dimensioning the tailstock

The length of the tailstock (ZR) and the diameter of the tailstock (XR) of the spindle screen are needed for the display of the tailstock in the simulation.

Procedure



1. Select the "Parameter" operating area.



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



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

See also

Machining with the counterspindle (Page 521)

Parameter	Description	Unit
ZC1	Main spindle chuck dimensions (inc)	mm
Jaw type	Dimensions of the forward edge or stop edge <ul style="list-style-type: none">Jaw type 1Jaw type 2	
ZC3	Chuck dimension, counter-spindle (inc) - only for a counter-spindle that has been set-up	mm
ZS3	Stop dimension, counter-spindle (inc) - only for a counter-spindle that has been set-up	mm
ZE3	Jaw dimension, counter-spindle (inc) - only for a counter-spindle that has been set-up and "Jaw type 1"	mm
XR	Tailstock diameter - only for tailstock that has been set-up	mm
ZR	Tailstock length - only for tailstock that has been set-up	mm

2.9 Displaying setting data lists

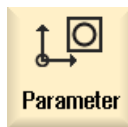
You can display lists with configured setting data.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure



1. Select the "Parameter" operating area.



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



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

2.10 Handwheel assignment

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



Software option

You require the "Extended operator functions" option for the handwheel offset (only for 828D).

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

- Geometry axes

When traversing, the geometry axes taken into account the actual 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 individually traversed, i.e. the actual machine state has no influence.

The also applies to channel machine axes, that are declared as geometry axes.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure



1. Select the "Machine" operating area.



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

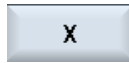


2. Press the menu forward key and the "Handwheel" softkey.
The "Handwheel" window appears.



A field for axis assignment will be offered for every connected handwheel.

3. Position the cursor in the field next to the handwheel with which you wish to assign the axis (e.g. No. 1).



4. 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.

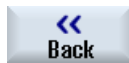


Selecting an axis also activates the handwheel (e.g., "X" is assigned to handwheel no. 1 and is activated immediately).



5. 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).

2.11 MDA

In "MDA" mode (Manual Data Automatic mode), you can enter G-code commands block-by-block and immediately execute them for setting up the machine.

You can load an MDA program straight from the Program Manager into the MDA buffer. You may also store programs which were rendered or changed in the MDA operating window into any directory of the Program Manager.



Software option

You require the "Extended operator functions" option to load and save MDA programs (for 828D).

2.11.1 Loading an MDA program from the Program Manager

Procedure



1. Select the "Machine" operating area.



2. Press the <MDI> key.

The MDI editor opens.



3. Press the "Load MDI" softkey.

A changeover is made into the Program Manager.

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.

2.11.2 Saving an MDA program

Procedure



1. Select the "Machine" operating area.



2. Press the <MDI> key.

The MDI editor opens.

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



4. Press the "Store MDI" softkey.

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

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

- OR -



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

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



6. Press the "OK" softkey.

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

- OR -

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



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

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

2.11.3 Executing an MDA program

Proceed as follows



1. Select the "Machine" operating area.



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



3. Input the desired G-code commands using the operator's keyboard.
4. Press the <CYCLE START> key.

The control executes the input blocks.

When executing the G-code commands, you can control 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

2.11.4 Deleting an MDA program

Precondition

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

Procedure



Press the "Delete blocks" softkey.

The program blocks displayed in the program window are deleted.

Working in manual mode

3.1 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.

3.2 Selecting a tool and spindle

3.2.1 T,S,M window

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

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

Your turning machine can also be equipped with a counterspindle (S3).

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





Note

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



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Display	Meaning
T	Input of the tool (name or location number) You can select a tool from the tool list using the "Select tool" softkey.
D	Cutting edge number of the tool (1 - 9)
Spindle	Spindle selection, identification with spindle number
Spindle M function	 Spindle off: Spindle is stopped
	 CCW rotation: Spindle rotates counterclockwise
	 CW rotation: Spindle rotates clockwise
	 Spindle positioning: Spindle is moved to the desired position.
Other M functions	Input of machine functions Refer to the machine manufacturer's table for the correlation between the meaning and number of the function.
G work offset	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.
Measurement unit	Selection of the unit of measurement (inch, mm). The setting made here has an effect on the programming.
Machining plane	Selection of the machining plane (G17(XY), G18 (ZX), G19 (YZ))
Gear stage	Specification of the gear stage (auto, I - V)
Stop position	Input of the spindle position in degrees

Note

Spindle positioning

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

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

3.2.2 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 entry field.
- OR -



Press the "Tool" softkey to open the tool list, position the cursor on the desired tool and press the "To Manual" softkey.



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



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



5. Press the <CYCLE START> key.

The tool is automatically swung into the machining position and the name of the tool displayed in the tool status bar.

3.2.3 Starting and stopping the spindle manually

Procedure



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



2. Select the desired spindle (e.g. S1) and enter the desired spindle speed or cutting speed in the right-hand entry field.



3. If the machine has a gearbox for the spindle, set the gearing step.
4. Select a spindle direction of rotation (clockwise or counterclockwise) in the "Spindle M function" field.



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



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



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

Note

Changing the spindle speed

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

3.2.4 Positioning the spindle

Procedure



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



2. Select the "Stop Pos." setting in the "Spindle M function" field.
The "Stop Pos." entry field appears.

3. Enter the desired spindle stop position.
The spindle position is specified in degrees.



4. Press the <CYCLE START> key.

The spindle is moved to the desired position.

Note

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

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

3.3 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.

3.3.1 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.

3.3.2 Traversing axes by a variable increment

Procedure



1. Select the "Machine" operating area.



Press the <JOG> key.



2. Press the "Settings" softkey.
The "Settings for Manual Operation" window is opened.

3. Enter the desired value for the "Variable increment" parameter.
Example: Enter 500 for a desired increment of 500 μm (0.5 mm).



4. Press the <Inc VAR> key.



5. Select the axis to be traversed.



6. 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.

3.4 Positioning axes

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

The feedrate / rapid traverse override is active during traversing.

Procedure



1. If required, select a tool.
2. Select the "JOG" operating mode.



3. Press the "Positions" softkey.
4. Enter the target position or target angle for the axis or axes to be traversed.
5. Specify the desired value for the feedrate F.
- OR -



Press the "Rapid traverse" softkey.
The rapid traverse is displayed in field "F".



6. Press the <CYCLE START> key.
The axis is traversed to the specified target position.

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

3.5 Manual retraction

After the interruption in machining due to power loss or a RESET at the machine control panel, you have the possibility to retract the tool in the JOG mode in the tool direction without damaging the tool or the workpiece.

The retraction function 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.

Both the Z axis as well as the spindle can be used for retraction for the thread.

The "Retraction" function must be set up by the machine manufacturer.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure

1. <RESET> interrupts the power supply to the machine and any running 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.

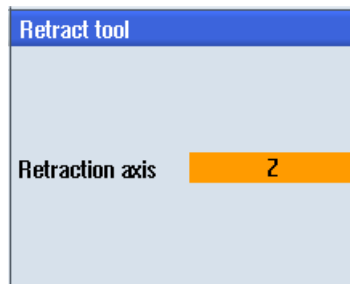
The softkey is available only when an active tool and retraction data are present.

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





7. Select the required axis in the "Retraction axis" selection box.



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



9. To exit the window, press the "Retract" softkey again when the tool is at the desired position.

3.6 Simple stock removal of workpiece

Some blanks 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, you can 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 via 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 "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

You cannot use the "Repos" function 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 "Stock removal" softkey.



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.

Table 3- 1

Parameter	Description	Unit
T	Tool name	
D	Cutting edge number	
F	Feedrate	mm/rev
S / V 	Spindle speed or constant cutting rate	rpm m/min
Spindle M function 	Direction of spindle rotation (only when ShopTurn is not active) <ul style="list-style-type: none"> •  •  	
Machining 	<ul style="list-style-type: none"> • ▽ (roughing) • ▽▽ (finishing) 	

3.6 Simple stock removal of workpiece

Parameter	Description	Unit
Position 	Machining position 	
Machining direction 	<ul style="list-style-type: none"> • Face • Longitudinal 	
X0	Reference point \emptyset (abs)	mm
Z0	Reference point (abs)	mm
X1 	End point X \emptyset (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

See also

Tool, offset value, feedrate and spindle speed (T, D, F, S, V) (Page 240)

3.7 Thread synchronizing

If you wish to re-machine a thread, it may be necessary to synchronize the spindle to the existing thread turn. This is necessary as by re-clamping the blank, an angular offset can occur in the thread.

Constraint

Thread synchronizing is not possible if a toolcarrier is used (B axis).

Note

Activating/deactivating thread synchronization

If a thread synchronization is active, then this is active for all of the following "Thread turning" machining steps.

Thread synchronization remains effective without deactivation even after the machining has been shutdown.

Requirement

The spindle is stationary.

One threading tool is active.

Procedure



1. Select the "JOG" operating mode.



2. Press the menu forward key and the "Thread synchr." softkey.



3. Thread the thread cutting tool into the thread turn as shown in the help screen.



4. Press the "Teach-in main spindle" softkey if you are working at the main spindle.

- OR -



Press the "Teach-in counterspindle" softkey if you are working at the counterspindle.

Note:

The thread synchronization is activated by teaching in a spindle. In this case, the synchronizing positions of axes X and Z and the synchronizing angle of spindle (Sn) are saved in the Machine and displayed in the screen form.

The selection boxes for main spindle and counterspindle indicate whether thread synchronization is active for the particular spindle (yes = active / no = not active).



5. Now carry out the "thread turning" machining step.
6. For the main spindle or counterspindle, select the "no" entry to deactivate thread synchronization.

See also

Thread turning (CYCLE99) (Page 336)

3.8 Default settings for manual mode

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

Presettings

Settings	Description
Type of feedrate	Here, you select the type of feedrate.
	<ul style="list-style-type: none">• G94: Axis feedrate/linear feedrate• G95: Rev. feedrate
Default feedrate G94	Enter the desired feedrate in mm/min.
Default feedrate G95	Enter the desired feedrate in mm/r.
Variable increment	Enter the desired increment for axis traversal by variable increments.
Spindle speed	Enter the desired spindle speed in rpm.

Proceed as follows



1. Select the "Machine" operating area.



2. Press the <JOG> key.



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



See also

Switching the unit of measurement (Page 72)

Machining the workpiece

4.1 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.

Requirements

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 zero 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.



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



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 controller is in "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 program blocks are not executed to the end. On the next start, machining is resumed from the point where it left off.

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 refer to the machine manufacturer's specifications.

4.2 Selecting a program

Procedure



1. Select the "Program manager" operating area.
The directory overview is opened.



2. Place the cursor on the directory containing the program that you want to select.
3. Press the <INPUT> key

- OR -



Press the <Cursor right> key.

The directory contents are displayed.



4. Place the cursor on the desired program.
5. Press the "Select" softkey.

The program is selected.

When the program has been successfully selected, an automatic changeover to the "Machine" operating area occurs.

4.3 Executing a trail program run

When testing a program, 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	The machining stops after every machine block (except for cycles)
SB2 Data block	The machining stops after every block, i.e. also for data blocks (except for cycles)
SB3 Single block, fine	The machining stops after every machine block (also in cycles)

Precondition

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

Procedure



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



2. Press the <SINGLE BLOCK> key.



3. Press the <CYCLE START> key.
Depending on the execution variant, the first block will be executed.
Then the machining stops.

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



4. Press the <CYCLE START> key.
Depending on the mode, the program will continue executing until the next stop.



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



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

See also

Setting for automatic mode (Page 174)

4.4 Displaying the current program block

4.4.1 Current block display

The window of the current block display shows you 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 title row.
- The program block which is just being processed appears colored.

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.

4.4.2 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 can 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 refer to the machine manufacturer's specifications.

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.

4.4.3 Display program level

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

Several program run throughs

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

Program example

N10 subprogram P25

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

Display of program level

The following information will be displayed:

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

Precondition

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

Procedure



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

4.5 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 possibilities

Depending on the state of the control system, you can make the following corrections using the Program editing function.

- Stop mode
Only program lines that have not yet been executed can be edited.
- Reset mode
All program lines can be edited.

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.

Requirement

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

Exit the editor using the "Close" softkey to return to the "Program manager" operating area.

4.6 Repositioning axes

After a program interruption in 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

Using the "Repos" function, you can return the tool to the contour in order to continue executing the program.

You cannot traverse the interrupt position, because it is blocked by the control system.

The feedrate/rapid traverse override is in effect

WARNING

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 and subsequently move the axes in manual mode after a program interrupt, the control automatically moves the axes during the switch to automatic mode and the subsequent start of the machining process in a straight line back to the 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.

Proceed as follows



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.



4.7 Starting machining at a specific point

4.7.1 Use block search

If you would only like to perform a certain section of a program on the machine, then you need not start the program from the beginning. You can also 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)
 - 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 or RESET), 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 it was previously possible to select an interruption point in a subprogram. It is then possible to change the program level up to the main program level and back to the level of the interruption point.
- Search pointer
 - Direct entry of the program path

Note

Searching for point in subprogram

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



Software option

You require the "Extended operator functions" option for the "Search pointer" function (only for 828D).

Cascaded search

You can start another search from the "Search target found" state. The cascading can be continued any number of times after every search target found.

Note

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

References

Function Manual Basic Functions; Block Search

Requirements

1. You have selected the desired program.
2. The controller is in the reset state.
3. The desired search mode is selected.

NOTICE
Collision-free start position
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.

See also

Selecting a program (Page 125)

4.7.2 Continuing program from search target

To continue the program at the desired position, press the <CYCLE START> key twice.

- 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.

By changing to the JOG REPOS mode, you can also manually traverse the tool from the current position to the setpoint position, if the setpoint position is not to be automatically approached after the program start

4.7.3 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.

4.7.4 Defining an interruption point as search target

Precondition

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



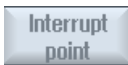
Software option

You require the "Extended operator functions" option (only for 828D).

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.

4.7.5 Entering the search target via search pointer

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



Software option

You require the "Extended operator functions" option for the "Search pointer" function (only for 828D).

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 specified target must always be unambiguous. 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.

4.7 Starting machining at a specific point

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.

4.7.6 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:	Pass counter If a program section 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

4.7.7 Block search mode

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

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

Search variants

Block search mode	Meaning
With calculation - without approach	<p>It is used in order to be able to approach a target position in any circumstance (e.g. tool change position).</p> <p>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.</p> <p>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.</p>
With calculation - with approach	<p>It is used to be able to approach the contour in any circumstance.</p> <p>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.</p>
With calculation - skip extcall	<p>This is used to speed-up a search with calculation when using EXTCALL programs: EXTCALL programs are not taken into account.</p> <p>Notice: Important information, e.g. modal functions, which are located in the EXTCALL program, are not taken into account. In this case, after the search target has been found, the program is not able to be executed. Information such as this should be programmed in the main program.</p>
Without calculation	<p>For a quick search in the main program.</p> <p>Calculations will not be performed during the block search, i.e. the calculation is skipped up to the target block.</p> <p>All settings required for execution have to be programmed from the target block (e.g. feedrate, spindle speed, etc.).</p>
With program test	<p>Multi-channel block search with calculation (SERUPRO).</p> <p>All blocks are calculated during the block search. Absolutely no axis motion is executed, however, all auxiliary functions are output.</p> <p>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.</p> <p>For single-channel systems, the coordination is supported with events running in parallel, e.g. synchronized actions.</p> <p>Note The search speed depends on MD settings.</p>

Note**Search mode for the ShopMill and ShopTurn programs**

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



Machine manufacturer

Please refer to the machine manufacturer's specifications.

References

For additional information, please refer to the following documentation:

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

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 appears.



4.8 Controlling the program run

4.8.1 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: Program processing without axis motion can also be activated with the function "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. Caution: Workpieces must not be machined 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 refer to the machine manufacturer's specifications.
DRF Handwheel offset	Enables an additional incremental zero offset while processing in automatic mode with an electronic handwheel. This function can be used to compensate for tool wear within a programmed block. Note: You require the "Extended operator functions" option to use the handwheel offset (for 828D).
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 only stops after blocks which perform a machine function in cycles. Select the desired setting using the <SELECT> key.
SKP	Skip blocks are skipped during machining.

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

Procedure



1. Select the "Machine" operating area.



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



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

See also

Setting for automatic mode (Page 174)

4.8.2 Skip blocks

It is possible to skip program blocks, which 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. Several consecutive blocks can also be skipped.

The statements in the skipped blocks are not executed, i.e. 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 refer to the machine manufacturer's specifications.



Software option

In order to have more than two skip levels, for 828D you require the "Extended operator functions" option.

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.

Procedure



1. Select the "Machine" operating area.

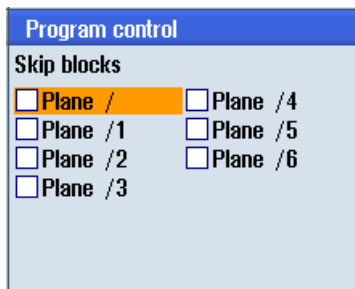


2. Press the <AUTO> or <MDA> key.



3. Press the "Prog. ctrl." and "Skip blocks" softkeys.

The "Program Control" window appears and shows a list of skip levels.



4.9 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.



Software option

You require the "Extended operator functions" option for the overstore function (for 828D).

Precondition

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

Procedure



1. Open the program 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.

4.9 Overstore



5. Press the "Back" softkey.
The "Overstore" window closes.



6. Press the <CYCLE START> key again.
The program selected before overstore 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.

4.10 Editing a program

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

Note

The maximum block length is 512 characters.

Calling the editor

- The editor is started via the "Program correction" function in the "Machine" operating area.
- 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, when this was not explicitly exited via the "Close" softkey.

Note

Please note that the changes to programs stored in the NC memory take immediate effect. You can exit the editor only after you have saved the changes.

If you are editing on a local drive or external drives, you can also exit the editor without saving, depending on the setting.

Exit the program correction mode using the "Close" softkey to return to the "Program manager" operating area.

See also

Editor settings (Page 157)

Correcting a program (Page 131)

Opening and closing the program (Page 699)

Generating a G code program (Page 206)

4.10.1 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.

- Exact expression

Activate this option if you want to search for place holders in program lines (e.g. "*" or "?") that you want to replace.

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
-

Requirement

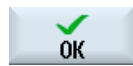
The desired program is opened in the editor.

Procedure



1. Press the "Search" softkey.
A new vertical softkey bar appears.
The "Search" window opens at the same time.
2. Enter the desired search term in the "Text" field.
3. Select "Whole words" if you want to search for whole words only.
- OR -
Activate the "Exact expression" checkbox if, for example, you want to search for place holders ("*", "?") in program lines.
4. Position the cursor in the "Direction" field and choose the search direction (forward, backward) with the <SELECT> key.





5. Press the "OK" softkey to start the search.

If the text you are searching for is found, the corresponding line is highlighted.





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

- OR -



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

Further search options

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

4.10.2 Replacing program text

You can find and replace text in one step.

Requirement

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 157)

4.10.3 Copying/pasting/deleting a program block

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.

Deleting program blocks



Use the "Cut" softkey to delete selected program blocks.

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.

Note

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

See also

Opening additional programs (Page 156)

4.10.4 Renumbering a program

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

Requirement

The program is opened in the editor.

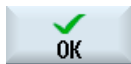
Procedure



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, select the program blocks whose block numbering you want to edit.

4.10.5 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 ShopTurn machining steps) to form program blocks.

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 ShopTurn 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 is opened.



3. Select the required program blocks that you wish to combine to form a block.



4. Press the "Build group" softkey. The "Build new group" window is opened.



5. Enter a designation for the block and press the "OK" softkey.



6. Press the ">>" and "View" softkeys.



7. Press the "Open all blocks" softkey if you wish to display the program with all blocks.



8. Press the "Close all blocks" softkey, if you wish to display the program again in a structured form.

4.10.6 Opening additional programs

You have the option of viewing and editing two 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.

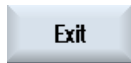


1. In the program manager, select the programs that you wish to open and view in the multiple editor and then press on the "Open" softkey.

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.

Requirement

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 153)

4.10.7 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 editable when "Yes" is available under "Number automatically".
Increment	Defines the increment used for the block numbers. The field is only editable when "Yes" is available under "Number automatically".
Display hidden lines	Yes: Hidden lines marked with "***HD" (hidden) will be displayed. No: Lines marked with ";*HD*" are not displayed. Note: Only visible program lines are taken into account with the "Search" and "Search and Replace" functions.
Display block end as an icon	The "CFLF" (line feed) symbol ¶ is displayed at the block end.
Scroll horizontally	Yes: A horizontal scrollbar is displayed. In this way, you can scroll horizontally to the end of long lines that would otherwise wrap.
Automatic save (only local and external drives)	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.
Visible programs	<ul style="list-style-type: none"> • 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.

Setting	Meaning
Width of the program with focus	Here, you enter the width of the program that has the input focus, in the editor as a percentage of the window width.
Cut only after selecting	Yes: The cut of program sections is possible only when program lines have been selected, i.e. the "Cut" softkey only then is enabled. No: The "Cut" softkey is available by default.

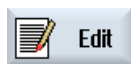
Note

All entries that you make here are effective immediately.

Procedure



1. Select the "Program" operating area



You have activated the editor.



2. Press the ">>" and "Settings" softkeys.
The "Settings" window appears.



3. Make the desired changes here and press the "OK" softkey to confirm your settings.

See also

Replacing program text (Page 151)

4.11 Mold making view

4.11.1 Mold making view

For large mold making programs, as provided by CAD systems, you have the option, using a fast view, to display the machining paths. This allows you to obtain a fast overview of the program and possibly correct it.

Checking the program

For instance, you can check whether

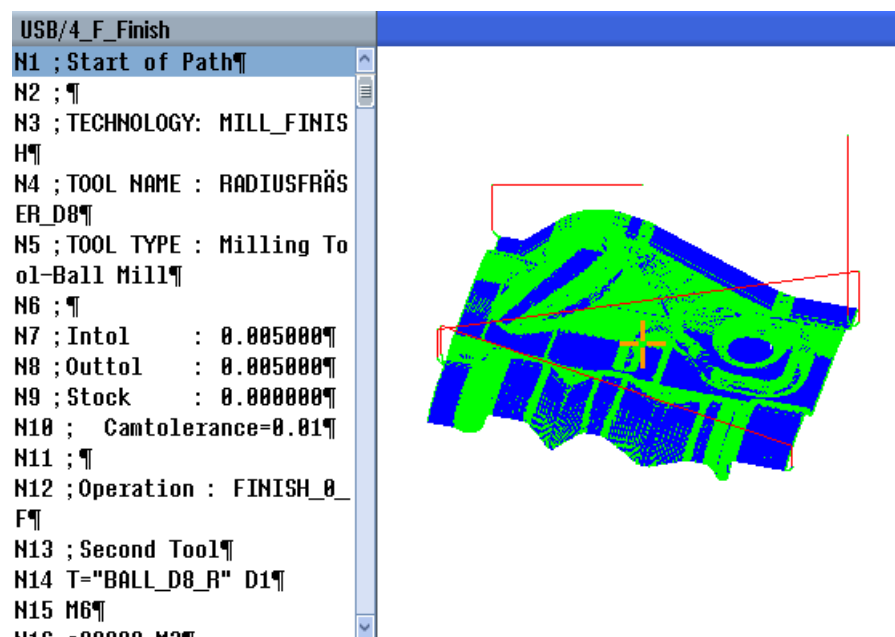
- The programmed workpiece has the correct form (shape)
- Whether there are large traversing errors
- If yes, which block must be corrected
- How the approach and retraction are performed

Simultaneous view of the program and mold making view

In the editor, next to the program block display, switch-in the graphic view.

At the left in the editor, if you set the cursor to an NC block with position data, then this NC block is selected in the graphic view.

If you select a point to the right in the graphic view - then the NC block is selected in the left-hand part of the editor - i.e. in the other direction. This is how you jump directly to a position in the program in order to edit a program block for example.



NC blocks that can be interpreted

Following NC blocks are supported for the mold building 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
 - Polynomial
POLY with X, Y, Z or PO[X] PO[Y] PO[Z]
 - B-splines
BSPLINE with degree SD (SD < 6) node PL weights PW
 - 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
 - Rotary axis programming with ORIAXES or ORIVECT using PO[A] POS[b] PO[C] for POLY
 - Orientation vector programming with ORIVECT using A3, B3, C3 for G0, G1, G2, G3, CIP
 - Orientation curve with ORICURVE using XH, YH, ZH, for G0, G1, G2, G3, CIP, POLY, BSPLINE
 - Orientation curve with ORICURVE using PO[XH] PO[YH] PO[ZH] for POLY
 - 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

Following NC blocks are **not** supported for the mold making view.

- Helix programming
- Rational polynomials
- Other G codes or language commands

All blocks that cannot be interpreted are simply ignored

Changing and adapting the mold making view

Just the same as for simulation and simultaneous recording, you have the option of changing and adapting the simulation graphical representation in order to achieve the optimum view.

- Increasing or reducing the size of the graphic
- Moving the graphic
- Rotating the graphic
- Changing the section

4.11.2 Starting the mold making view

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 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 graphic is displayed in the right half of the editor.

All of the points and paths programmed in the part program are shown in the graphical representation.



5. Press the "Graphic" softkey in order to hide the graphic and to view the program as usual in the editor

- OR -



Press the "NC blocks" softkey in order to hide the G code blocks and therefore only to display the graphic.

4.11.3 Specifically jump to the program block

If you notice anything peculiar in the graphic or identify an error, then from this location, you can directly jump to the program block involved to possibly edit the program.

Preconditions

- 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 is shown in the diagram.



2. Using the cursor keys, move the cross-hairs to the appropriate position in the graphic.



3. Press the "Select NC block" softkey.
The cursor jumps to the associated program block in the editor.

4.11.4 Searching for program blocks

Using the "Search" function, you can search for specific program blocks as well as edit programs; you can do this in one step by replacing the text that you are searching for by a new text.

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 150)
Replacing program text (Page 151)

4.11.5 Changing the view

4.11.5.1 Enlarging or reducing the graphical representation

Precondition

- The mold making view has been started.
- The "Graphic" softkey is active.

Procedure



...



1. Press the <+> and <-> keys if you wish to enlarge or reduce the graphic display.

The graphic display enlarged or reduced from the center.



- OR -

Press the "Details" and "Zoom +" softkeys if you wish to increase the size of the segment.



- OR -

Press the "Details" and "Zoom -" softkeys if you wish to decrease the size of the segment.



- OR -

Press the "Details" and "Auto zoom" softkeys if you wish to automatically adapt the segment to the size of the window.

The automatic scaling function "Fit to size" takes account of the largest expansion of the workpiece in the individual axes.

Note

Selected section

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

4.11.5.2 Modifying the viewport

Use the magnifying glass if you would like to move, increase or reduce the size of the section of the mold making view, e.g. to view details or display the complete workpiece.

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.

4.12 Displaying G functions and auxiliary functions

4.12.1 Selected G functions

16 selected G groups are displayed in the "G Function" window.

Within a G group, the G function currently active in the controller is displayed.

Some G codes (e.g. G17, G18, G19) are immediately active after switching the machine control on.

Which G codes are always active depends on the settings.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

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 dimensioning inches/metric (e.g. G70, G700)
G group 14	Workpiece dimensioning absolute/incremental (G90)
G group 15	Feedrate type (e.g. G93, G961, G972)
G group 16	Feedrate override on 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)

Group	Meaning
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 dimensioning inches/metric (e.g. G70, G700)
G group 14	Workpiece dimensioning absolute/incremental (G90)
G group 15	Feedrate type (e.g. G93, G961, G972)
G group 16	Feedrate override on 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



Machine

1. Select the "Machine" operating area.



2. Press the <JOG>, <MDA> 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 again.

The G groups selection displayed in the "G Functions" window may differ.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

References

For more information about configuring the displayed G groups, refer to the following document:

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

4.12.2 All G functions

All G groups and their group numbers are listed in the "G Functions" window.

Within a G group, only the G function currently active in the controller is displayed.

Additional information in the footer

The following additional information is displayed in the footer:

- Current 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 zero offsets
- Spindle speed
- Path feedrate
- Active tool

Procedure

1. Select the "Machine" operating area.



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

...



3. Press the ">>" and "All G functions" softkeys.
The "G Functions" window is opened.

**4.12.3 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.

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.

References

Programming Guide Job Planning (PGA) Chapter: Motion-synchronous 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 automatic mode

Note

Numbers ranging from 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.

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.

...



4.13 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 refer to the machine manufacturer's specifications.

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 track how much of the current program has been completed as a percentage by using the progress bar.
The display only appears when the program is run a second time.
If you are executing the program from an external location, the program loading progress is displayed here.
- 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 workpiece 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". This value can be corrected if necessary.

After the defined number of workpieces is reached, the current workpieces display is automatically reset to zero.

See also

Entering the number of workpieces (Page 246)

4.14 Setting for automatic mode

Before machining a workpiece, you can test the program in order to identify programming errors early on. Use the dry run feedrate for this purpose.

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

Dry run feedrate

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

Reduced rapid traverse

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

Displaying measurement results

Using an MMC command, you can display measurement results in a part program:

You can define the following settings:

- 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.

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 is opened.



4. In "DRY run feedrate," enter the desired dry run speed.
5. Enter the desired percentage in the "Reduced rapid traverse RG0" field.
RG0 has not effect if you do not change the specified amount of 100%.



6. Enter "Automatic" in the "Display measurement result" box if the measurement result window should be automatically opened, or "Manual", if the measurement result window should be opened by pressing the "Measurement result" softkey.

References

Programming Manual Measuring Cycles / 840D sl/828D

Note

The feedrate can be changed while the operation is running.

See also

Program control (Page 143)

Simulating machining

5.1 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 uses the correct proportions of the workpiece, tools, chuck, counterspindle and tailstock for the screen display.

For the spindle chuck and the tailstock, dimensions are used that are entered into the "Spindle Chuck Data" window.

For non-cylindrical blanks, the chuck closes up to the contour of the cube or polygon.

Depth display

The depth infeed is color-coded. The depth display indicates the actual depth at which machining is currently taking place. The following applies to the color-coded depth display: "The deeper, the darker".

Definition of 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 zero offset.

Programming a blank (example)

```
G54 G17 G90
WORKPIECE( , , , "Cylinder", 112, 0, -50, -80, 00, 155, 100)
T="NC-SPOTDRILL_D16
```

Machine references

The simulation is implemented as workpiece simulation. This means that it is not assumed that the zero offset has already been precisely scratched or is known. In spite of this, unavoidable Machine references are in the programming, such as for example, the tool change point in the Machine, the park position for the counterspindle in the Machine or the position of the counterspindle slide. Depending on the actual zero offset - in the worst case - these machine references can mean that collisions are shown in the simulation that would not occur for a realistic zero offset - or vice versa, collisions are not shown, which could occur for a realistic zero offset. This is the reason that in ShopTurn programs, in the case of a simulation, the program header calculates an appropriate zero offset for the main spindle - or where relevant for the counterspindle - from the specified chuck dimensions.

Programmable frames

All frames and zero offsets are taken into account in the simulation.

Note

Manually swiveled axes

Note that swivel movement in simulation and during simultaneous recording is also displayed when the axes are swiveled manually at the start.

Display of the traversing paths

The traversing paths of the tool are shown in color. Rapid traverse is red and the feedrate is green.

Note

Displaying the tailstock

The tailstock is only visible with the option "ShopMill/ShopTurn".



Machine manufacturer

Please also refer to the machine manufacturer's specifications.

References

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

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.

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:

- Side view
- Half section
- Front view
- 3D view
- 2-window

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 option "3D simulation of the finished part" for the 3D view.

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

Determining the program runtime

The program runtime is determined when executing the simulation. The program runtime is 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 screen. 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 zero offset to the workpiece coordinate system.

The simultaneous recording starts at the position at which the machine is currently located.

Constraint

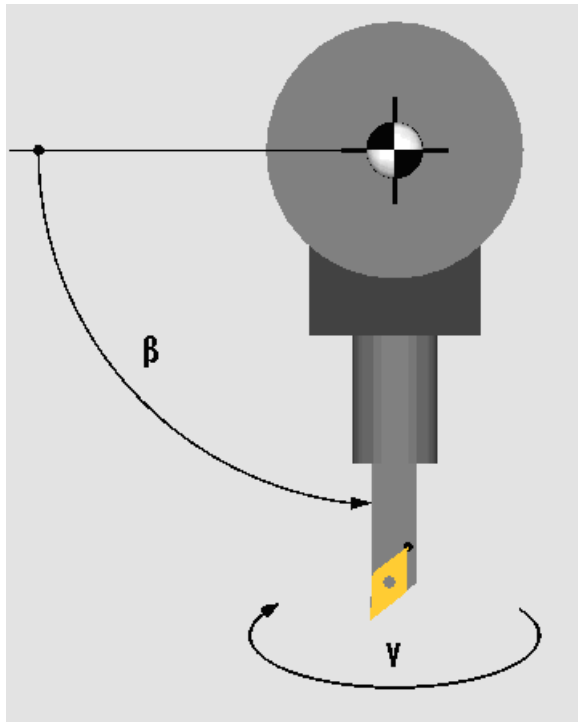
- Referencing: G74 from a program run does not function.
- Alarm 15110 "REORG block not possible" is not displayed.
- Compile cycles are only partly supported.
- No PLC support.
- Axis containers are not supported.
- Swivel tables with non-swiveling offset vectors are not supported.

Supplementary conditions

- All of the existing data records (toolcarrier / TRAORI, TRANSMIT, TRACYL) are evaluated and must be correctly commissioned for correct simulation.
- Transformations with swiveled linear axis (TRAORI 64 - 69) as well as OEM transformations (TRAORI 4096 - 4098) are not supported.
- Changes to the toolcarrier or transformation data only become effective after Power On.
- Transformation change and swivel data record change are supported. However, a real kinematic change is not supported, where a swivel head is physically changed.
- 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.

Example

An example for supported kinematics is a lathe with B axis:



Lathe with B axis

See also

Spindle chuck data (Page 96)

5.2 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

Feedrate override is also active during simulation.

You can change the feedrate during the simulation via the user interface.

0 %: The simulation stops.

100 %: The program is executed as quickly as possible.

See also

Changing the feedrate (Page 192)

Simulating the program block by block (Page 193)

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 required program.

The selected program is opened in the "Program" operating area in the editor.



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.



Software option

You require the option "3D simulation of the finished part" for the 3D view.

5.3 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 the programming.

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.



Software option

You require the option "Simultaneous recording (real-time simulation)" for the simultaneous recording.

Procedure



1. Load a program in the "AUTO" mode.
2. Press the "Prog. ctrl." softkey and activate the checkboxes "PRT no axis movement" and "DRY run feedrate".

The program is executed without axis movement. The programmed feedrate is replaced by a dry run feedrate.



3. Press the "Sim. rec." softkey.



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.

5.4 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 the 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 unmachined part 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, then the traversing paths generated in the intermediate time will not be displayed.
-

5.5 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:

- Side view
- Half section
- Front view
- 3D view
- 2-window

5.5.1 Side view



1. Start the simulation.
2. Press the "Side view" softkey.

The side view shows the workpiece in the Z-X plane.

Changing the display

You can increase or decrease the size of the simulation graphic and move it, as well as change the segment.

5.5.2 Half section



1. Start the simulation.
2. Press the "Other views" and "Half section" softkeys.

The half view shows the workpiece cut in the Z-X plane.

Changing the display

You can increase or decrease the size of the simulation graphic and move it, as well as change the segment.

5.5.3 Face view



1. Start the simulation.
2. Press the "Other views" and "Face view" softkeys.



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

Changing the display

You can increase or decrease the size of the simulation graphic and move it, as well as change the segment.

5.5.4 3D view



1. Start the simulation.
2. Press the "3D view" softkey.



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 197)

5.5.5 2-window



1. Start the simulation.
2. Press the "Additional views" and "2-window view" softkeys.

The 2-window view contains a side view (left-hand window) and a front view (right-hand window) of the workpiece. The viewing direction is always from the front to the cutting surface even if machining is to be performed from behind or from the back side.

Changing the display

You can increase or decrease the size of the simulation graphic and move it, as well as change the segment.

5.6 Graphical display

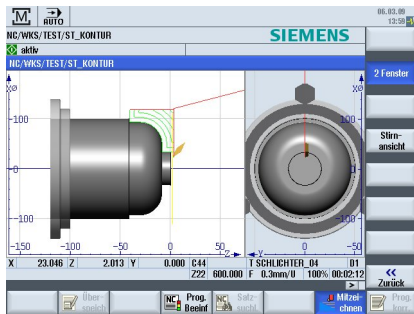


Figure 5-1 2-window view

Active window

The currently active window has a lighter background than the other view windows.

Switch over the active window using the <Next Window> key.

You can change the workpiece display here, e.g. increase or decrease the size, turn it and move it.

Some of the actions that you perform in the active window also have a simultaneous effect in other view windows.

Display of the traversing paths

- Rapid traverse = red
- Feed = green

5.7 Editing the simulation display

5.7.1 Blank display

You have the option of replacing the blank defined in the program or to define a blank for programs in which a blank definition cannot be inserted.

Note

The unmachined part can only be entered if simulation or simultaneous recording is in the reset state.

Procedure



1. The simulation or the simultaneous recording is started.
2. Press the ">>" and "Blank" softkeys.
The "Blank Input" windows opens and displays the pre-assigned values.
3. Enter the desired values for the dimensions.
4. Press the "Accept" softkey to confirm your entries. The newly defined workpiece is displayed.

5.7.2 Showing and hiding the tool path

The path display follows the programmed tool path of the selected program. The path is continuously updated as a function of the tool movement. The tool paths can be shown or hidden as required.

Procedure



1. The simulation or the simultaneous recording is started.
2. Press the ">>" softkey.
The tool paths are displayed in the active view.
3. Press the softkey to hide the tool paths.
The tool paths are still generated in the background and can be shown again by pressing the softkey again.
4. Press the "Delete tool path" softkey.
All of the tool paths recorded up until now are deleted.

5.8 Program control during the simulation

5.8.1 Changing the feedrate




You can change the feedrate at any time during the simulation.


You can track the changes in the status line.


Note

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

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%.



- OR -
Press the "Override 100%" softkey to set the feedrate to the maximum value.


- OR -
Press the "<<" softkey to return to the main screen and perform the simulation with changed feedrate.


Note








Toggle between "Override +" and "Override -"

-  Press the <CTRL> and <Cursor down> or <Cursor up> keys to toggle between the "Override +" and "Override -" softkeys.
-  

5.8.2 Simulating the program block by block



You can control the program execution during simulation, i.e. execute a program block by block, as when executing a program.

Procedure

- | | |
|---|--|
| | 1. Simulation is started. |
|  | 2. Press the "Program control" and "Single block" softkeys. |
|  | |
|  | 3. Press the "Back" and "Start SBL" softkeys. |
|  | The pending block of the program 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. |
|  | |

Note

Enabling/disabling single block

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

5.9 Editing and adapting a simulation graphic

5.9.1 Enlarging or reducing the graphical representation

Precondition

The simulation or the simultaneous recording is started.

Procedure



...



1. Press the <+> and <-> keys if you wish to enlarge or reduce the graphic display.

The graphic display enlarged or reduced from the center.

- OR -



Press the "Details" and "Zoom +" softkeys if you wish to increase the size of the segment.

- OR -



Press the "Details" and "Zoom -" softkeys if you wish to decrease the size of the segment.

- OR -



Press the "Details" and "Auto zoom" softkeys if you wish to automatically adapt the segment to the size of the window.

The automatic scaling function "Fit to size" takes account of the largest expansion of the workpiece in the individual axes.

Note

Selected section

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

5.9.2 Panning a graphical representation

Precondition

The simulation or the simultaneous recording is started.

Procedure



1. Press a cursor key if you wish to move the graphic up, down, left, or right.



5.9.3 Rotating the graphical representation

In the 3D view you can rotate the position of the workpiece to view it from all sides.

Precondition

Simulation has been started and the 3D view is selected.

Procedure



1. Press the "Details" softkey.



2. Press the "Rotate view" softkey.



2. 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.



5.9.4 Modifying the viewport

If you would like to move, enlarge or decrease the size of the segment of the graphical display, e.g. to view details or display the complete workpiece, use the magnifying glass.

Using the magnifying glass, you can define your own segment and then increase or decrease 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 section.

5.9.5 Defining cutting planes

In the 3D view, you have the option of "cutting" the workpiece and therefore displaying certain views in order to show hidden contours.

Precondition

The simulation or the simultaneous recording is started.

Procedure



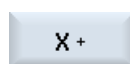
1. Press the "Details" softkey.



2. Press the "Cut" softkey.

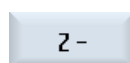


The workpiece is displayed in the cut state.



3. Press the corresponding softkey to shift the cutting plane in the required direction.

...



5.10 Displaying simulation alarms

Alarms might occur during simulation. If an alarm occurs during a simulation run, a window opens in the operating window to display it.

The alarm overview contains the following information:

- Date and time
- Deletion criterion
Specifies with which softkey the alarm is acknowledged
- Alarm number
- Alarm text

Precondition

Simulation is running and an alarm is active.

Procedure



1. Press the "Program control" and "Alarm" softkeys.
The "Simulation Alarms" window is opened and a list of all pending alarms is displayed.

Press the "Acknowledge alarm" softkey to reset the simulation alarms indicated by the Reset or Cancel symbol.
The simulation can be continued.
- OR -
Press the "Simulation Power On" softkey to reset a simulation alarm indicated by the Power On symbol.

Creating a G code program

6.1 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.

6.2 Program views

You can display a G code program in various ways.

- Program view
- Parameter screen, either with help screen or graphic view

Program view

The program view in the editor provides an overview of the individual machining steps of a program.

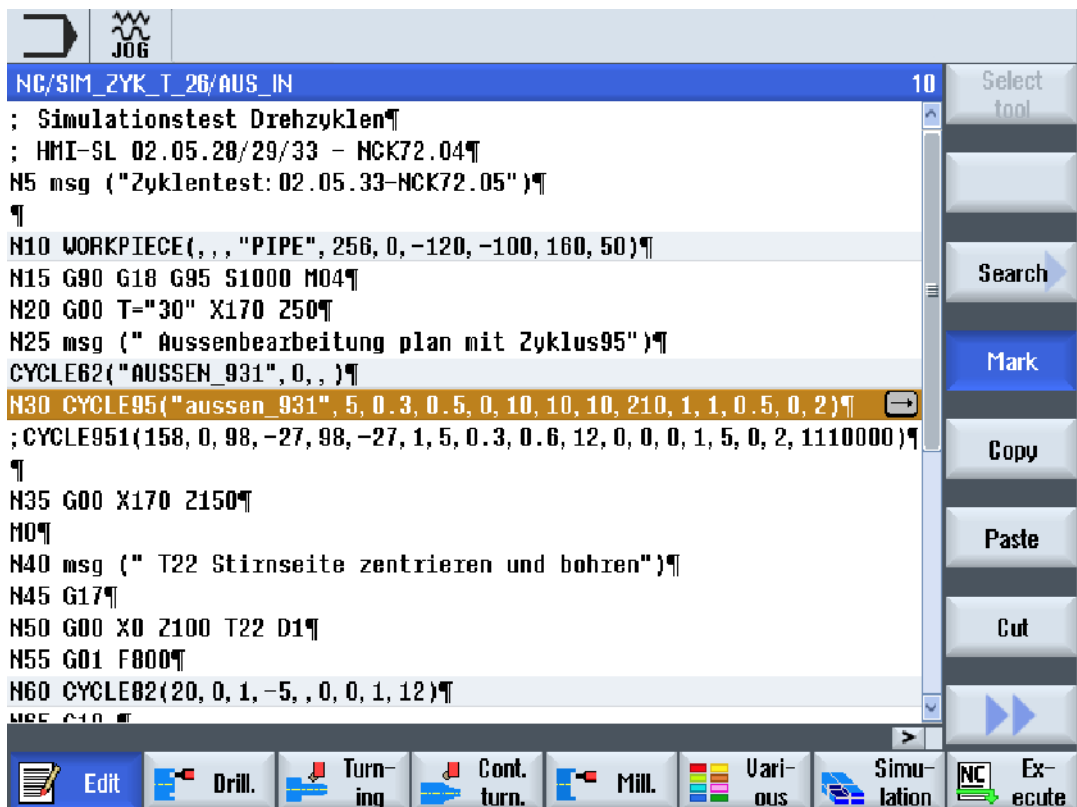


Figure 6-1 Program view of a G code program



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.

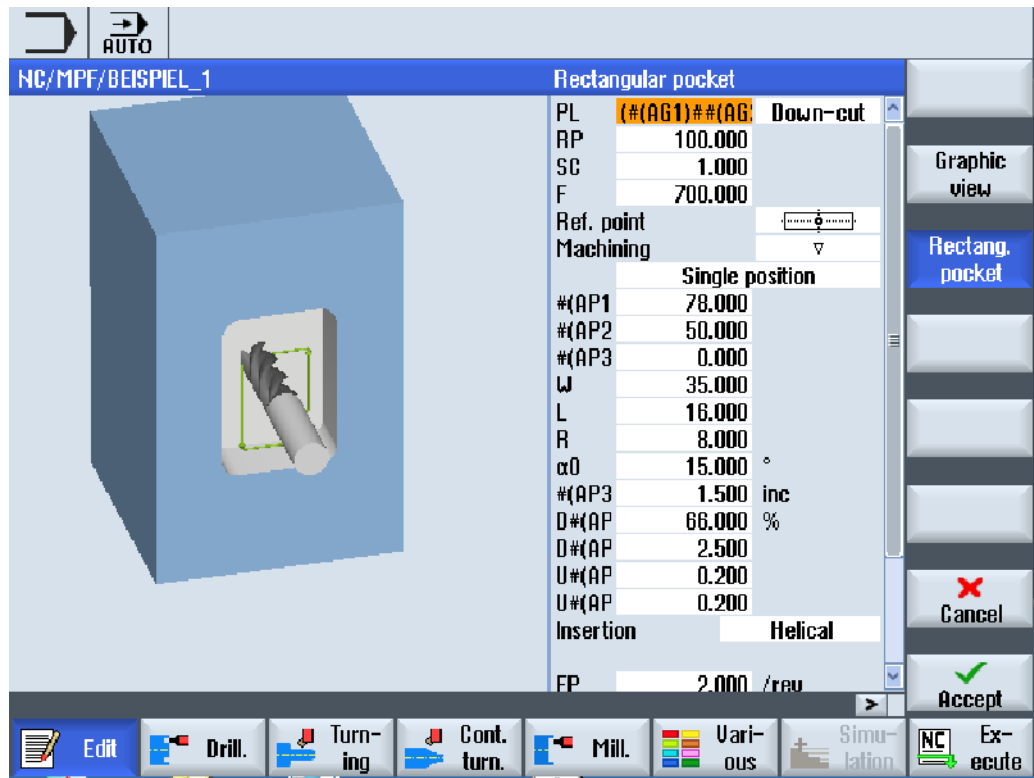


Figure 6-2 Parameter screen with help display

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 color symbols

Red arrow = tool traverses in rapid traverse

Green arrow = tool moves with the machining feedrate

Parameter screen with graphic view



Using the "Graphic view" softkey, you can toggle between the help screen and the graphic view in the screen.

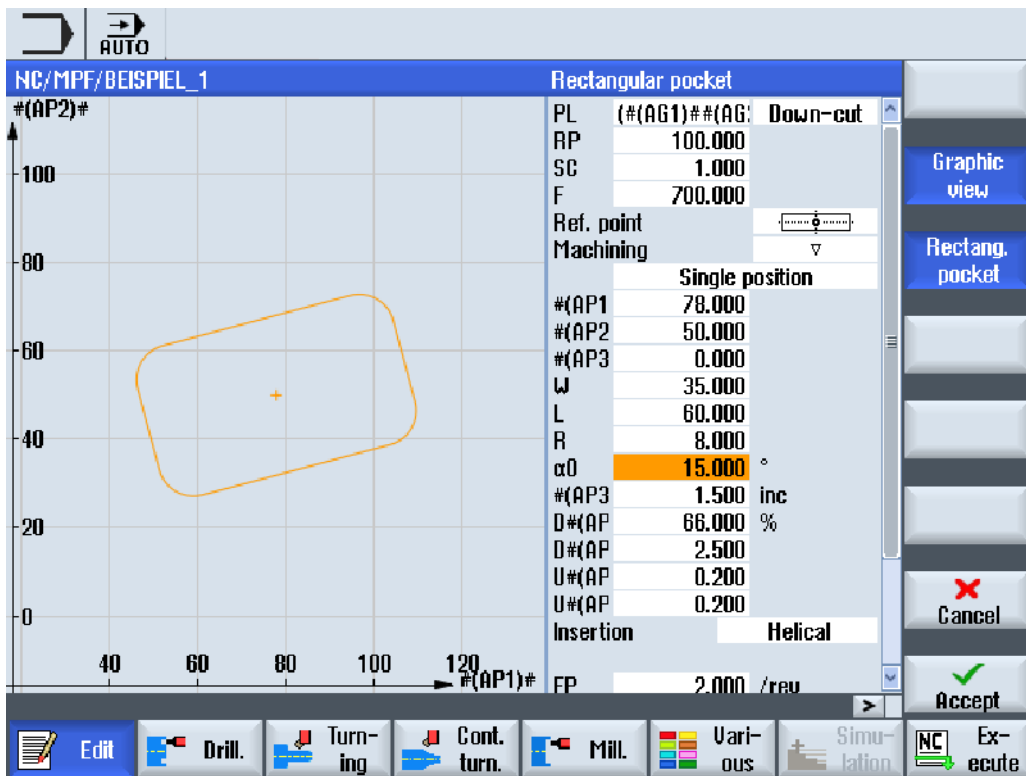


Figure 6-3 Parameter screen with a graphical view of a G code program block

6.3 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

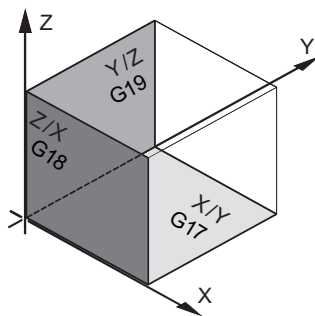
Blank input (Page 208)

6.4 Fundamentals

6.4.1 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

6.4.2 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.

6.4.3 Programming a tool (T)

Calling a tool

1. You are in a 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.

6.5 Generating a G code program

Create a separate program for each new workpiece that you would like to produce. The program contains the individual machining steps that must be performed to produce the workpiece.

Part programs in the G code can be created under the "Workpieces" folder or under the "Part programs" folder.

Procedure



1. Select the "Program Manager" operating area.



2. Select the required archiving location.

Creating a new part program



3. Position the cursor on the folder "Part programs" and press the "New" softkey.



The "New G Code Program" window opens.



4. Enter the required name and press the "OK" softkey.
The name can contain up to 28 characters (name + dot + 3-character extension). You can use any letters (except accented), digits or the underscore symbol (_).

The program type (MPF) is set by default.

The project is created and opened in the Editor.

Creating a new part program for a workpiece



5. Position the cursor on the folder "Workpieces" and press the "New" softkey.



The "New G Code Program" window opens.



6. Select the file type (MPF or SPF), enter the desired name of the program and press the "OK" softkey.

The project is created and opened in the Editor.

7. Enter the desired G code commands.

See also

Changing a cycle call (Page 218)

Selection of the cycles via softkey (Page 211)

Creating a new workpiece (Page 704)

6.6 Blank input

6.6.1 Function

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 (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 the blank <ul style="list-style-type: none"> • Main spindle • Counterspindle Note: If the machine does not have a counterspindle, then the entry field "Data for" is not applicable.	
Blank 	Selecting the blank <ul style="list-style-type: none"> • Centered cuboid • Tube • Cylinder • Polygon • Delete 	
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

6.7 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 always repeat.

You will find the following parameters in every input screen for a cycle in a G code program.

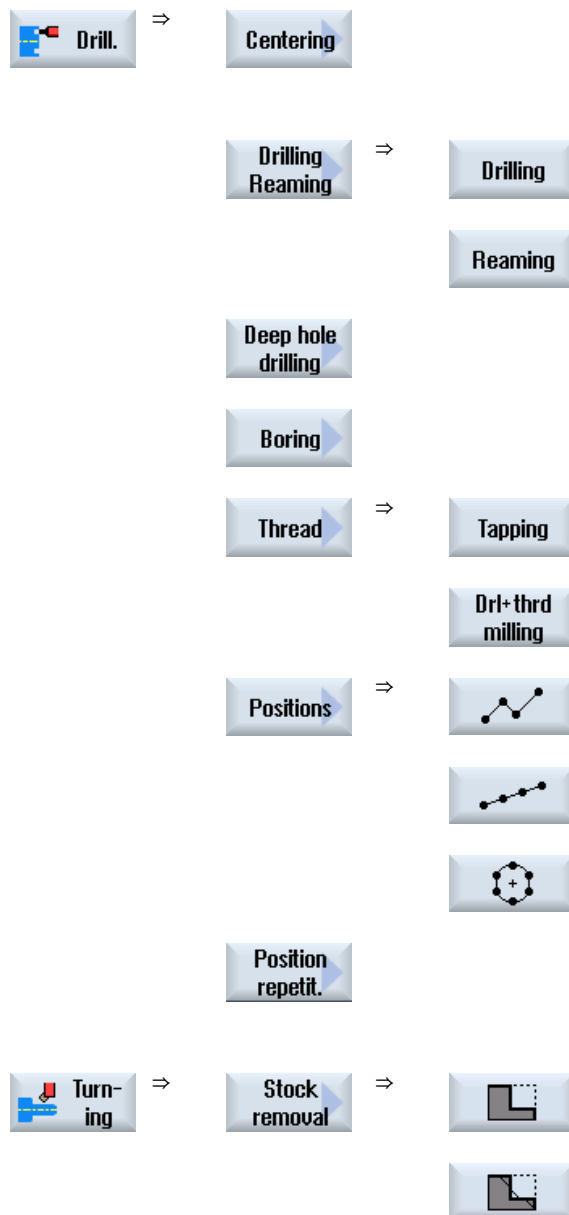
Parameter	Description	Unit
PL 	Each input screen has a selection box for the planes, if the planes have not been specified by NC machine data. Machining plane: <ul style="list-style-type: none"> • G17 (XY) • G18 (ZX) • G19 (YZ) 	
Milling direction  - only for milling	When machining a pocket, a longitudinal slot or a spigot, the machining direction (climbing or conventional) and the spindle direction are taken into account in the tool list. The pocket is then machined in a clockwise or counterclockwise direction. During path milling, the programmed contour direction determines the machining direction.	
RP	Retraction plane (abs) During machining the tool traverses 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 traverses at the machining feedrate away from the workpiece to the safety clearance level. 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. 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) Acts in relation to the reference point. The direction in which the safety clearance is active is automatically determined by the cycle. The safety clearance must be entered as an incremental value (without sign).	mm
F	Feedrate The feedrate F (also referred to as the machining feedrate) specifies the speed at which the axes move when machining 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.	

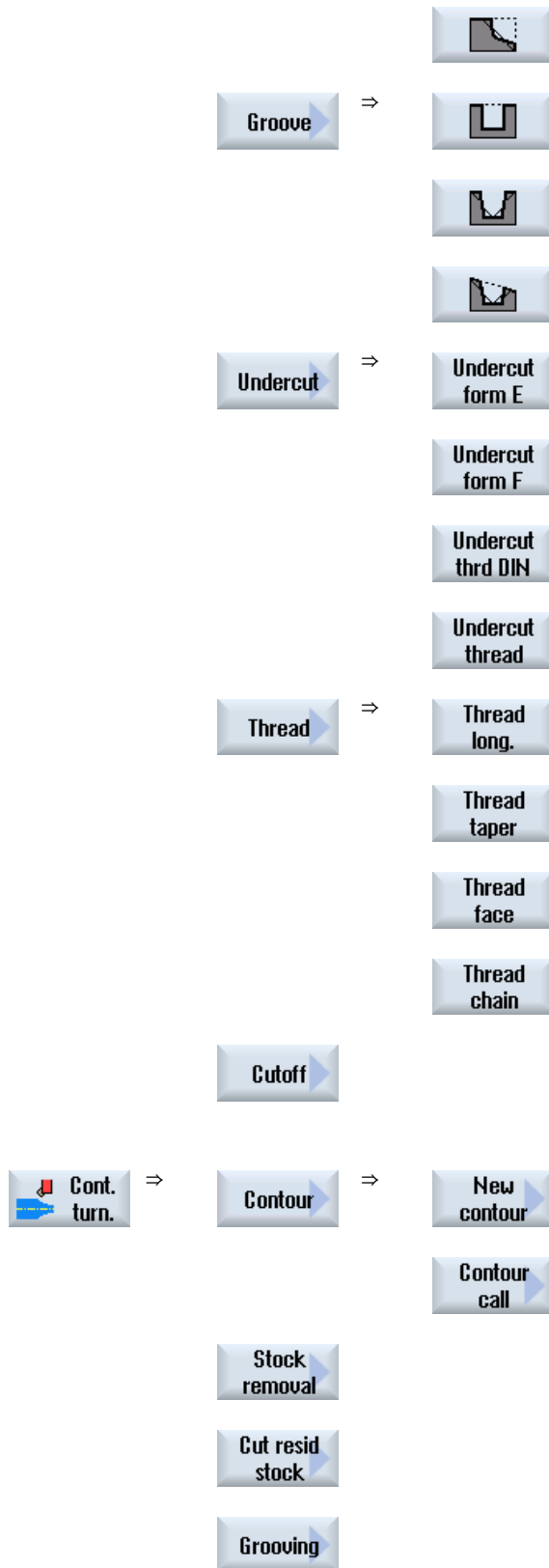
6.8 Selection of the cycles via softkey

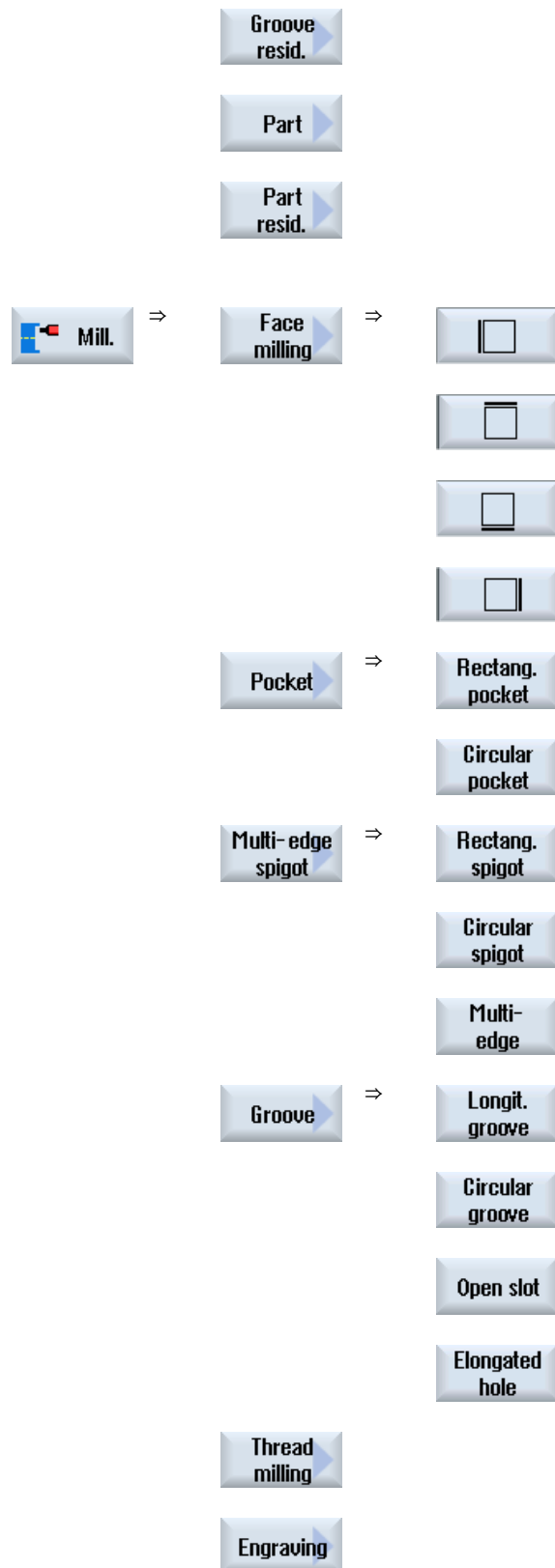
Overview of the machining steps

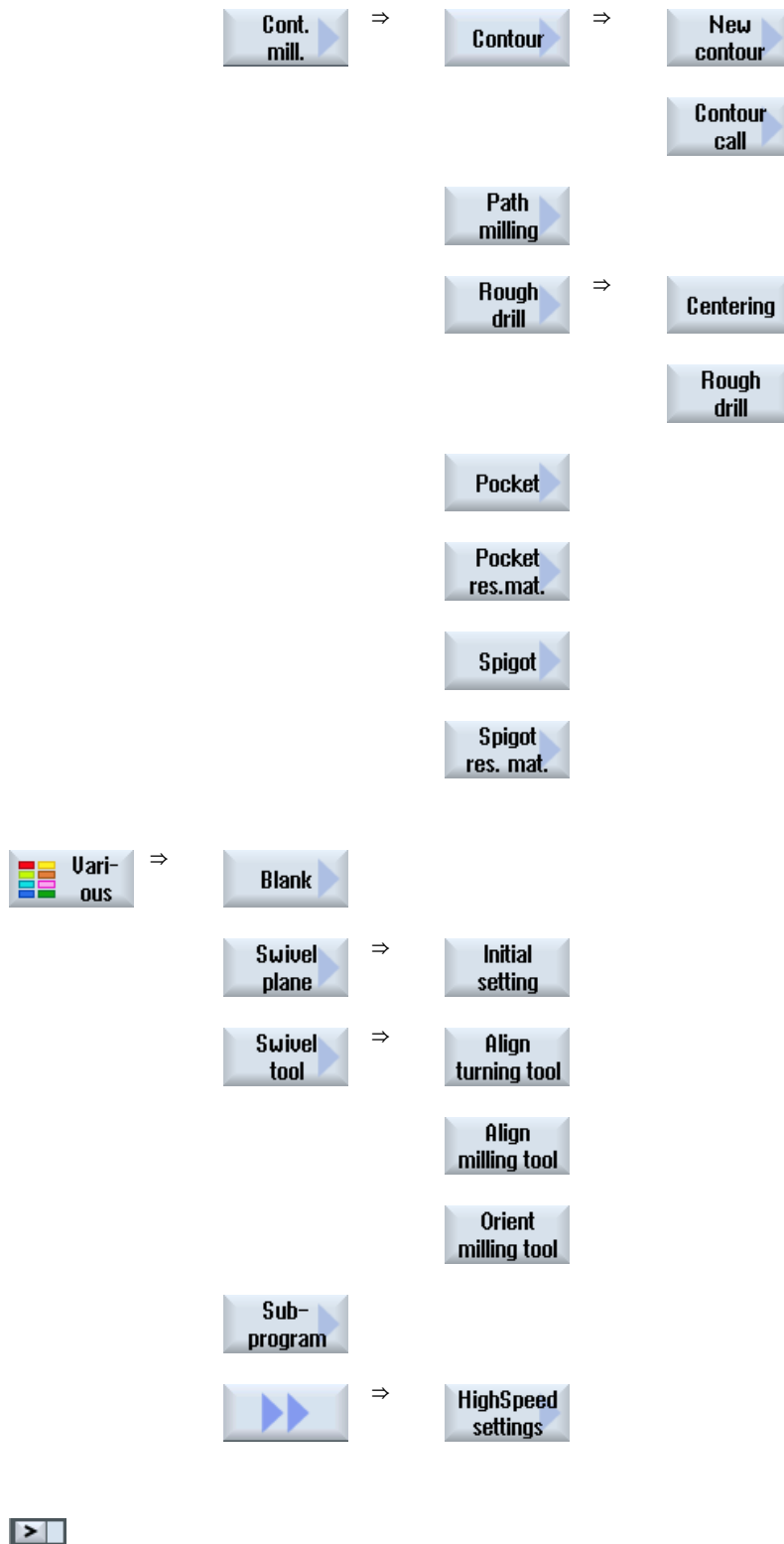
The following machining steps are available.

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.











⇒

A menu tree with all of the available measuring versions of the measuring cycle function "Measure workpiece" can be found in the following reference:

Programming Manual Measuring cycles / SINUMERIK 840D sl/828D



⇒

A menu tree with all of the available measuring versions of the measuring cycle function "Measure tool" can be found in the following reference:

Programming Manual Measuring cycles / SINUMERIK 840D sl/828D

See also

General (Page 285)

Generating a G code program (Page 206)

6.9 Calling technology cycles

6.9.1 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.

For additional information, please refer to the following references:

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

Cycle support

Example



...



1. Use the softkeys to select whether you want support for programming contours, turning, drilling or milling cycles.

2. Select the desired cycle using the softkey in the vertical softkey bar.
3. Enter the parameters and press the "Accept" softkey.

The cycle is transferred to the editor as G code.

6.9.2 Setting data for cycles

Cycle functions can be influenced and configured using machine and setting data.

For additional information, please refer to the following references:

Commissioning Manual SINUMERIK Operate / SINUMERIK 840D sl

6.9.3 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 note 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.

6.9.4 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)
```

6.9.5 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 206)


6.9.6 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.

6.9.7 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.

6.10 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".

References

You will find a more detailed description on how to use measuring cycles in:
Programming Manual Measuring cycles / SINUMERIK 840D sl/828D

Creating a ShopTurn program

7.1 Graphic program control, ShopTurn programs

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 ShopTurn machining step programs.

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)

7.2 Program views

You can display a ShopTurn program in various views:

- Machining schedule
- Programming graphics
- Parameter screen, either with help display or programming graphics

Machining schedule

The machining schedule in the editor provides an overview of the individual machining steps of a program.

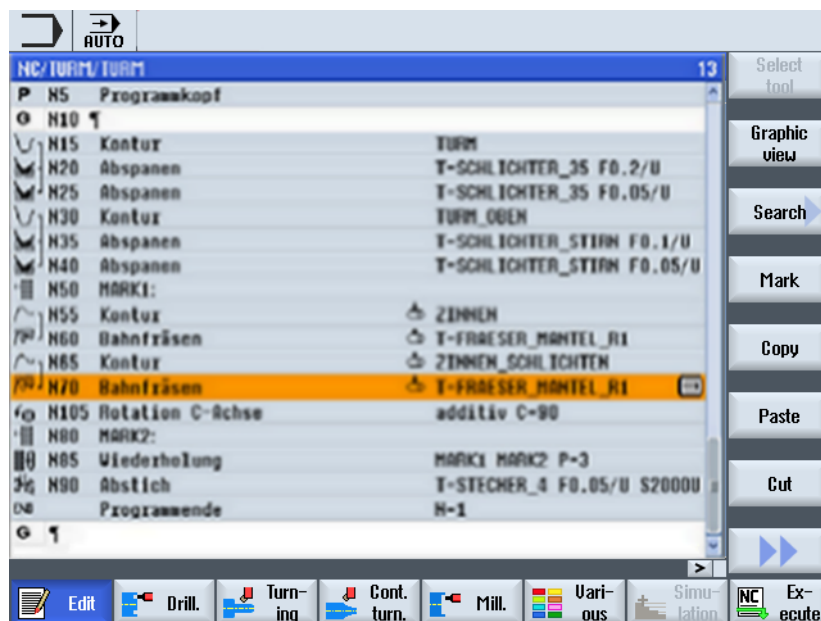
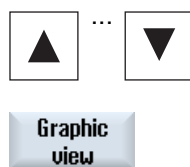


Figure 7-1 Machining schedule of a ShopTurn program



1. You can move between the program blocks in the machining schedule using the <Cursor up> and <Cursor down> keys.
2. Press the "Graphics view" softkey to display the programming graphics.

Programming graphics

The programming graphics show the contour of the workpiece as a dynamic graphic with dotted lines. The program block selected in the machining schedule is highlighted in color in the programming graphics.

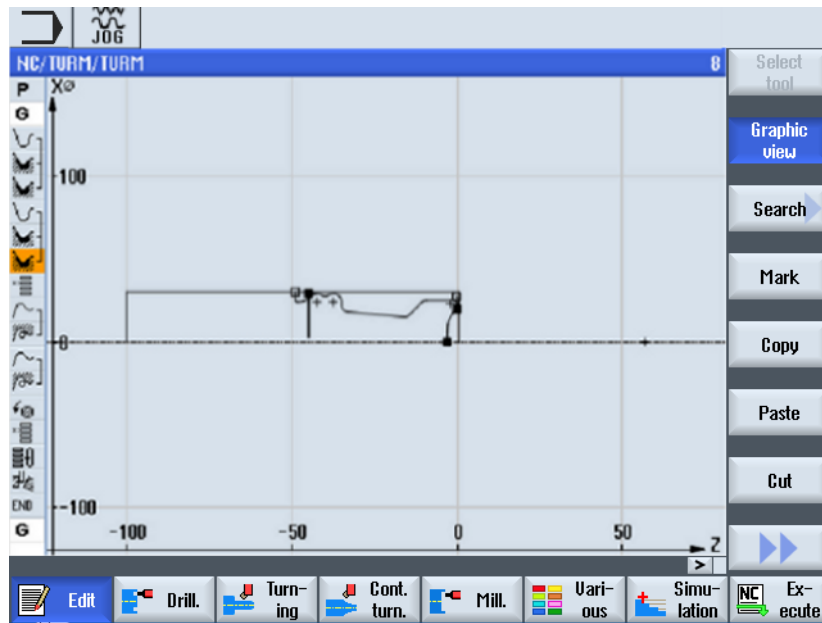


Figure 7-2 Programming graphics of a ShopTurn program

Parameter screen with help display and programming graphics



1. Press the <Cursor right> key to open a selected program block or cycle in the machining schedule.

The associated parameter screen with help screen is then displayed.



2. Press the "Graphic views" softkey.

The programming graphics of the selected program block is displayed.

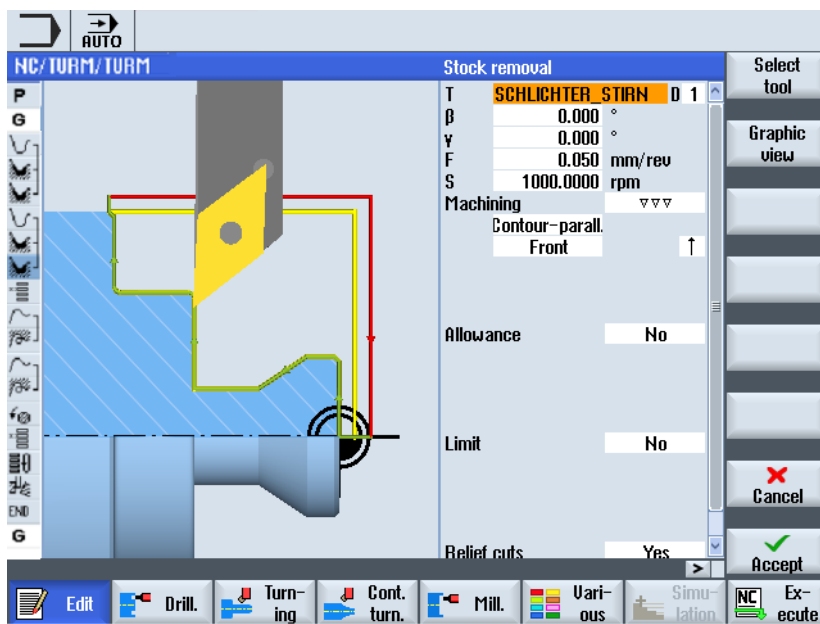


Figure 7-3 Parameter screen with dynamic help display

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.



In the screen, you can toggle between the help display and the program graphics using the "Graphic view".

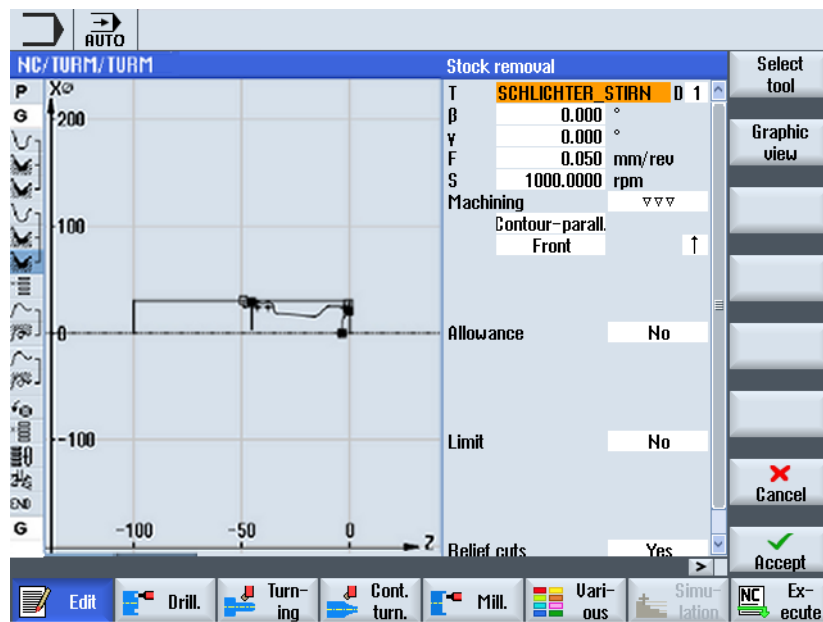


Figure 7-4 Parameter screen with programming graphics

7.3 Program structure

A machining step program is divided into three sub-areas:

- Program header
- Program blocks
- End of program

These sub-areas form a process 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 turning", "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 process plan.

In the technology blocks, specify how and in what form the machining should take place, e.g. centering first, and then drilling. In the positioning blocks, determine the positions for the drilling or milling machining, e.g. position the drill-holes in a full circle on the face surface.

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.

See also

Entering the number of workpieces (Page 246)

7.4 Fundamentals

7.4.1 Machining planes

A workpiece can be machined on different planes. Two coordinate axes define a machining plane. On lathes with X, Z, and C axes, three planes are available:

- Turning
- Face
- Peripheral

Machining planes, face and peripheral

The face and peripheral machining planes require the CNC-ISO functions "Front surface machining" (Transmit) and "Cylindrical peripheral transformation" (Tracyl) to be set up.

These functions are a software option.

Additional Y axis

For lathes with an additional Y axis, the machining planes are expanded to include two more planes:

- Face Y
- Peripheral Y

Therefore, the face and peripheral planes are called Face C and Peripheral C.

Inclined axis

If the Y axis is an oblique axis (i.e. the axis is not perpendicular to the others), you can also select the "Face Y" and "Peripheral Y" machining planes and program the traversing movements in Cartesian coordinates. The control system then automatically transforms the programmed traversing movements of the Cartesian coordinate system into the traversing movements of the oblique axis.

The CNC-ISO function "Oblique Axis" (Traang) is required for the purpose of transforming the programmed traversing movements.

This function is a software option.

Selecting the machining plane

The machining plane selection is integrated into the parameter screen forms of the individual drilling and milling cycles. For turning cycles and for "axial drilling" and "axial threads", the turning plane is automatically selected. For the "straight" and "circle" functions, you must specify the machining plane separately.

The settings for the machining plane always act modally, i.e. until you select another plane.

The machining planes are defined as follows:

Turning

The turning machining plane corresponds to the X/Z plane (G18).

Face/Face C

The Face/Face C machining plane corresponds to the X/Y plane (G17). For machines without a Y axis, however, the tools can only move in the X/Z plane. The X/Y coordinates that have been entered are automatically transformed into a movement in the X and C axis.

You can use face surface machining with a C axis for drilling and milling if, for instance, you want to mill a pocket on the face surface. You can choose between the forward or rear face surface for this purpose.

Peripheral/Peripheral C

The Peripheral/Peripheral C machining plane corresponds to the Y/Z plane (G19). For machines without a Y axis, however, the tools can only move in the Z/X plane. The Y/Z coordinates that you entered are automatically transformed into a movement in the C and Z axis.

You can use peripheral surface machining with a C axis for drilling and milling if, for instance, you want to mill a slot with constant depth on the peripheral surface. You can choose between the inner or outer surface for this purpose.

Face Y

The face Y machining plane corresponds to the X/Y plane (G17). You can use the face surface machining with a Y axis for drilling and milling if, for instance, you want to mill a pocket on the face surface. You can choose between the forward or rear face surface for this purpose.

Peripheral Y

The peripheral Y machining plane corresponds to the Y/Z plane (G19). You can use peripheral surface machining with a Y axis for drilling and milling if, for instance, you want to mill a pocket with a flat base on the peripheral surface or drill holes that do not point to the center. You can choose between the inner or outer surface for this purpose.

7.4.2 Machining cycle, approach/retraction

Approaching and retracting during the machining cycle always follows the same pattern if you have not defined a special approach/retraction cycle.

If your machine has a tailstock, you can also take this into consideration when traversing.

The retraction for a cycle ends at the safety clearance. Only the subsequent cycle moves to the retraction plane. This enables a special approach/retraction cycle to be used.

Note

When selecting the traversing paths, the tool tip is always taken into account; i.e. the tool expansion is not considered. Therefore, you should ensure that the retraction planes are an appropriate distance away from the workpiece.

Approach/retraction sequence in a machining cycle

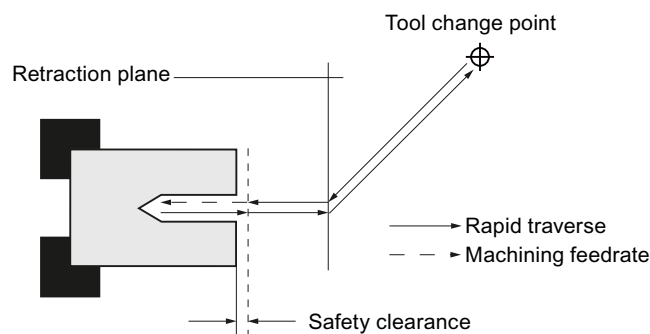


Figure 7-5 Machining cycle, approach/retraction

- The tool traverses in rapid traverse along the shortest path from the tool change point to the retraction plane, which runs parallel to the machining plane.
- After this, the tool traverses in rapid traverse to the safety clearance.
- Following this, the workpiece is then machined with the programmed machining feedrate.
- After machining, the tool retracts with rapid traverse to the safety clearance.
- The tool then continues to traverse vertically in rapid traverse to the retraction plane.
- From there, the tool traverses in rapid traverse along the shortest path to the tool change point. If the tool does not need to be changed between two machining processes, the tool traverses from the retraction plane to the next machining cycle.

The spindle (main, tool, or counter-spindle) begins to rotate immediately after the tool change.

You define the tool change point, the retraction plane, and the safety clearance in the program header.

Taking into account the tailstock

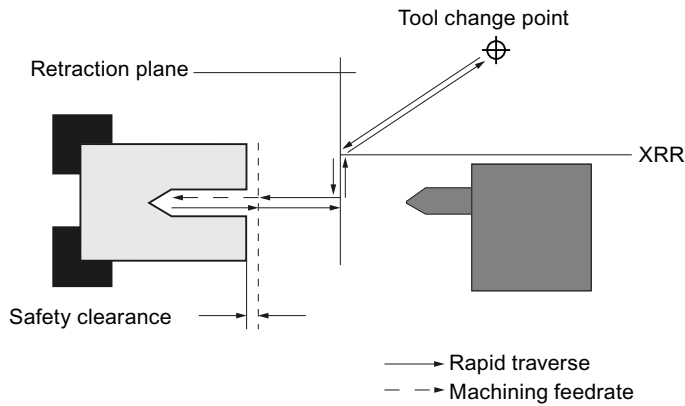


Figure 7-6 Approach/retraction taking into account the tailstock

- The tool traverses in rapid traverse from the tool change point along the shortest path to the retraction plane XRR from the tailstock.
- After this, the tool traverses in rapid traverse on the retraction plane in the X direction.
- After this, the tool traverses in rapid traverse to the safety clearance.
- Following this, the workpiece is then machined with the programmed machining feedrate.
- After machining, the tool retracts with rapid traverse to the safety clearance.
- The tool then continues to traverse vertically in rapid traverse to the retraction plane.
- After this, the tool traverses in the X direction to the retraction plane XRR from the tailstock.
- From there, the tool traverses in rapid traverse along the shortest path to the tool change point. If the tool does not need to be changed between two machining processes, the tool traverses from the retraction plane to the next machining cycle.

You define the tool change point, the retraction plane, the safety clearance, and the retraction plane for the tailstock in the program header.

See also

Programming the approach/retraction cycle (Page 258)

Program header (Page 237)

7.4.3 Absolute and incremental dimensions

When generating a machining step program, you can input positions in absolute or incremental dimensions, depending on how the workpiece drawing is dimensioned.

You can also use a combination of absolute and incremental dimensions, i.e. one coordinate as an absolute dimension and the other as an incremental dimension.

For the face axis (the X axis, in this case), in the machine data it is established whether the diameter or radius is programmed in absolute or incremental dimensions.

Please refer to the machine manufacturer's specifications.

Absolute dimensions (ABS)

With absolute dimensions, all position specifications refer to the zero point of the active coordinate system.

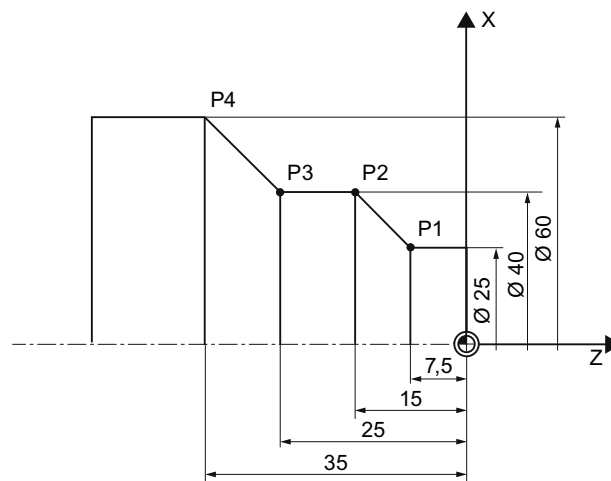


Figure 7-7 Absolute dimensions

The position specifications for the points P1 to P4 in absolute dimensions refer to the zero point:

P1: X25 Z-7.5

P2: X40 Z-15

P3: X40 Z-25

P4: X60 Z-35

Incremental dimensions (INC)

With incremental dimensions (also referred to as sequential dimensions) a position specification refers to the previously programmed point, i.e. the input value corresponds to the path to be traversed. As a rule, the plus/minus sign does not matter when entering the incremental value, only the absolute value of the increment is evaluated. For some parameters, the plus/minus sign specifies the traversing direction. These exceptions are identified in the parameter table of the individual functions.

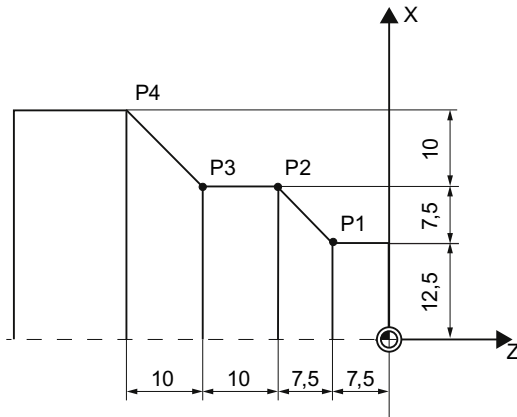


Figure 7-8 Incremental dimensions

The position specifications for points P1 to P4 in incremental dimensions are as follows:

P1: X12.5 Z-7.5 (relative to the zero point)

P2: X7.5 Z-7.5 (relative to P1)

P3: X0 Z-10 (relative to P2)

P4: X10 Z-10 (relative to P3)

7.4.4 Polar coordinates

You can specify positions using right-angled coordinates or polar coordinates.

If a point in a workpiece drawing is defined by a value for each coordinate axis, you can easily input the position into the parameter screen form using right-angled coordinates. For workpieces that are dimensioned with arcs or angular data, it is often easier if you input the positions using polar coordinates.

You can only program polar coordinates for the functions "Straight circle" and "Contour milling."

The point from which dimensioning starts in polar coordinates is called the "pole".

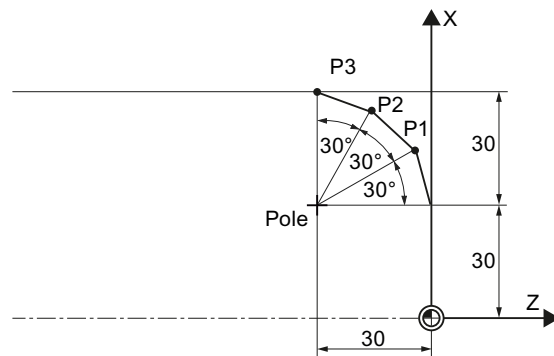


Figure 7-9 Polar coordinates

The position specifications for the pole and points P1 to P3 in polar coordinates are:

Pole: X30 Z30 (relative to the zero point)

P1: L30 $\alpha 30^\circ$ (relative to the pole)

P2: L30 $\alpha 60^\circ$ (relative to the pole)

P3: L30 $\alpha 90^\circ$ (relative to the pole)

7.4.5 Clamping the spindle

The "Clamp spindle" function must be set up by the machine manufacturer.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Note for selecting the clamp spindle function under ShopTurn

The machine manufacturer also specifies whether ShopTurn will clamp the spindle automatically if this would facilitate machining, or if you can decide the types of machining for which the spindle should be clamped.

If you are to decide the types of machining for which the spindle is to be clamped, the following applies:

You should note that when machining in the end face/end face C and peripheral surface/peripheral surface C planes, clamping only remains active for contour milling and drilling operations. On the other hand, when machining in the end face Y/end face B and peripheral surface Y planes, clamping is modal, i.e. it remains active until the machining plane is changed.

7.5 Creating a ShopTurn 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 defined. ShopTurn programs can be created in a new workpiece or under the folder "Part programs".

Procedure

Creating a ShopTurn program



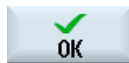
1. Select the "Program Manager" operating area.



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 "ShopTurn" 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 "ShopTurn" program type is selected. The editor is opened and the "Program header" parameter screen is displayed.

Filling out the program header



5. Select a zero offset.

6. Enter the dimensions of the blank and the parameter, which are effective over the complete program, e.g. dimension units in mm or inch, tool axis, retraction plane, safety clearance and machining direction.



Press the "Teach TC position" softkey if you want to set the actual position of the tool as a tool change point.

The tool's coordinates are transferred into parameters XT and ZT.

Teaching in the tool change point is only possible if you have selected the machine coordinate system (Machine).



7. Press the "Accept" softkey.

The process plan is displayed. Program header and end of program are created as program blocks.

The end of program is automatically defined.

The retraction for a cycle ends at the safety clearance. Only the subsequent cycle moves to the retraction plane. This enables a special approach/retraction cycle to be used.

Changes to the retraction plane therefore take effect when retracting from the previous machining operation.

When selecting the traversing paths, the tool tip is always taken into account; i.e. the tool expansion is not considered. Therefore, you should ensure that the retraction planes are an appropriate distance away from the workpiece.

See also

Changing program settings (Page 248)

Programming the approach/retraction cycle (Page 258)









Creating a new workpiece (Page 704)

7.6 Program header

In the program header, set the following parameters, which are effective for the complete program.

Parameters	Description	Unit
Measurement unit U	The setting of the measurement unit in the program header only refers to the position data in the actual program. All other data, such as feedrate or tool offsets, are entered in the unit of measure that you have set for the entire machine.	mm inch
Work offset U	The work offset in which the zero point of the workpiece is saved. You can also delete the default value of the parameter if you do not want to specify a work offset.	
Blank U	Define the form and dimensions of the workpiece:	
	<ul style="list-style-type: none"> Cylinder 	
XA	Outer diameter \varnothing	mm
	<ul style="list-style-type: none"> Polygon 	
N	Number of edges	
SW / L U	Width across flats Edge length	
	<ul style="list-style-type: none"> Centered cuboid 	
W	Width of blank	mm
L	Length of blank	mm
	<ul style="list-style-type: none"> Tube 	
XA	Outer diameter \varnothing	mm
XI U	Inner diameter \varnothing (abs) or wall thickness (inc)	mm
ZA	Initial dimension	mm
ZI U	Final dimension (abs) or final dimension in relation to ZA (inc)	mm
ZB	Machining dimension (abs) or machining dimension in relation to ZA (inc)	mm
Retraction U	The retraction area indicates the area outside of which collision-free traversing of the axes must be possible.	
	<ul style="list-style-type: none"> simple 	
XRA U	Retraction plane X external \varnothing (abs) or retraction plane X referred to XA (inc)	
XRI U	- only for "pipe" blank Retraction plane X internal \varnothing (abs) or retraction plane X referred to XI (inc)	
ZRA U	Retraction plane Z front (abs) or retraction plane Z referred to ZA (inc)	
	<ul style="list-style-type: none"> extended - not for a "pipe" blank 	
XRA U	Retraction plane X external \varnothing (abs) or retraction plane X referred to XA (inc)	
XRI U	Retraction plane X internal \varnothing (abs) or retraction plane X referred to XI (inc)	

7.6 Program header

Parameters	Description	Unit
ZRA 	Retraction plane Z front (abs) or retraction plane Z referred to ZA (inc)	
	<ul style="list-style-type: none"> all 	
XRA 	Retraction plane X external \emptyset (abs) or retraction plane X referred to XA (inc)	
XRI 	Retraction plane X internal \emptyset (abs) or retraction plane X referred to XI (inc)	
ZRA 	Retraction plane Z front (abs) or retraction plane Z referred to ZA (inc)	
ZRI 	Retraction plant Z rear	
Tailstock 	<ul style="list-style-type: none"> Yes No 	
XRR	Retraction plane tailstock – (only "Yes" for tailstock)	
Tool change point 	<p>Tool change point, which must be approached by the revolver with its zero point.</p> <ul style="list-style-type: none"> Work (Workpiece Coordinate System) Machine (Machine Coordinate System) <p>Notes</p> <ul style="list-style-type: none"> The tool change point must be far enough outside the retraction area that it is not possible for any tool to protrude into the retraction area while the revolver is moving. Ensure that the tool change point is relative to the zero point of the revolver and not the tool tip. 	
XT	Tool change point X \emptyset	
ZT	Tool change point Z	
SC	<p>The safety clearance defines how close the tool can approach the workpiece in rapid traverse.</p> <p>Note</p> <p>Enter the safety clearance without sign into the incremental dimension.</p>	
S	<p>Spindle speed (maximum main spindle speed)</p> <p>If you want to machine the workpiece with a constant cutting rate, the spindle speed must be increased as soon as the workpiece diameter becomes smaller. Since the speed cannot be increased at will, you can set a speed limit for the main spindle (S1) and for the counter-spindle (S3), depending on the shape, size, and material of the workpiece or collet.</p> <p>The machine manufacturer only sets one speed limit for the machine, i.e. none that are dependent on the workpiece.</p> <p>Please refer to the machine manufacturer's specifications.</p>	rev/min
Mach. direction of rotation 	<p>Milling direction</p> <ul style="list-style-type: none"> Conventional Climbing 	

7.7 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, correction (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 in the tool list 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.



The "Tool selection" window is opened.

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.

7.8 Tool, offset value, feedrate and spindle speed (T, D, F, S, V)

The following parameters should be entered for every 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 for 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 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.

CAUTION

If, for tools with several cutting edges, you specify the incorrect cutting edge number and move the tool, then collisions can occur. Always ensure that you enter the correct cutting edge number.

Radius compensation

The tool radius compensation is automatically taken into account during all machining cycles, with the exception of path milling and straight.

For path milling and straight lines, you have the option of programming the machining with or without radius compensation. The tool radius compensation is modal for straight lines, i.e. you have to deselect the radius compensation again 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 axes move when machining the workpiece. The machining feedrate is entered in mm/min, mm/rev or in mm/tooth.

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.

For milling and turning cycles, the feedrate during roughing is relative to the milling or cutting center point. This also applies to finishing, with the exception of contours with inner curves. In this case, the feedrate is relative to the contact point between the tool and the workpiece.

The maximum feedrate is determined via machine data.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

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)

The spindle speed S specifies the number of spindle revolutions per minute (rpm) and is programmed along with a tool. The speed specified relates to the main spindle (S1) or counter-spindle (S3) when turning and axial drilling, and to the tool spindle (S2) when drilling and milling.

The spindle starts immediately after the tool change. The spindle stops upon reset, program end, or a tool change. The spindle's direction of rotation is specified in the tool list for each tool.

Cutting rate (V)

The cutting rate V is a circumferential velocity (m/min) and is programmed together with a tool, as an alternative to the spindle speed. The cutting rate is relative to the main spindle (V1) or the counter-spindle (V3) for turning and axial drilling, and corresponds to the circumferential velocity of the workpiece at the point that is currently being machined.

However, for drilling and milling, the cutting rate is relative to the tool spindle (V2) and corresponds to the peripheral speed at which the cutting edge of the tool machines the workpiece.

Converting the spindle speed (S) / cutting rate (V) when milling

As an alternative to the cutting rate, you can also program the spindle speed.

For the milling cycles, the cutting rate (m/min) that is entered is automatically converted into the spindle speed (rpm) using the tool diameter - and vice versa.

Machining

When machining some cycles, you can choose between roughing, finishing, or complete machining. For certain milling cycles, finishing edge or finishing base are possible.

- Roughing
One or more machining operations with depth infeed
- Finishing
Single machining operation
- Edge finishing
Only the edge of the object is finished
- Base finishing
Only the base of the object is finished
- Complete machining
Roughing and finishing with one tool in a
- machining step

If you want to rough and finish using two different tools, you must call the machining cycle twice (1st block = roughing; 2nd block = finishing). The programmed parameters are retained when the cycle is called for the second time.

7.9 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.

7.10 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.

NOTICE

Machining several workpieces

The program repeat function is not suitable to program repeat machining of parts.

In order to repeatedly machine the same workpieces 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.


3. Press the "Set marker" and "Accept" softkeys.
A start marker is inserted behind the actual block.


4. Enter the program blocks that you want to repeat later.



5. Press the "Set marker" and "Accept" softkeys again.
An end marker is inserted after the actual block.



6. Continue programming up to the point where you want to repeat the program blocks.



7. Press the "Various" and "Repeat progr." softkeys.



8. Enter the names of the start and end markers and the number of times the blocks are to be repeated.



9. Press the "Accept" softkey.
The marked program blocks are repeated.

7.11 Entering 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.

If your machine has a bar loader for example, you can program the reloading of the workpiece and then the actual machining at the beginning of the program. At the end, cut off the completed workpiece.

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.

Workpieces can be completely automatically produced in this fashion.

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 172)

7.12 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.

7.13 Changing program settings

Function

All parameters specified in the program header with the exception of the blank shape and the unit of measurement can be changed at any point in the program. It is also possible to change the basic setting for the direction of rotation of machining in the case of milling.

The settings in the program header are modal, i.e. they remain active until they are changed.

Retraction

A changed retraction plane takes effect starting with the safety clearance of the last cycle, because the additional retraction from the next cycle is accepted.

Machining direction

The machining direction of rotation (climbing or conventional) is defined as the direction of movement of the milling tooth with respect to the workpiece, i.e. ShopTurn evaluates this parameter in conjunction with the direction of rotation of the spindle for milling, with the exception of path milling.

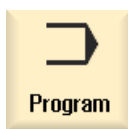
The basic setting for the machining direction is programmed in a machine data.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure










1. Select the "Program" operating area.



2. Press the "Various" and "Settings" softkeys. The "Settings" input window opens.



Parameters

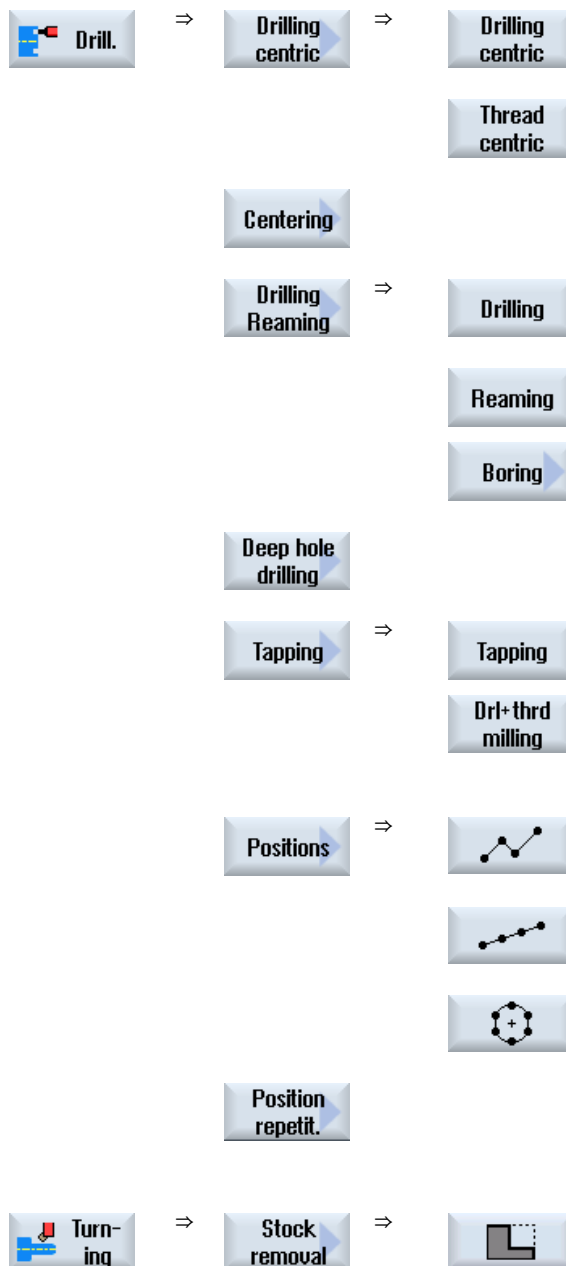
Parameter	Description	Unit
Retraction 	Lift mode <ul style="list-style-type: none"> • simple • Extended • all 	
XRA 	Retraction plane X external \emptyset (abs) or retraction plane X referred to XA (inc)	mm
XRI 	Retraction plane X internal \emptyset (abs) or retraction plane X referred to XI (inc) - (only for retraction "extended" and "all")	mm
ZRA 	Retraction plane Z front (abs) or retraction plane Z referred to ZA (inc)	mm
ZRI	Retraction plane Z rear – (only for retraction "all")	mm
Tailstock 	Yes <ul style="list-style-type: none"> • Tailstock is displayed for simulation / simultaneous recording • When approaching/retracting, the retraction logic is taken into account No	
XRR	Retraction plane – (only "Yes" for tailstock)	mm
Tool change point 	Tool change point <ul style="list-style-type: none"> • Work (Workpiece Coordinate System) • Machine (Machine Coordinate System) 	
XT	Tool change point X	mm
ZT	Tool change point Z	mm
SC	Safety clearance (inc) Acts in relation to the reference point. The direction in which the safety clearance is active is automatically determined by the cycle.	mm
S1	Maximum speed, main spindle	rev/min
Machining direction 	Milling direction: <ul style="list-style-type: none"> • Climbing • Conventional 	

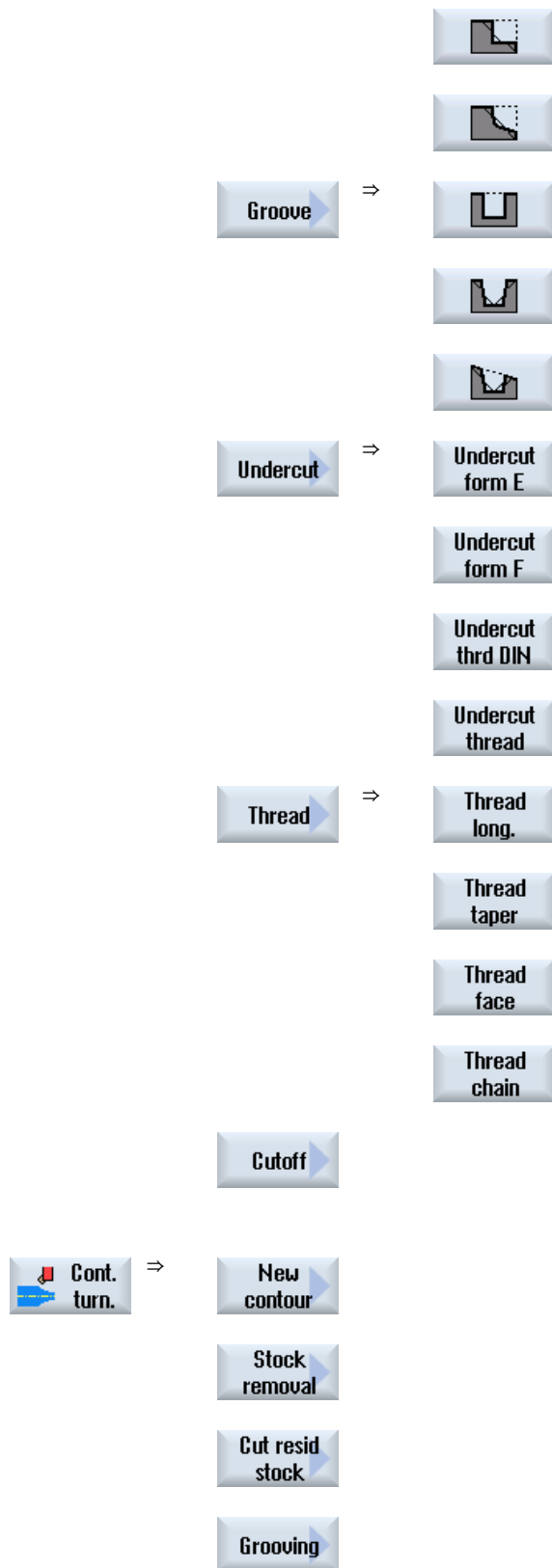
7.14 Selection of the cycles via softkey

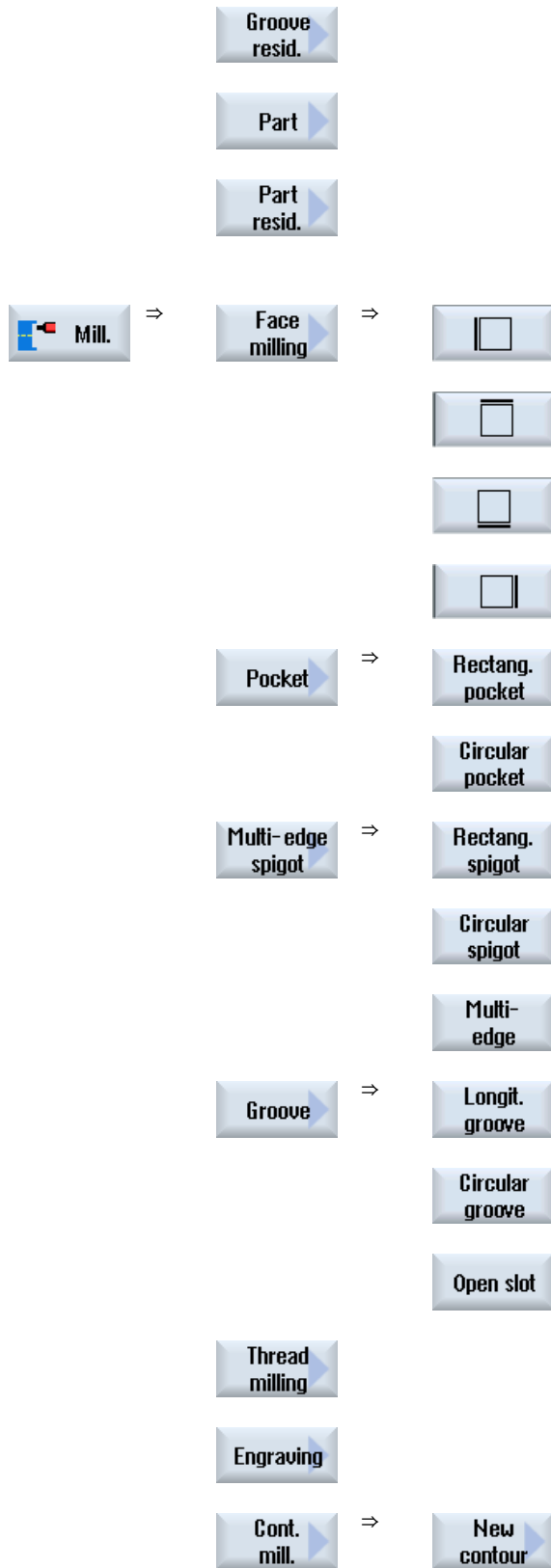
Overview of the machining steps

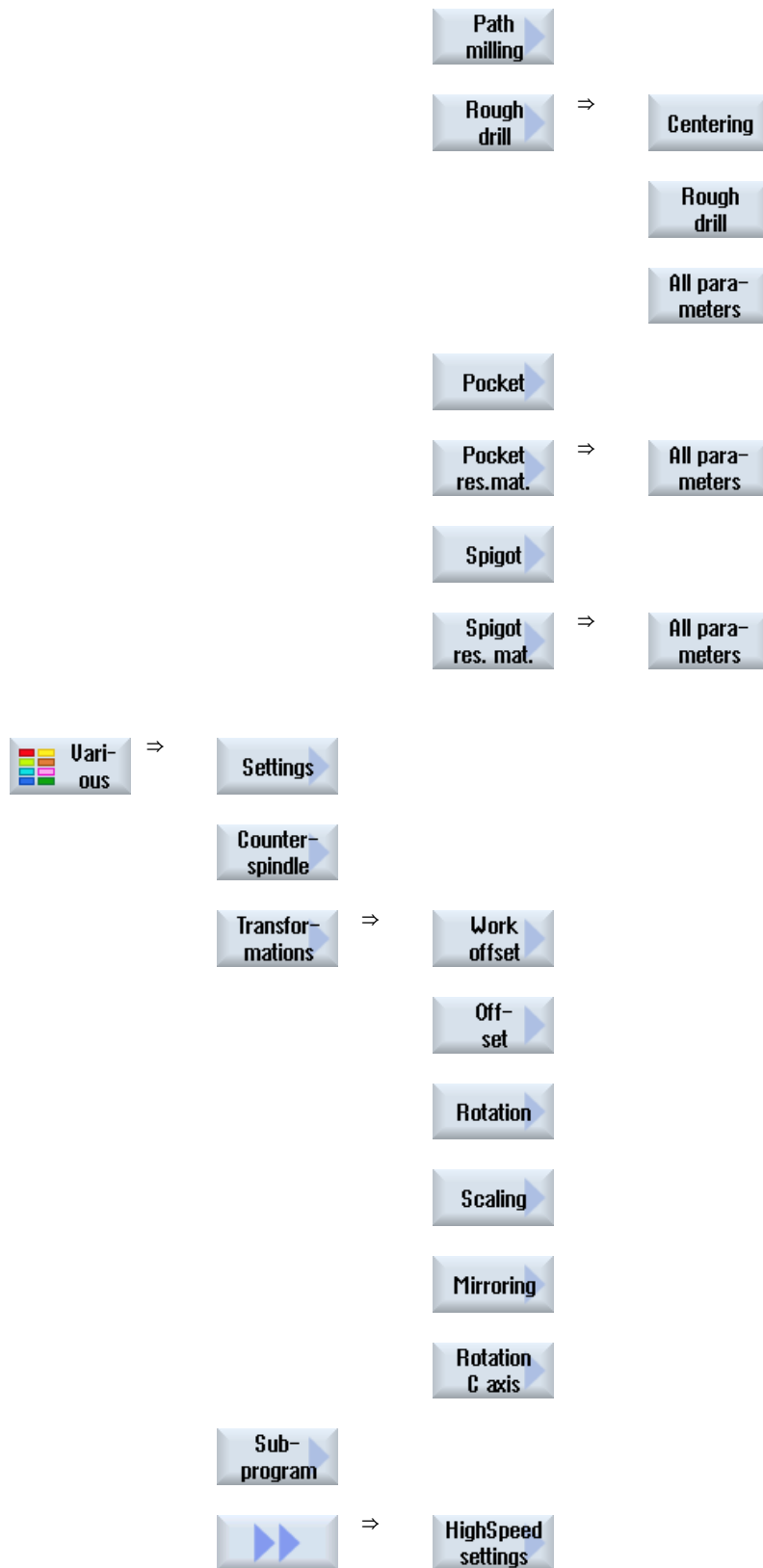
The following machining steps are available.

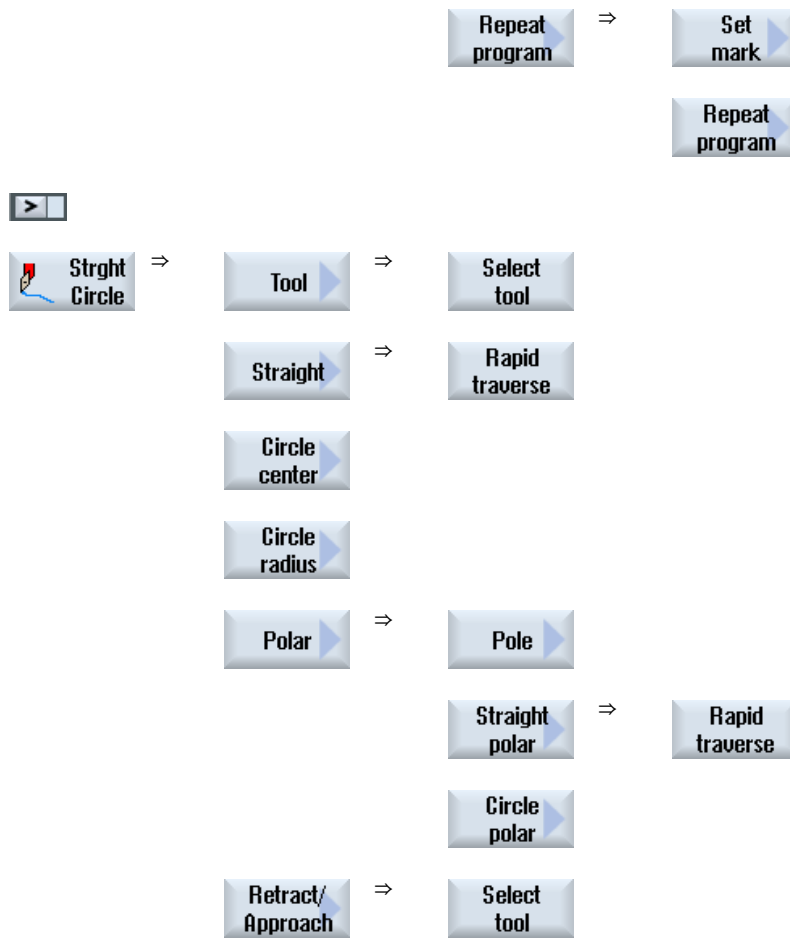
All of the cycles/functions available in the control are shown in this display. However, at a specific system, only the steps possible corresponding to the selected technology can be selected.











⇒ A menu tree with all of the available measuring versions of the measuring cycle function "Measure workpiece" can be found in the following reference:

Programming Manual Measuring cycles / SINUMERIK 840D sl/828D



⇒ A menu tree with all of the available measuring versions of the measuring cycle function "Measure tool" can be found in the following reference:

Programming Manual Measuring cycles / SINUMERIK 840D sl/828D

7.15 Calling technology functions

7.15.1 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.

7.15.2 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 as follows:

- The entry field is displayed with a colored background (orange).
- The comment line displays a note.
- If the parameter entry field is selected with the cursor, the note is also displayed as a 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.

7.15.3 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)

7.15.4 Setting data for technological functions

Technological functions can be influenced and corrected using machine or setting data.

For additional information, please refer to the following documentation:

Commissioning Manual SINUMERIK Operate / SINUMERIK 840D sl

7.15.5 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.

7.15.6 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.

7.16 Programming the approach/retraction cycle

If you wish to shorten the approach/retraction for a machining cycle or solve a complex geometrical situation when approaching/retracting, you can generate a special cycle. In this case, the approach/retraction strategy normally used is not taken into account.

You can insert the approach/retraction cycle between any machining step program blocks, but not within linked program blocks.

Starting point

The starting point for the approach/retraction cycle is the safety clearance approached after the last machining operation.

Tool change

If you want to perform a tool change, you can move the tool through a total of three positions (P1 to P3) to the tool change point and through a maximum of three additional positions (P4 to P6) to the next starting point. If the tool does not need to be changed, however, you have a total of six positions available for the approach to the next starting position.

If three or six positions are not sufficient for the approach/retraction, you can call the cycle several times in succession to program further positions.

 CAUTION

Note that the tool will move from the last position programmed in the approach/retraction cycle directly to the starting point for the next machining operation.
--

See also

Machining cycle, approach/retraction (Page 229)

Procedure










Press the menu forward key and the "Straight Circle" softkey.



Press the "Approach/retract" softkey.

Table 7- 1

Parameters	Description	Unit
F1 	Feedrate to approach the first position Alternatively, rapid traverse	mm/min
X1	1. position \varnothing (abs) or 1st position (inc)	mm (in)
Z1	1. position (abs or inc)	mm
F2 	Feedrate for approach to the second position Alternatively, rapid traverse	mm/min
X2	2. position \varnothing (abs) or 2nd position (inc)	mm
Z2	2. position (abs or inc)	mm
F3 	Feedrate to approach the third position Alternatively, rapid traverse	mm/min
X3	3. position \varnothing (abs) or 3rd position (inc)	mm
Z3	3. position (abs or inc)	mm
Tool change 	TlChngPt: Approach the tool change point from the last programmed position and carry out a tool change direct: Tool is not changed at the tool change position, but at the last programmed position No: Tool is not changed	
T	Tool name - (only for "direct" tool change)	
D	Cutting edge number - (only for "direct" tool change)	
F4 	Feedrate for approach to the fourth position Alternatively, rapid traverse	mm/min
X4	4. position \varnothing (abs) or 4th position (inc)	mm
Z4	4. position (abs or inc)	mm
F5 	Feedrate to approach the fifth position Alternatively, rapid traverse	mm/min
X5	5. position \varnothing (abs) or 5th position (inc)	mm
Z5	5. position (abs or inc)	mm
F6 	Feedrate to approach the sixth position Alternatively, rapid traverse	mm/min
X6	6. position \varnothing (abs) or 6th position (inc)	mm
Z6	6. position (abs or inc)	mm

7.17 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".

References

You will find a more detailed description on how to use measuring cycles in:
Programming Manual Measuring cycles / SINUMERIK 840D sl/828D

7.18 Example: Standard machining

General information

The following example is described in detail as ShopTurn 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:

Roughing tool_80	80°, R0.6
Roughing tool_55	55°, R0.4
Finishing tool	35°, R0.4
Plunge cutter	Plate width 4
Threading tool_2	
Drill_D5	Ø5
Miller_D8	Ø8

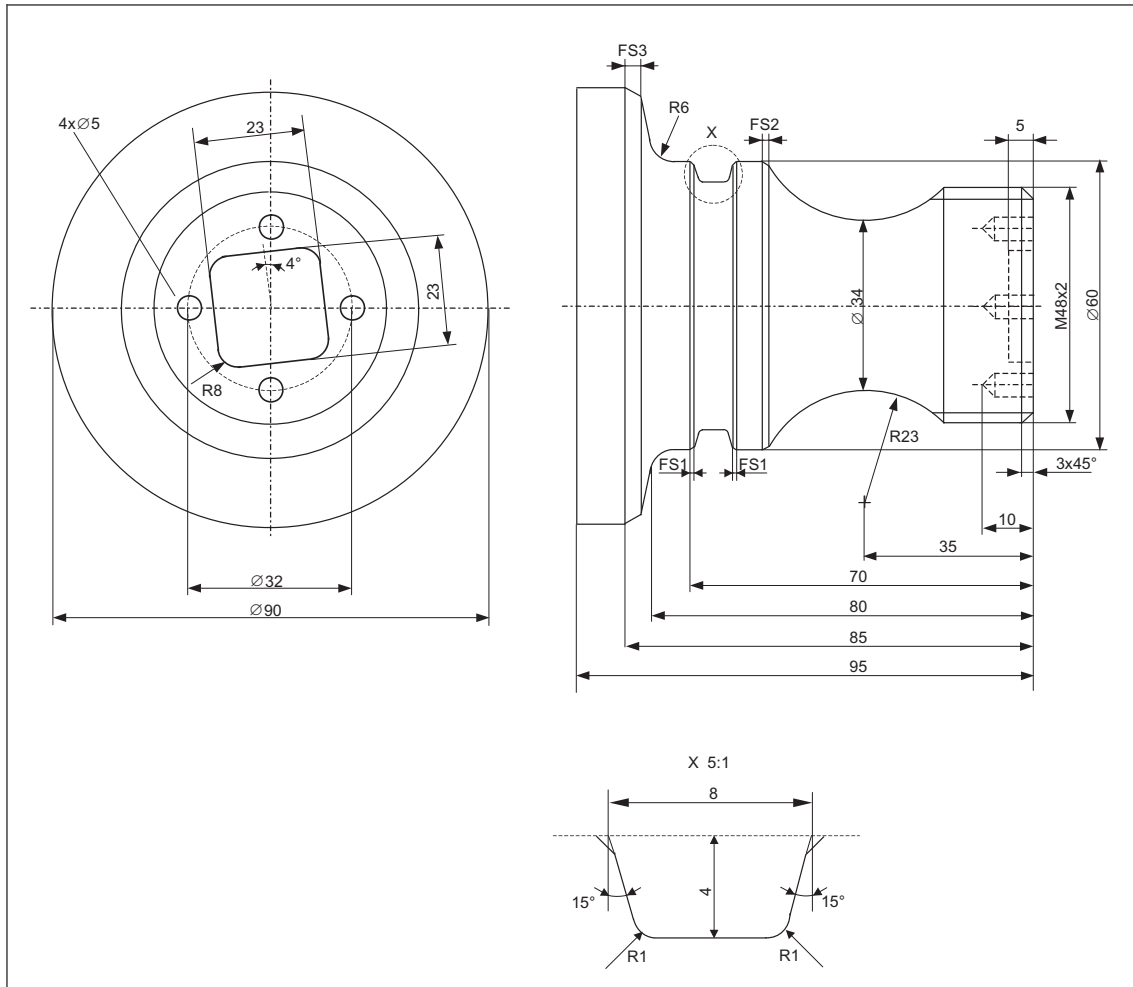
Adapt the cutting data to the tools used and the specific application conditions at the machine.

Blank

Dimensions: Ø90 x 120

Material: Aluminum

7.18.1 Workpiece drawing



7.18.2 Programming

1. Program header

1. Specify the blank.

Measurement unit mm

Blank	Cylinder
XA	90abs
ZA	+1.0abs
ZI	-120abs
ZB	-100abs
Retraction	simple
XRA	2inc
ZRA	5inc
Tool change point	Machine
XT	160abs
ZT	409abs
SC	1
S1	4000rev/min
Machining direction	Climbing



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. Stock removal cycle for facing



1. Press the "Turning" and "Stock removal" softkeys.

2. Select the machining strategy.

3. Enter the following technology parameters:

T Roughing **D1** **F** 0.300 mm/rev **V** 350 m/min
tool_80

4. Enter the following parameters:

Machining Roughing (▽)

Position 

Direction Face (parallel to the X axis)

X0 90abs

Z0 2abs

X1 -1.6abs

Z1 0abs

D 2inc

UX 0inc

UZ 0.1inc



5. Press the "Accept" softkey.

3. Input of blank contour with contour computer



1. Press the "Cont. turn." and "New contour" softkeys.
The "New Contour" input window opens.



2. Enter the contour name (in this case: Cont_1).
The contour calculated as NC code is written as internal subprogram between a start and an end marker containing the entered name.



3. Press the "Accept" softkey.
The "Starting point" entry field opens.
4. Enter the starting point of the contour.
X 60abs Z 0abs



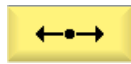
5. Press the "Accept" softkey.
6. Enter the following contour elements and acknowledge using the "Accept" softkey.



- 6.1 Z -40abs



- 6.2 X 80abs Z -45abs



- 6.3 Z -65abs



- 6.4 X 90abs Z -70abs



- 6.5 Z -95abs



- 6.6 X 0abs



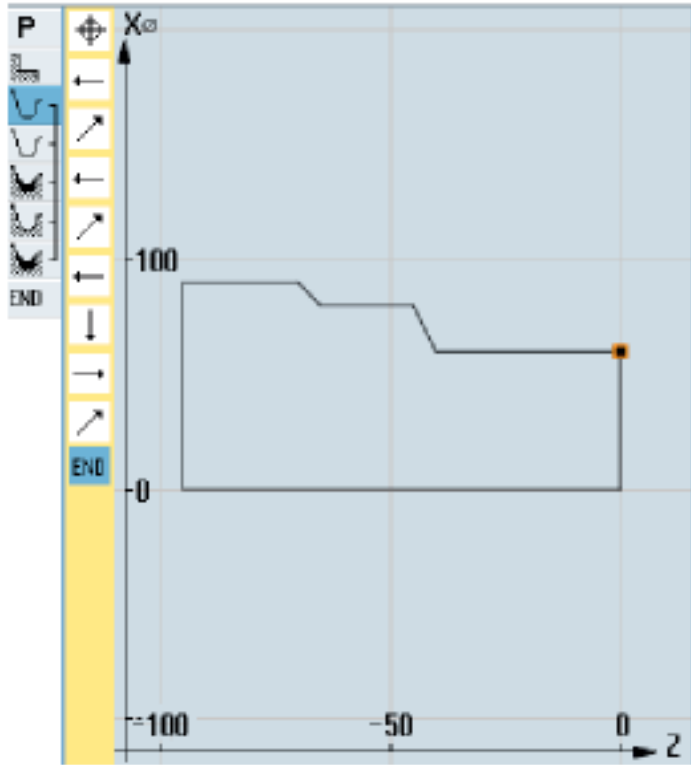
- 6.7 Z 0abs



- 6.8 X 60abs Z 0abs



7. Press the "Accept" softkey.
It is only necessary to enter the blank contour when using a pre-machined blank.



Blank contour

4. Input of finished part with contour computer



1. Press the "Cont. turn." and "New contour" softkeys.
The "New Contour" input window opens.

2. Enter the contour name (in this case: Cont_2).
The contour calculated as NC code is written as internal subprogram between a start and an end marker containing the entered name.

3. Press the "Accept" softkey.
The "Starting point" entry field opens.

4. Specify the contour starting point of the contour.
X 0abs Z 0abs

5. Press the "Accept" softkey.

6. Enter the following contour elements and acknowledge using the "Accept" softkey.

- 6.1 X 48abs FS 3

- 6.2 α2 90°

- 6.3 Direction of rotation 2

R 23abs X 60abs K -35abs I 80 abs
Afterwards, entry fields are inactive.

- 6.4 Using the "Dialog selection" softkey, select a required contour element and confirm using the "Dialog accept" softkey. The entry fields are active again. Enter the additional parameters.

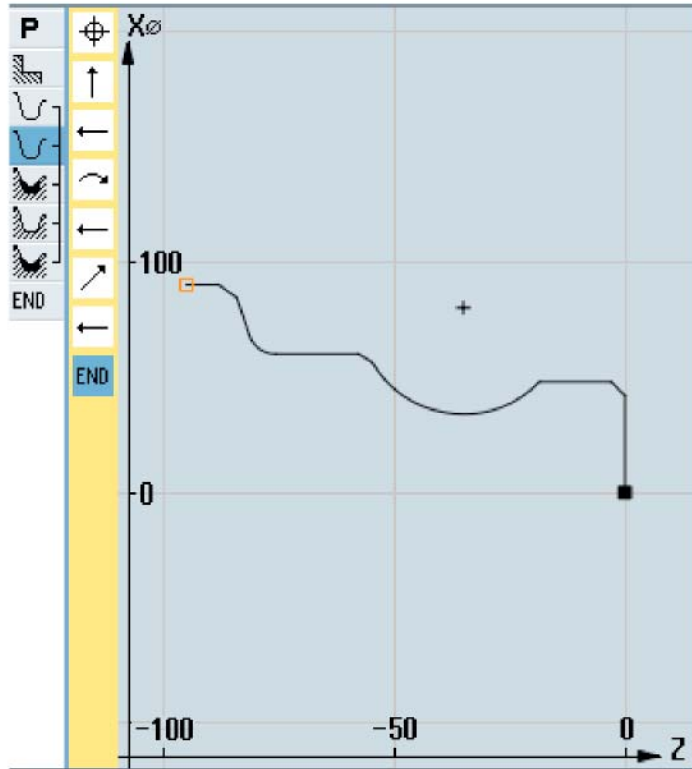
FS 2
6.5 Z -80abs R 6

- 6.6 X 90abs Z -85abs FS3

- 6.7 Z -95abs



7. Press the "Accept" softkey.



Finished-part contour

5. Stock removal (roughing)



1. Press the "Cont. turn." and "Stock removal" softkeys.
The "Stock Removal" input window opens.

2. Enter the following technology parameters:
T Roughing tool 80 D1 F 0.350 mm/rev V 400 m/min

3. Enter the following parameters:

Machining Roughing (▽)

Machining direction Longitudinal

Position external

Machining direction

(from front to back)

D 4.000inc

Cutting depth

UX 0.4inc

UZ 0.2inc

DI 0

BL Cylinder

XD 0inc

ZD 0inc

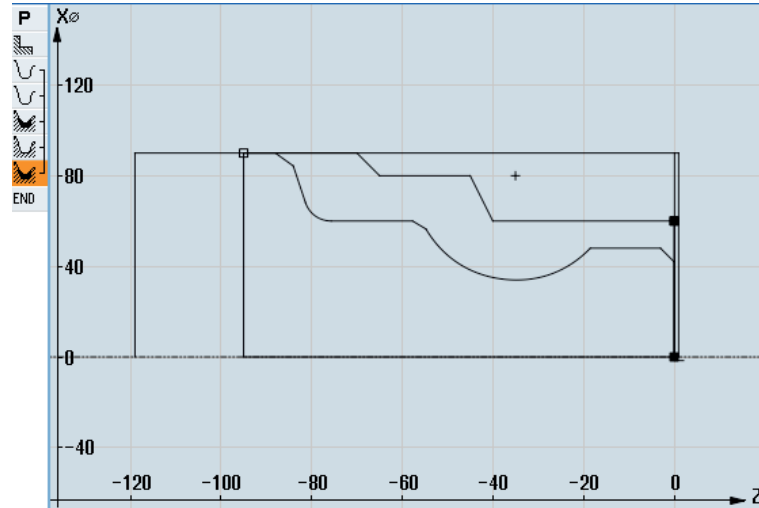
Relief cuts No

Set machining area limits No



4. Press the "Accept" softkey.

If a blank programmed under "CONT_1" is used, under parameter "BL", the "Contour" blank description should be selected instead of "Cylinder". When selecting "Cylinder", the workpiece is cut from the solid material.



Stock removal contour

6. Solid machine residual material



1. Press the "Cont. turn." and "St. remov. resid." softkeys.
The "Stock removal residual material" input window opens.

2. Enter the following technology parameters:
T Roughing tool_55 D1 F 0.35 mm/rev V 400 m/min

3. Enter the following parameters:
Machining Roughing (▽)
Machining direction Longitudinal
Position external
Machining direction ←

D 2inc

Cutting depth ⇔

UX 0.4inc

UZ 0.2inc

DI 0

Relief cuts Yes

FR 0.200mm/rev

Set machining area limits No



4. Press the "Accept" softkey.

7. Stock removal (finishing)



1. Press the "Cont. turn." and "Stock removal" softkeys.
The "Stock Removal" input window opens.

2. Enter the following technology parameters:
T Finishing tool_D1 F 0.1 mm/rev V 450 m/min

3. Enter the following parameters:
Machining Finishing (▽▽▽)

Machining direction Longitudinal

Position external

Machining direction 

(from front to back)

Allowance No

Relief cuts Yes

Set machining area limits No



4. Press the "Accept" softkey.

8. Groove (roughing)



1. Press the "Turning", "Groove" and "Groove with inclines" softkeys.
The "Groove 1" entry field opens.

2. Enter the following technology parameters:
T Grooving **D1** **F 0.150 mm/rev** **V 220 m/min**
tool

3. Enter the following parameters:

Machining Roughing (▽)

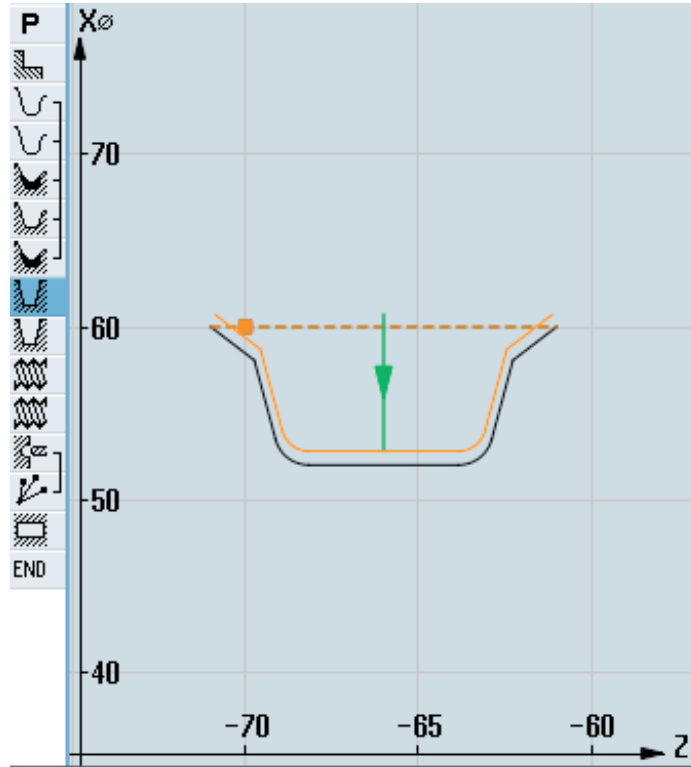
Groove position 

Reference point 

X0	60abs
Z0	-70
B2	8inc
T1	4inc
α1	15Degrees
α2	15Degrees
FS1	1
R2	1
R3	1
FS4	1
D	2inc
UX	0.4inc
UZ	0.2inc
N	1



- 4. Press the "Accept" softkey.



Contour, groove

10. Longitudinal threads M48 x2 (roughing)



1. Press the "Turning", "Thread" and "Thread longitudinal" softkeys. The "Longitudinal thread" entry field opens.

2. Enter the following parameters:

T	Threading tool_2	D1
Table	without	
P	2mm/rev	
G	0	
S	995rev/min	
Machining type	Roughing (∇)	
Infeed: Constant cutting cross-section	Diminishing	
Thread	External thread	
X0	48abs	
Z0	0abs	
Z1	-25abs	
LW	4inc	
LR	4inc	
H1	1.227inc	
αP	30Degrees	
Infeed		
ND	5	
U	0.150inc	
VR	1inc	
Multiple threads	No	
α0	0Degrees	



3. Press the "Accept" softkey.

11. Longitudinal threads M48 x 2 (finishing)



1. Press the "Turning", "Thread" and "Thread longitudinal" softkeys. The "Longitudinal thread" entry field opens.

2. Enter the following parameters:

T	Threading tool_2	D1
Table	without	
P	2mm/rev	
G	0	
S	995rev/min	
Machining type	Finishing (▽▽▽)	
Thread	External thread	
X0	48abs	
Z0	0abs	
Z1	-25abs	
LW	4inc	
LR	4inc	
H1	1.227inc	
αP	30Degrees	
Infeed		
NN	2	
VR	1inc	
Multiple threads	No	
α0	0Degrees	



3. Press the "Accept" softkey.

12. Drilling



1. Press the "Drilling", "Drilling reaming" and "Drilling" softkeys. The "Drilling" input window opens.

2. Enter the following technology parameters:
T Drill_D5 D1 F 0.1 mm/rev V 50 m/min

3. Enter the following parameters:
Machined surface Face C
Drilling depth Tip
Z1 10inc
DT 0s



4. Press the "Accept" softkey.

13. Positioning



1. Press the "Drilling", "Positions" and "Freely Programmable Positions" softkeys. The "Positions" input window opens.

2. Enter the following parameters:
Machined surface Face C
Coordinate system Polar
Z0 0abs
C0 0abs
L0 16abs
C1 90abs
L1 16abs
C2 180abs
L2 16abs
C3 270abs
L3 16abs



3. Press the "Accept" softkey.

14. Milling the rectangular pocket



1. Press the "Milling", "Pocket" and "Rectangular pocket" softkeys.
The "Rectangular Pocket" input window opens.

2. Enter the following technology parameters:
T Miller_D8 **D1** **F** 0.030 mm/tooth **V** 200 m/min

3. Enter the following parameters:

Machined surface	Face C
Machining type	Roughing (∇)
Machining position	Single position
X0	0abs
Y0	0abs
Z0	0abs
W	23
L	23
R	8
α0	4Degrees
Z1	5inc
DXY	50%
DZ	3
UXY	0.1mm
UZ	0
Insertion	Vertical
FZ	0.015mm/tooth



4. Press the "Accept" softkey.

7.18.3 Results/simulation test

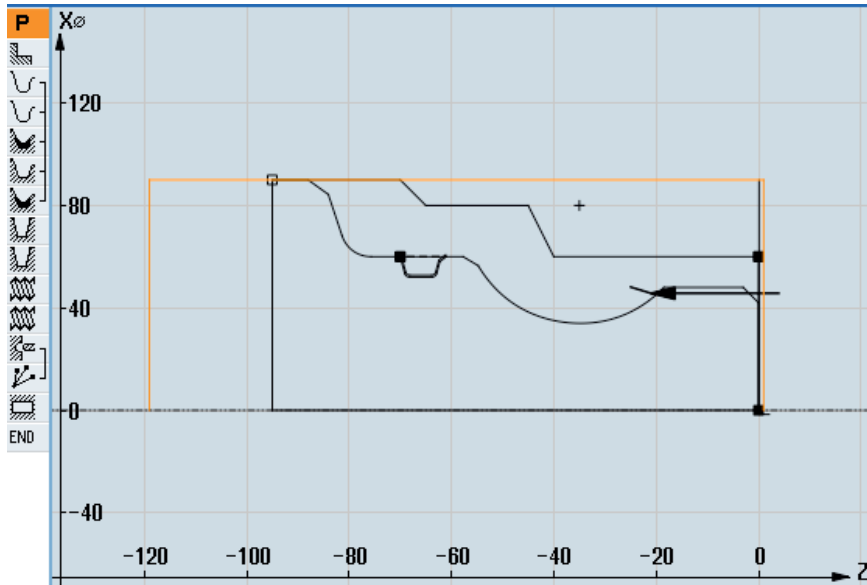


Figure 7-10 Programming graphics

P	Program header	Work offset G54
	Stock removal	T=Schrupper_80 F0.3/rev U=350m Face X0=90
	Contour	KONT_1
	Contour	KONT_2
	Stock removal	T=Schrupper 80 F0.35/rev U=400m
	Residual cutting	T=Schrupper_55 F0.35/rev U=400m
	Stock removal	T=Schlichter F0.1/rev U=450m
	Groove	T=Stecher F0.15/rev U=220m X0=60 Z0=-70
	Groove	T=Stecher F0.1/rev U=220m X0=60 Z0=-70
	Thread long.	T=Gewindestahl_2 P2mm/rev S=995rev Outside
	Thread long.	T=Gewindestahl_2 P2mm/rev S=995rev Outside
	Drilling	⊕ T=Bohrer_D5 F0.1/min U=50m Z1=10inc
	001: Positions	⊕ Z0=0 C0=0 L0=16 C1=90 L1=16 C2=180 L2=16
	Rectang.pocket	⊕ T=Fräser_D8 F0.03/min U=200m X0=0 Y0=0
END	End of program	

Figure 7-11 Process plan

Program test by means of simulation

During simulation, the current program is calculated in its entirety and the result displayed in graphic form.

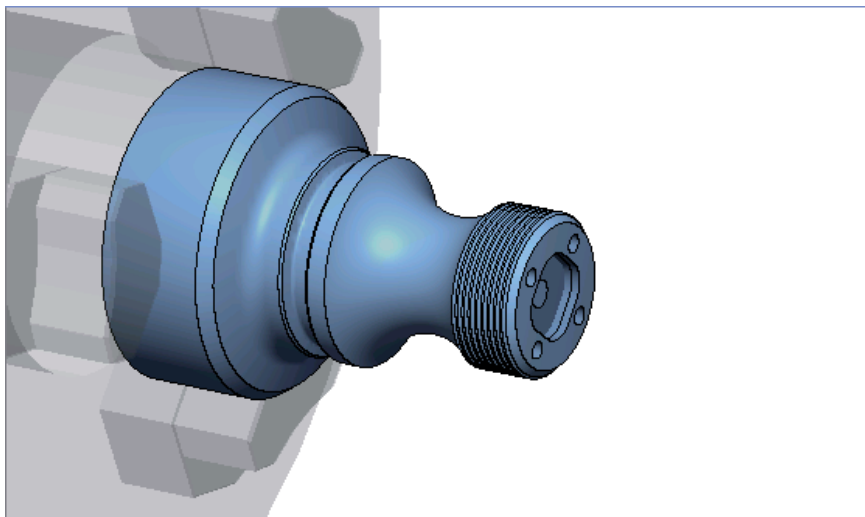


Figure 7-12 3D view

7.18.4 G code machining program

```

N1 G54
N2 WORKPIECE(,,"", "CYLINDER",192,2,-120,-100,90)
N3 G0 X200 Z200 Y0
;*****
N4 T="ROUGHING TOOL_80" D1
N5 M06
N6 G96 S350 M04
N7 CYCLE951(90,2,-1.6,0,-1.6,0,1,2,0,0.1,12,0,0,0,1,0.3,0,2,1110000)
N8 G96 S400
N9 CYCLE62(,2,"E_LAB_A_CONT_2","E_LAB_E_CONT_2")
N10 CYCLE952("STOCK
REMOVAL_1",,"BLANK_1",2301311,0.35,0.15,0,4,0.1,0.1,0.4,0.2,0.1,0,1,0,0,,,,,2,2,,,0,1,,0,12,1110110)
N11 G0 X200 Z200
;*****
N12 T="ROUGHING TOOL_55" D1
N13 M06
N14 G96 S400 M04
N15 CYCLE952("STOCK
REMOVAL_2",,"BLANK_1", "Blank_1",1301311,0.35,0.2,0,2,0.1,0.1,0.4,0.2,0.1,0,1,0,,,,,2,2,,,0,1,,0,112,1
100110)
N16 G0 X200 Z200
;*****
N17 T="FINISHING TOOL" D1
N18 M06
N19 G96 S450 M04
N20 CYCLE952("STOCK
REMOVAL_3",,"",1301321,0.1,0.5,0,1,9,0.1,0.1,0.2,0.1,0.1,0,1,0,0,,,,,2,2,,,0,1,,0,12,1000110)
N21 G0 X200 Z200
;*****
N22 T="GROOVING TOOL" D1
N23 M06
N24 G96 S220 M04
N25 CYCLE930(60,-70,5.856406,8,4,,0,15,15,1,1,1,1,0.2,2,1,10110,,1,30,0.15,1,0.4,0.2,2,1001010)
N26 CYCLE930(60,-70,5.856406,8,4,,0,15,15,1,1,1,1,0.2,2,1,10120,,1,30,0.1,1,0.1,0.1,2,1001110)
N27 G0 X200 Z200
;*****
N28 T="THREADING TOOL_2" D1
N29 M06
N30 G97 S995 M03
N31 CYCLE99(0,48,-25,,4,4,1.226,0.1,30,0,5,0,2,1100103,4,1,0.2815,0.5,0,0,1,0,0.707831,1,,,,,2,0)
N32 CYCLE99(0,48,-25,,4,4,1.226,0.02,30,0,3,2,2,1210103,4,1,0.5,0.5,0,0,1,0,0.707831,1,,,,,2,0)
N33 G0 X200 Z200

```

```
;*****  
N34 T="DRILL_D5" D1  
N35 M06  
N36 SPOS=0  
N37 SETMS(2)  
N38 M24 ; couple-in driven tool, machine-specific  
N39 G97 S3183 M3  
N40 G94 F318  
N41 TRANSMIT  
N42 MCALL CYCLE82(1,0,1,,10,0,0,1,11)  
N43 HOLES2(0,0,16,0,30,4,1010,0,,1)  
N44 MCALL  
N45 M25 ; couple out driven tool, machine-specific  
N46 SETMS(1)  
N47 TRAFOOF  
N48 G0 X200 Z200  
;*****  
N49 T="MILLER_D8"  
N50 M6  
N51 SPOS=0  
N52 SETMS(2)  
N53 M24  
N54 G97 S1989 M03  
N55 G95 FZ=0.15  
N56 TRANSMIT  
N57 POCKET3(20,0,1,5,23,23,8,0,0,4,3,0,0,0.12,0.08,0,11,50,8,3,15,0,2,0,1,2,11100,11,111)  
N58 M25  
N59 TRAFOOF  
N60 DIAMON  
N61 SETMS(1)  
N62 G0 X200 Z200  
N63 M30  
;*****  
N64 E_LAB_A_CONT_1: ;#SM Z:3  
;#7__DlgK contour definition begin - Don't change!;*GP*;*RO*;*HD*  
G18 G90 DIAMOF;*GP*  
G0 Z0 X30 ;*GP*  
G1 Z-40 ;*GP*  
Z-45 X40 ;*GP*  
Z-65 ;*GP*  
Z-70 X45 ;*GP*  
Z-95 ;*GP*  
X0 ;*GP*  
Z0 ;*GP*
```

7.18 Example: Standard machining

```

X30 ;*GP*
;CON,2,0.0000,1,1,MST:0,0,AX:Z,X,K,I;*GP*;*RO*;*HD*
;S,EX:0,EY:30;*GP*;*RO*;*HD*
;LL,EX:-40;*GP*;*RO*;*HD*
;LA,EX:-45,EY:40;*GP*;*RO*;*HD*
;LL,EX:-65;*GP*;*RO*;*HD*
;LA,EX:-70,EY:45;*GP*;*RO*;*HD*
;LL,EX:-95;*GP*;*RO*;*HD*
;LD,EY:0;*GP*;*RO*;*HD*
;LR,EX:0;*GP*;*RO*;*HD*
;LA,EX:0,EY:30;*GP*;*RO*;*HD*
;#End contour definition end - Don't change!;*GP*;*RO*;*HD*
E_LAB_E_CONT_1:
N65 E_LAB_A_CONT_2: ;#SM Z:4
;#7__DlGK contour definition begin - Don't change!;*GP*;*RO*;*HD*
G18 G90 DIAMOF;*GP*
G0 Z0 X0 ;*GP*
G1 X24 CHR=3 ;*GP*
Z-18.477 ;*GP*
G2 Z-55.712 X30 K=AC(-35) I=AC(40) ;*GP*
G1 Z-80 RND=6 ;*GP*
Z-85 X45 CHR=3 ;*GP*
Z-95 ;*GP*
;CON,V64,2,0.0000,0,0,MST:0,0,AX:Z,X,K,I;*GP*;*RO*;*HD*
;S,EX:0,EY:0,ASE:90;*GP*;*RO*;*HD*
;LU,EY:24;*GP*;*RO*;*HD*
;F,LFASE:3;*GP*;*RO*;*HD*
;LL,DIA:225/0,AT:90;*GP*;*RO*;*HD*
;ACW,DIA:210/0,EY:30,CX:-35,CY:40,RAD:23;*GP*;*RO*;*HD*
;LL,EX:-80;*GP*;*RO*;*HD*
;R,RROUND:6;*GP*;*RO*;*HD*
;LA,EX:-85,EY:45;*GP*;*RO*;*HD*
;F,LFASE:3;*GP*;*RO*;*HD*
;LL,EX:-95;*GP*;*RO*;*HD*
;#End contour definition end - Don't change!;*GP*;*RO*;*HD*
E_LAB_E_CONT_2:

```


Programming technology functions (cycles)

8.1 Drilling

8.1.1 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.

8.1 Drilling

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

See also

Selection of the cycles via softkey (Page 211)

8.1.2 Centering (CYCLE81)

8.1.2.1 Function

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.

Clamping the spindle

For ShopTurn, "Clamp spindle" function can be set up by the machine manufacturer.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

See also

Clamping the spindle (Page 234)

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 ShopTurn program to be processed has been created and you are in the editor.











2. Press the "Drilling" softkey.



3. Press the "Centering" softkey.
The "Centering" input window opens.

Parameters, G code program			Parameters, ShopTurn 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

8.1 Drilling

Parameter	Description	Unit
Machining position  (only for G code)	<ul style="list-style-type: none"> • Single position Drill hole at programmed position • Position pattern Position with MCALL 	
Z0 (only for G code)	Reference point Z	mm
Machining surface  (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y • Peripheral surface C • Peripheral surface Y 	
Position  (only for ShopTurn)	<ul style="list-style-type: none"> • At the front (face) • At the rear (face) • Outside (peripheral surface) • Inside (peripheral surface) 	
  (only for ShopTurn)	Clamp/release spindle The function must be set up by the machine manufacturer.	
Centering 	<ul style="list-style-type: none"> • 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 	<ul style="list-style-type: none"> • Dwell time (at final drilling depth) in seconds • Dwell time (at final drilling depth) in revolutions 	s rev

8.1.3 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.

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.

See also

Clamping the spindle (Page 234)

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 ShopTurn 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.

8.1 Drilling

Parameters, G code program			Parameters, ShopTurn 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)	<ul style="list-style-type: none"> • Single position Drill hole at programmed position • Position pattern Position with MCALL 	
Z0 (only for G code)	Reference point Z	mm
Machining surface  (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y • Peripheral surface C • Peripheral surface Y 	
Position  (only for ShopTurn)	<ul style="list-style-type: none"> • At the front (face) • At the rear (face) • Outside (peripheral surface) • Inside (peripheral surface) 	
  (only for ShopTurn)	Clamp/release spindle The function must be set up by the machine manufacturer.	
Drilling depth 	<ul style="list-style-type: none"> • 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. <p>Note: If it is not possible to define an angle for the drill in the tool management, it will not be possible to select tip or shank (always tip, 0 field)</p>	

Parameter	Description	Unit
Z1 	Drilling depth (abs) or drilling depth in relation to Z0 (inc) It is inserted into the workpiece until it reaches Z1.	mm
DT 	<ul style="list-style-type: none"> • Dwell time (at final drilling depth) in seconds • Dwell time (at final drilling depth) in revolutions 	s rev

8.1.4 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.

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.

See also

Clamping the spindle (Page 234)

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.

8.1 Drilling




Procedure



1. The part program or ShopTurn 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, ShopTurn 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)	<ul style="list-style-type: none"> • 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 ShopTurn)	Feedrate during retraction	mm/min mm/rev
Machining surface (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y • Peripheral surface C • Peripheral surface Y 	
Position (only for ShopTurn)	<ul style="list-style-type: none"> • At the front (face) • At the rear (face) • Outside (peripheral surface) • Inside (peripheral surface) 	

Parameter	Description	Unit
  (only for ShopTurn)	Clamp/release spindle The function must be set up by the machine manufacturer.	
Z1	Drilling depth (abs) or drilling depth in relation to Z0 (inc) It is inserted into the workpiece until it reaches Z1.	mm
DT 	<ul style="list-style-type: none"> • 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

8.1.5 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.

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.

Lift

For "lift off contour", the retraction distance D and the tool orientation angle α can either be defined via machine data or in the parameter screen. If both parameters are pre-assigned via machine data, they do not appear in the parameter screen.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

NOTICE

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.

See also

Clamping the spindle (Page 234)

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 ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Drilling" softkey.
3. Press the softkey "Boring" for G code.

- OR -



3. Press the softkeys "Drilling Reaming" and "Boring" for ShopTurn








The "Boring" input window opens.

Parameters, G code program			Parameters, ShopTurn 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)	<ul style="list-style-type: none"> Single position Drill hole at programmed position. Position pattern Position with MCALL 	
DIR (only for G code)	Direction of rotation <ul style="list-style-type: none"> 	
Z0 (only for G code)	Reference point Z	mm
Machining surface (only for ShopTurn)	<ul style="list-style-type: none"> Face C Face Y Peripheral surface C Peripheral surface Y 	
Position (only for ShopTurn)	<ul style="list-style-type: none"> At the front (face) At the rear (face) Outside (peripheral surface) Inside (peripheral surface) 	

8.1 Drilling

Parameter	Description	Unit
  (only for ShopTurn)	Clamp/release spindle The function must be set up by the machine manufacturer.	
Z1 	Drilling depth (abs) or drilling depth in relation to Z0 (inc)	mm
DT 	<ul style="list-style-type: none"> • Dwell time at final drilling depth in seconds • Dwell time at final drilling depth in revolutions 	s rev
SPOS	Spindle stop position	Degrees
Lift mode 	<ul style="list-style-type: none"> • Do not lift off contour The cutting edge is not fully retracted, but traverses back to the safety clearance in rapid traverse. <ul style="list-style-type: none"> • Lift The cutting edge is retracted from the hole edge and then moved back to the retraction plane.	
DX (only G Code)	Retraction distance in the X direction (incremental) - (for lift-off only)	mm
DY (only G code)	Retraction distance in the Y direction (incremental) - (for lift-off only)	mm
DZ (only G code)	Retraction distance in the Z direction (incremental) - (for lift-off only)	mm
D (only ShopTurn)	Retraction distance (incremental) - (for lift only)	mm

8.1.6 Deep-hole drilling (CYCLE83)

Function

With the "Deep-hole drilling" 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. The following can be specified:

- Number of infeed steps constant or decreasing (via programmable degression factor)
- Chip breaking without lifting or swarth 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

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.

See also

Clamping the spindle (Page 234)

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 $F = F \cdot FD1$ [%] 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 $F = F \cdot FD1$ [%] 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.

8.1 Drilling







Procedure






1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Drilling" softkey.
3. Press the "Deep-hole drilling" softkey.
The "Deep-hole Drilling" input window opens.

Parameters, G code program			Parameters, ShopTurn 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)	<ul style="list-style-type: none"> • Single position Drill hole at programmed position • Position pattern Position with MCALL 	
Z0 (only for G code)	Reference point Z	mm
Machining surface (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y • Peripheral surface C • Peripheral surface Y 	
Position (only for ShopTurn)	<ul style="list-style-type: none"> • At the front (face) • At the rear (face) • Outside (peripheral surface) • Inside (peripheral surface) 	
 (only for ShopTurn)	Clamp/release spindle The function must be set up by the machine manufacturer.	

Parameter	Description	Unit
Machining 	<ul style="list-style-type: none"> Chip removal The drill is retracted from the workpiece for stock removal. Chipbreaking The drill is retracted by the retraction distance V2 for chipbreaking. 	
Drilling depth 	<ul style="list-style-type: none"> 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. <p>Note: If it is not possible to define an angle for the drill in the tool management, it will not be possible to select tip or shank (always tip, 0 field)</p>	
Z1 	Drilling depth (abs) or drilling depth in relation to Z0 (inc) It is inserted into the workpiece until it reaches Z1.	mm
D  (only for G code)	1. Drilling depth (abs) or 1st drilling depth in relation to Z0 (inc)	mm
D (only for ShopTurn)	Maximum depth infeed	mm
FD1	Percentage for the feedrate for the first infeed	%
DF 	<p>Infeed:</p> <ul style="list-style-type: none"> Amount for each additional infeed Percentage for each additional infeed <p>DF = 100%: Infeed increment remains constant DF < 100%: Infeed increment is reduced in direction of final drilling depth.</p> <p>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.</p>	mm %
V1	<p>Minimum infeed - (only for DF in %)</p> <p>Parameter V1 is only available if DF < 100 has been programmed.</p> <p>If the infeed increment becomes very small, a minimum infeed can be programmed in parameter "V1".</p> <p>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.</p>	mm
V2	<p>Retraction distance after each machining step – (for chipbreaking only)</p> <p>Distance by which the drill is retracted for chipbreaking.</p> <p>V2 = 0: The tool is not retracted but is left in place for one revolution.</p>	mm
Limit distance (for stock removal only) 	<ul style="list-style-type: none"> Manual The limit distance must be entered manually. Automatic The limit distance is calculated by the cycle. 	

8.1 Drilling

Parameter	Description	Unit
V3	Limit distance – (for stock removal only and manual limit distance) Distance to the last infeed depth that the drill approaches in rapid traverse after stock removal.	mm
DTB (only for G code) 	<ul style="list-style-type: none"> Dwell time at drilling depth in seconds Dwell time at drilling depth in revolutions 	s rev
DT 	<ul style="list-style-type: none"> Dwell time at final drilling depth in seconds Dwell time at final drilling depth in revolutions 	s rev
DTS (only for G code) 	<ul style="list-style-type: none"> Dwell time for stock removal in seconds Dwell time for stock removal in revolutions 	s rev

8.1.7 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, chipbreaking or retraction from the workpiece for stock 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 if the spindle is equipped with a measuring system.

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.

See also

Clamping the spindle (Page 234)

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

One cut:

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 during stock removal

1. The tool drills at the programmed spindle speed S (dependent on %S) as far as the first 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 during chipbreaking

1. The tool drills at the programmed spindle speed S (dependent on %S) as far as the first infeed depth (maximum infeed depth D).
2. Spindle stop and dwell time DT.
3. The tool retracts by the retraction distance V2 for chipbreaking.
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 refer to the machine manufacturer's specifications.

Procedure











1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Drilling" softkey.
3. Press the "Thread" and "Tapping" softkeys. The "Tapping" input window opens.

Parameters, G code program			Parameters, ShopTurn program		
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  (only for G code)	<ul style="list-style-type: none"> with compensating chuck without compensating chuck 	
Machining position  (only for G code)	<ul style="list-style-type: none"> Single position Drill hole at programmed position Position pattern Position with MCALL 	
Z0 (only for G code)	Reference point Z	mm
Machining - (with compensating chuck)  (only for G code)	<p>You can select the following technologies for tapping:</p> <ul style="list-style-type: none"> with encoder Tapping with spindle encoder without encoder Tapping without spindle encoder; <ul style="list-style-type: none"> Define "Pitch" parameter 	
SR (only for ShopTurn)	Spindle speed for retraction - (only for spindle speed "S")	rev/min
VR (only for ShopTurn)	Constant cutting rate for retraction - (only for constant cutting rate "V")	m/min
Machining surface  (only for ShopTurn)	<ul style="list-style-type: none"> Face C Face Y Peripheral surface C Peripheral surface Y 	
Position  (only for ShopTurn)	<ul style="list-style-type: none"> At the front (face) At the rear (face) Outside (peripheral surface) Inside (peripheral surface) 	
  (only for ShopTurn)	Clamp/release spindle The function must be set up by the machine manufacturer.	
Z1 	End point of the thread (abs) or thread length (inc) - only for G code and "ShopTurn machining surface face") It is inserted into the workpiece until it reaches Z1.	mm
Pitch - (only machining without encoder) (only for G code)	<ul style="list-style-type: none"> User input Pitch is obtained from the input Active feedrate Pitch is obtained from the feedrate 	

8.1 Drilling

Parameter	Description	Unit
Thread (only for G code) 	Direction of rotation of the thread <ul style="list-style-type: none"> • Righthand thread • Lefthand thread (only in mode "without compensating chuck")	
Table 	Thread table selection: <ul style="list-style-type: none"> • without • ISO metric • Whitworth BSW • Whitworth BSP • UNC 	
Selection 	Selection, table value: e.g. <ul style="list-style-type: none"> • 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 ... <ul style="list-style-type: none"> • in MODULUS: $MODULUS = Pitch/\pi$ • 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. <ul style="list-style-type: none"> • in mm/rev • in inch/rev The pitch is determined by the tool used.	MODULUS Turns/" mm/rev in/rev
αS (only for G code)	Starting angle offset - (only for tapping without compensating chuck)	Degrees
S (only for G code)	Spindle speed - (only for tapping without compensating chuck)	rev/min
Machining (not for G code "with compensating chuck") 	The following machining technologies can be selected: <ul style="list-style-type: none"> • 1 cut The thread is drilled in one cut without interruption. • Chipbreaking The drill is retracted by the retraction amount V2 for chipbreaking. • Chip removal The drill is retracted from the workpiece for stock removal. 	
D	Maximum depth infeed - (only when used without compensating chuck, stock removal or chipbreaking)	mm

Parameter	Description	Unit
Retraction U	Retraction distance - (for chipbreaking only) <ul style="list-style-type: none"> Manual Retraction distance after each machining step (V2) Automatic Without retraction distance after each machining step 	
V2	Retraction distance after each machining step – (only without compensating chuck, chipbreaking and manual retraction) Distance by which the drill is retracted for chipbreaking. V2 = automatic: The tool is retracted by one revolution.	mm
DT (only for G code)	Dwell time at final drilling depth in seconds	s
SR (only for G code)	Spindle speed for retraction - (only for when a compensating chuck is not used)	rev/min
SDE U (only for G code)	Direction of rotation after end of cycle: <ul style="list-style-type: none">    	
Technology U	<ul style="list-style-type: none"> Yes <ul style="list-style-type: none"> Exact stop Precontrol Acceleration Spindle No 	
Exact stop (only for technology, yes) U	<ul style="list-style-type: none"> 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 is reached 	
Precontrol (only for technology, yes) U	<ul style="list-style-type: none"> Behavior the same as it was before the cycle was called FFWON: with precontrol FFWOF: without precontrol 	
Acceleration (only for technology, yes) U	<ul style="list-style-type: none"> 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) U	<ul style="list-style-type: none"> Speed controlled: Spindle for MCAL; speed controlled operation Position controlled: Spindle for MCALL; position controlled operation 	

8.1.8 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.

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.

See also

Clamping the spindle (Page 234)

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 ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Drilling" softkey.
3. Press the "Thread" and "Cut thread" softkeys.
The "Drilling and thread milling" input window opens.



Parameters, G code program			Parameters, ShopTurn 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)	<ul style="list-style-type: none"> • 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
Machining surface (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y • Peripheral surface C • Peripheral surface Y 	
Position (only for ShopTurn)	<ul style="list-style-type: none"> • At the front (face) • At the rear (face) • Outside (peripheral surface) • Inside (peripheral surface) 	

Parameter	Description	Unit
F2 U	Feedrate for thread milling	mm/min mm/tooth
Table U	Thread table selection: <ul style="list-style-type: none"> without ISO metric Whitworth BSW Whitworth BSP UNC 	
Selection - (not for table "Without") U	Selection, table value: e.g. <ul style="list-style-type: none"> M3; M10; etc. (ISO metric) W3/4"; etc. (Whitworth BSW) G3/4"; etc. (Whitworth BSP) N1" - 8 UNC; etc. (UNC) 	
P U - (selection only possible for "Table without selection")	Pitch ... <ul style="list-style-type: none"> in MODULUS: $\text{MODULUS} = \text{Pitch}/\pi$ in turns per inch: Used with pipe threads, for example. <p>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.</p> <ul style="list-style-type: none"> in mm/rev in inch/rev <p>The pitch is determined by the tool used.</p>	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 U	<ul style="list-style-type: none"> Climbing: Mill thread in one cycle. Conventional: Mill thread in one cycle. Climbing - conventional: Mill thread in two cycles: rough cutting is performed by conventional milling with defined allowances, then finish cutting is performed with climb milling with milling feedrate FS. 	
FS U	Finishing feedrate rate - (only for climbing - conventional milling)	mm/min mm/tooth

8.1.9 Positions and position patterns

Function

After you have programmed the technology (cycle call), you must program the positions. Several position patterns are available:

- Arbitrary positions
- Position on a line, on a grid or frame
- Position on a full or pitch circle

Programming a position pattern in ShopTurn

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.

Note

The number of positions that can be programmed in the one "Positions" step is limited to a maximum of 400!

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 320)").

8.1.9.1 Approach/retraction

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 technological follow-up 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

- ShopTurn

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 hole circle are machined counterclockwise.

- G codes

For G code, for lines/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 hole circle are machined counterclockwise.













8.1.10 Arbitrary positions (CYCLE802)**8.1.10.1 Function****Function**

















The "Arbitrary positions" cycle allows you to program positions freely, i.e. rectangular or polar. Individual positions are approached in the order in which you program them. Press softkey "Delete all" to delete all positions programmed in X/Y.

Procedure

1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Drilling" softkey.
3. Press the "Positions" and "Arbitrary positions" softkeys. The "Positions" input window opens.

8.1 Drilling

Parameter	Description	Unit
LAB (only for G code)	Repeat jump label for position	
PL  (only for G code)	Machining plane	
Machining surface  (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y • Peripheral surface C • Peripheral surface Y 	
Position  (only for ShopTurn)	<ul style="list-style-type: none"> • At the front (face) • At the rear (face) • Outside (peripheral surface) • Inside (peripheral surface) 	
Coordinate system  (only for ShopTurn)	<ul style="list-style-type: none"> • right-angled or polar Dimensions in right-angled coordinates or polar coordinates (only for face C and face Y) • right-angled or cylindrical Dimensions in right-angled coordinates or cylindrical coordinates - (only for peripheral surface C) 	
X0 Y0 X1  ...X8  Y1  ...Y8  (only for G code)	X coordinate of 1st position (abs) Y coordinate of 1st position (abs) X coordinate for additional positions (abs or inc) Y coordinate for additional positions (abs or inc)	mm mm mm mm
Z0 CP X0 Y0 X1  ... X7  Y1  ... Y7  (only for ShopTurn)	Face C and face Y - at right angles: Z coordinate of the reference point (abs) Positioning angle for machining area (only for face Y) X coordinate of 1st position (abs) Y coordinate of 1st position (abs) X coordinate for additional positions (abs or inc) Incremental dimension: The sign is also evaluated. Y coordinate for additional positions (abs or inc) Incremental dimension: The sign is also evaluated.	mm Degrees mm mm mm mm

Parameter	Description	Unit
Z0 CP C0 L0 C1  ... C7  L1  ... L7  (only for ShopTurn)	Face C and face Y - polar: Z coordinate of the reference point (abs) Positioning angle for machining area (only for face Y) C coordinate of 1st position (abs) 1. Position of hole with reference to Y axis (abs) C coordinate for additional positions (abs or inc) Incremental dimension: The sign is also evaluated. Distance to position (abs or inc) Incremental dimension: The sign is also evaluated.	mm Degrees Degrees mm Degrees mm
X0 Y0 Z0 Y1  ...Y7  Z1  ...Z7  (only for ShopTurn)	Peripheral surface - at right angles: Cylinder diameter \varnothing (abs) Y coordinate of 1st position (abs) Z coordinate of 1st position (abs) Y coordinate for additional positions (abs or inc) Incremental dimension: The sign is also evaluated. Z coordinate for additional positions (abs or inc) Incremental dimension: The sign is also evaluated.	mm mm mm mm mm
C0 Z0 C1  ...C7  Z1  ... Z7  (only for ShopTurn)	Peripheral surface C - cylindrical: C coordinate of 1st position (abs) 1. Position of hole with reference to Z axis (abs) C coordinate for additional positions (abs or inc) Incremental dimension: The sign is also evaluated. Additional positions in the Z axis (abs or inc) Incremental dimension: The sign is also evaluated.	Degrees mm Degrees mm
X0 C0 Y0 Z0 Y1  ...Y7  Z1  ...Z7  (only for ShopTurn)	Peripheral surface Y: Reference point in X direction (abs) Positioning angle for machining surface Y coordinate of 1st position (abs) Z coordinate of 1st position (abs) Y coordinate for additional positions (abs or inc) Incremental dimension: The sign is also evaluated. Z coordinate for additional positions (abs or inc) Incremental dimension: The sign is also evaluated.	mm Degrees mm mm mm mm

8.1.11 Position pattern line (HOLES1), grid or frame (CYCLE801)

Function

You can program the following pattern using the "Position pattern" cycle:

- Line (HOLES1)

In the "Line" selection option you can program any number of positions at equal distances along a line.

- Grid (CYCLE801)

You can use the "Grid" selection option to program any number of positions 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 (CYCLE801)





You can use the "Frame" selection option to program any number of positions 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 ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Drilling" softkey.
3. Press the "Positions" and "Line" softkeys.
The "Position Pattern" input window opens.

Parameter	Description	Unit
LAB (only for G code)	Repeat jump label for position	
PL  (only for G code)	Machining plane	
Machining surface  (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y • Peripheral surface C • Peripheral surface Y 	
Position  (only for ShopTurn)	<ul style="list-style-type: none"> • At the front (face) • At the rear (face) • Outside (peripheral surface) • Inside (peripheral surface) 	
Position pattern 	Selection option for the following patterns: <ul style="list-style-type: none"> • Line • Matrix • Box 	
X0 Y0 $\alpha 0$ (only for G-Code)	X coordinate of the reference point X (abs) This position must be programmed absolutely in the 1st call. Y coordinate of the reference point Y (abs) This position must be programmed absolutely in the 1st call. Angle of rotation of the line referred to the X axis Positive angle: Line is rotated counterclockwise. Negative angle: Line is rotated clockwise.	mm mm Degrees
Z0 X0 Y0 $\alpha 0$ (only for ShopTurn)	Face C: Z coordinate of the reference point (abs) X coordinate of the reference point – first position (abs) Y coordinate of the reference point – first position (abs) Angle of rotation of line in relation to the X axis Positive angle: Line is rotated counterclockwise. Negative angle: Line is rotated clockwise.	mm mm mm Degrees
Z0 CP X0 Y0 $\alpha 0$ (only for ShopTurn)	Face Y: Z coordinate of the reference point (abs) Positioning angle for machining area X coordinate of the reference point – first position (abs) Y coordinate of the reference point – first position (abs) Angle of rotation of line in relation to the X axis Positive angle: Line is rotated counterclockwise. Negative angle: Line is rotated clockwise.	mm Degrees mm mm Degrees

8.1 Drilling

Parameter	Description	Unit
X0	Peripheral surface C: Cylinder diameter \varnothing (abs)	mm
Y0	Y coordinate of the reference point – first position (abs)	mm
Z0	Z coordinate of the reference point – first position (abs)	mm
$\alpha 0$ (only for ShopTurn)	Angle of rotation of line with reference to Y axis Positive angle: Line is rotated counterclockwise. Negative angle: Line is rotated clockwise.	Degrees
X0	Peripheral surface Y: X coordinate of the reference point (abs)	mm
C0	Positioning angle for machining surface	Degrees
Y0	Y coordinate of the reference point – first position (abs)	mm
Z0	Z coordinate of the reference point – first position (abs)	mm
$\alpha 0$ (only for ShopTurn)	Angle of rotation of line with reference to Y axis Positive angle: Line is rotated counterclockwise. Negative angle: Line is rotated clockwise.	Degrees
L0	Line Distance of the 1st position to reference point	mm
L	Distance between the positions	mm
N	Number of positions - (for line position pattern only)	
αX	Grid or frame 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	

8.1.12 Circle position pattern (HOLES2)

Function

You can program holes on a full circle or pitch circle with defined radius with the "Circle position pattern" cycle. The basic angle of rotation (α_0) 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.





Procedure







1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Drilling" softkey.
3. Press the "Positions" and "Full circle" softkeys. The "Position circle" input window opens. In the "Full/pitch circle" field, use the cursor to select "Full circle".

Parameter	Description	Unit
LAB (only for G code)	Repeat jump label for position	
PL (only for G code)	Machining plane	
Circular pattern 	Selection option for the following patterns: <ul style="list-style-type: none"> • Pitch circle • Full circle 	
X0	X coordinate of the reference point (abs)	mm
Y0	Y coordinate of the reference point (abs)	mm
α_0	Starting angle for first position referred to the X axis. Positive angle: Circle is rotated counterclockwise. Negative angle: Circle is rotated clockwise.	Degrees
α_1	Advance angle - (for pitch circle only)	Degrees
R	Radius	mm
N	Number of positions	
Positioning (only for G code)	<ul style="list-style-type: none"> • 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. 	

8.1 Drilling

Parameter	Description	Unit
Machining surface  (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y • Peripheral surface C • Peripheral surface Y 	
Position  (only for ShopTurn)	<ul style="list-style-type: none"> • At the front (face) • At the rear (face) • Outside (peripheral surface) • Inside (peripheral surface) 	
Position  (only for ShopTurn)	Selection options for the following positions - (only for face C/Y) <ul style="list-style-type: none"> • center • off-center 	
center/ off-center Z0 X0 Y0 α 0 α 1 R N Positioning  (only for ShopTurn)	<p>Face C:</p> Position circle center on the face surface Position circle off-center on the face surface Z coordinate of the reference point (abs) X coordinate of the reference point (abs) – (only for off-center) Y coordinate of the reference point (abs) – (only for off-center) Starting angle for first position referred to the X axis. Positive angle: Circle is rotated counterclockwise. Negative angle: Circle is rotated clockwise. Advance angle - (for pitch circle only) Radius Number of positions	mm mm mm Degrees Degrees mm
	<ul style="list-style-type: none"> • 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. 	

Parameter	Description	Unit
center/ off-center Z0 CP X0 or L0  Y0 or C0  $\alpha 0$ $\alpha 1$ R N Positioning  (only for ShopTurn)	<p>Face Y:</p> <p>Position circle center on the face surface</p> <p>Position circle off-center on the face surface</p> <p>Z coordinate of the reference point (abs)</p> <p>Positioning angle for machining area</p> <p>X coordinate of the reference point (abs) or reference point length, polar – (only for off-center)</p> <p>Y coordinate of the reference point (abs) or reference point angle, polar – (only for off-center)</p> <p>Starting angle for first position referred to the X axis. Positive angle: Circle is rotated counterclockwise. Negative angle: Circle is rotated clockwise.</p> <p>Advance angle - (for pitch circle only)</p> <p>Radius</p> <p>Number of positions</p> <ul style="list-style-type: none"> • 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. 	mm Degrees mm mm Degrees Degrees Degrees mm
X0 Z0 $\alpha 0$ $\alpha 1$ N only for ShopTurn)	<p>Peripheral surface C:</p> <p>Cylinder diameter \varnothing (abs)</p> <p>Z coordinate of the reference point (abs)</p> <p>Starting angle for first position referred to the Y axis. Positive angle: Circle is rotated counterclockwise. Negative angle: Circle is rotated clockwise.</p> <p>Advance angle - (for pitch circle only)</p> <p>Number of positions</p>	mm mm Degrees Degrees
X0 C0 Y0 Z0 $\alpha 0$ $\alpha 1$ N R Positioning  (only for ShopTurn)	<p>Peripheral surface Y:</p> <p>X coordinate of the reference point (abs)</p> <p>Positioning angle for machining surface</p> <p>Y coordinate of the reference point (abs)</p> <p>Z coordinate of the reference point (abs)</p> <p>Starting angle for first position referred to the Y axis. Positive angle: Circle is rotated counterclockwise. Negative angle: Circle is rotated clockwise.</p> <p>Advance angle - (for pitch circle only)</p> <p>Number of positions</p> <p>Radius</p> <ul style="list-style-type: none"> • 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. 	mm Degrees mm mm Degrees Degrees Degrees mm

8.1.13 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 ShopTurn 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 coordinates (X, Y) as well as a checkbox with the state (activated = on / deactivated = off) are displayed.
 The actual 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.

8.1.14 Repeating positions

8.1.14.1 Function

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 ShopTurn). 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 ShopTurn 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 ShopTurn)	Enter the number of the position pattern	

8.2 Rotate

8.2.1 General

In all turning cycles apart from contour turning (CYCLE95), in the combined roughing and finishing mode, when finishing it is possible to reduce the feedrate as a percentage.

**Machine manufacturer**

Please also refer to the machine manufacturer's specifications.

8.2.2 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 method

- Roughing

In 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 would reduce the infeed depth to 2.5 to create 4 equally sized cuts.

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 is raised immediately. The angle beyond which rounding is performed is stored in a machine data element.



Machine manufacturer

Please also refer to the machine manufacturer's instructions.

If the tool does not round the corner at the end of the cut, it is raised by the safety distance or a value specified in the machine data at 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 also refer to the machine manufacturer's instructions.



- **Finishing**


Finishing is performed in the same direction as roughing. The cycle automatically selects and deselects tool radius compensation during finishing.

Approach/retraction


1. The tool first moves at rapid traverse to the starting point of the machining operation calculated internally in the cycle (reference point + safety distance).
2. The tool moves to the first infeed depth at rapid traverse.
3. The first cut is made at machining feedrate.
4. The tool rounds the contour at machining feedrate or is raised at rapid traverse (see "Roughing").
5. The tool is moved at rapid traverse to the starting point for the next infeed depth.
6. The next cut is made at machining feedrate.
7. Steps 4 to 6 are repeated until the final depth is reached.
8. The tool moves back to the safety distance at rapid traverse.

Procedure


1. The part program or ShopTurn 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.


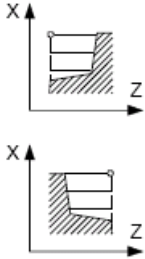
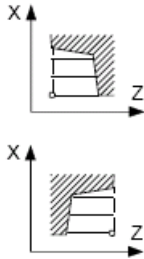
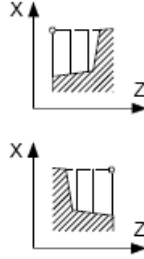
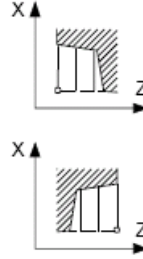





- OR
 - Stock removal cycle with oblique lines, radii, or chamfers.
 

The "Stock Removal 3" input window opens.

Parameters, G code program			Parameters, ShopTurn program		
PL	Machining plane		T	Tool name	
SC	Safety clearance	mm	D	Cutting edge number	
F	Feedrate	*	F	Feedrate	mm/rev
			S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
Machining 	<ul style="list-style-type: none"> • ▽ (roughing) • ▽▽▽ (finishing) 	
Position 	Machining position:    	
Machining direction	Stock removal direction (transverse or longitudinal) in the coordinate system	
	Parallel to the Z axis (longitudinal)	Parallel to the X axis (transverse)

8.2 Rotate

Parameter	Description	Unit
	<div style="display: flex; justify-content: space-around;"> <div style="text-align: center;"> <p>External</p>  </div> <div style="text-align: center;"> <p>Internal</p>  </div> <div style="text-align: center;"> <p>External</p>  </div> <div style="text-align: center;"> <p>internal</p>  </div> </div>	
X0	Reference point in X \varnothing (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)	
Z1 	End point Z (abs) or end point Z in relation to Z0 (inc)	
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
	<p>Parameter selection of intermediate point</p> <p>The intermediate point can be determined through position specification or angle. The following combinations are possible - (not for stock removal 1 and 2)</p> <ul style="list-style-type: none"> • XM ZM • XM α1 • XM α2 • α1 ZM • α2 ZM • α1 α2 	
XM 	Intermediate point X \varnothing (abs) or intermediate point X in relation to X0 (inc)	mm
ZM 	Intermediate point Z (abs or inc)	mm
α 1	Angle of the 1st edge	Degrees
α 2	Angle of the 2nd edge	Degrees

* Unit of feedrate as programmed before the cycle call

8.2.3 Groove (CYCLE930)

Function

You can use the "Groove" cycle to manufacture symmetrical and asymmetrical grooves on any straight contour elements.

You can machine outer or inner grooves in the longitudinal or transverse directions. 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 $D + \text{safety clearance}$ with rapid traverse.
4. The tool cuts a groove next to the first groove with infeed depth $2 \cdot D$.
5. The tool moves back by $D + \text{safety clearance}$ with rapid traverse.
6. The tool cuts alternating in the first and second groove with the infeed depth $2 \cdot D$, until the final depth $T1$ is reached.






Between the individual grooves, the tool moves back by $D + \text{safety clearance}$ 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 ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Turning" softkey.
 
3. Press the "Groove" softkey.
 
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.

Parameters, G code program			Parameters, ShopTurn program		
PL	Machining plane		T	Tool name	
SC	Safety clearance	mm	D	Cutting edge number	
F	Feedrate	*	F	Feedrate	mm/rev
			S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
Machining 	<ul style="list-style-type: none"> ▽ (roughing) ▽▽▽ (finishing) ▽ + ▽▽▽ (roughing and finishing) 	
Position 	Groove position: 	
X0	Reference point in X \emptyset	mm
Z0	Reference point in Z	mm
B1	Groove width	mm
T1	Groove depth \emptyset (abs) or groove depth referred to X0 or Z0 (inc)	mm
D	<ul style="list-style-type: none"> 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, refer to 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
$\alpha 1, \alpha 2$	Flank angle 1 or flank angle 2 - (only for grooves 2 and 3) Asymmetric grooves can be described by separate angles. The angles can be between 0 and $< 90^\circ$.	Degrees
FS1...FS4 or R1...R4 	Chamfer width (FS1...FS4) or rounding radius (R1...R4) - (only for grooves 2 and 3)	mm
$\alpha 0$	Angle of the incline – (only for groove 3)	Degrees

* Unit of feedrate as programmed before the cycle call

8.2.4 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 ShopTurn 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.







Parameters, G code program (undercut, form E)			Parameters, ShopTurn program (undercut, form E)		
PL	Machining plane		T	Tool name	
SC	Safety clearance	mm	D	Cutting edge number	
F	Feedrate	*	F	Feedrate	mm/rev
			S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
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 \emptyset	mm
Z0	Reference point Z	mm
X1 	Allowance in X \emptyset (abs) or allowance in X (inc)	mm
VX 	Cross feed \emptyset (abs) or cross feed (inc)	mm

* Unit of feedrate as programmed before the cycle call

8.2 Rotate

Parameters, G code program (undercut, form F)			Parameters, ShopTurn program (undercut, form F)		
PL	Machining plane		T	Tool name	
SC	Safety clearance	mm	D	Cutting edge number	
F	Feedrate	*	F	Feedrate	mm/rev
			S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
Position 	Form F machining position: 	
	Undercut size according to DIN table: e.g.: F0.6 x 0.3 (undercut form F)	
X0	Reference point X \emptyset	mm
Z0	Reference point Z	mm
X1 	Allowance in X \emptyset (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 \emptyset (abs) or cross feed (inc)	mm

* Unit of feedrate as programmed before the cycle call

8.2.5 Thread undercuts (CYCLE940)

Function

You can use the "Thread undercut DIN" or "Thread undercut" cycle 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 ShopTurn 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.









8.2 Rotate

Parameters, G code program (undercut, thread DIN)			Parameters, ShopTurn program (undercut, thread DIN)		
PL	Machining plane		T	Tool name	
SC	Safety clearance	mm	D	Cutting edge number	
F	Feedrate	*	F	Feedrate	mm/rev
			S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
Machining 	<ul style="list-style-type: none"> ▽ (roughing) ▽▽▽ (finishing) ▽ + ▽▽▽ (roughing and finishing) 	
Position 	Machining position: 	
Machining direction 	<ul style="list-style-type: none"> Longitudinal Parallel to the contour 	
Form 	<ul style="list-style-type: none"> Normal (form A) Short (form B) 	
P 	Thread pitch (select from the preset DIN table or enter)	mm/rev
X0	Reference point X \emptyset	mm
Z0	Reference point Z	mm
α	Insertion angle	Degrees
VX 	Cross feed \emptyset (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

Parameters, G code program (undercut, thread)			Parameters, ShopTurn program (undercut, thread)		
PL	Machining plane		T	Tool name	
SC	Safety clearance	mm	D	Cutting edge number	
F	Feedrate	*	F	Feedrate	mm/rev
			S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
Machining 	<ul style="list-style-type: none"> ▽ (roughing) ▽▽▽ (finishing) ▽ + ▽▽▽ (roughing and finishing) 	
Machining direction 	<ul style="list-style-type: none"> Longitudinal Parallel to the contour 	
Position 	Machining position: 	
X0	Reference point X \emptyset	mm
Z0	Reference point Z	mm
X1 	Undercut depth referred to X \emptyset (abs) or undercut depth referred to X (inc)	mm
Z1 	Allowance Z (abs or inc)	mm
R1	Rounding radius 1	mm
R2	Rounding radius 2	mm
α	Insertion angle	Degrees
VX 	Cross feed \emptyset (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

8.2.6 Thread turning (CYCLE99)

Function

You can use the "Longitudinal thread", "Tapered thread" or "Face thread" cycle 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.

See also

Setting data for cycles (Page 216)

Thread synchronizing (Page 119)

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.











Procedure for longitudinal thread, tapered thread, or face thread

1. The part program or ShopTurn 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.





8.2 Rotate

Parameters, G code program (thread, longitudinal)			Parameters, ShopTurn program (thread, longitudinal)		
PL	Machining plane		T	Tool name	
			D	Cutting edge number	
			S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
Table 	Thread table selection: <ul style="list-style-type: none"> 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: <ul style="list-style-type: none"> 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: $G = \frac{ P_e^2 - P^2 }{2 * Z_1} \text{ [mm/rev}^2\text{]}$ 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/rev ²
Machining 	<ul style="list-style-type: none"> ∇ (roughing) ∇∇∇ (finishing) ∇ + ∇∇∇ (roughing and finishing) 	

Parameter	Description	Unit
Infeed (only for ∇ and $\nabla + \nabla\nabla\nabla$) 	<ul style="list-style-type: none"> Linear: Infeed with constant cutting depth Degressive: Infeed with constant cutting cross-section 	
Thread 	<ul style="list-style-type: none"> Internal thread External thread 	
X0	Reference point X from thread table \varnothing (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
LW 	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.	mm
or LW2 	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
or LW2 = LR 		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 	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	mm
or α P	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. This means that 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 $\nabla + \nabla\nabla\nabla$)	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











8.2 Rotate

Parameter	Description	Unit	
U	Finishing allowance in X and Z – (only for ∇ and $\nabla + \nabla\nabla\nabla$)	mm	
NN	Number of noncuts - (only for $\nabla\nabla\nabla$ and $\nabla + \nabla\nabla\nabla$)		
VR	Return distance (inc)	mm	
Multiple threads 	No		
	$\alpha 0$	Starting angle offset	Degrees
	Yes		
	N	Number of thread turns The thread turns are distributed evenly across the periphery 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: 	<ul style="list-style-type: none"> • Complete, or • from turn N1 N1 (1...4) start thread N1 = 1...N  or • only thread NX NX (1...4) 1 from N threads  		

Parameters, G code program (thread, conical)			Parameters, ShopTurn program (thread, conical)		
PL	Machining plane		T	Tool name	
			D	Cutting edge number	
			S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
P U	<ul style="list-style-type: none"> 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	<p>Change in thread pitch per revolution - (only for P = mm/rev or in/rev)</p> <p>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.</p> <p>If the start and end pitch of the thread are known, the pitch change to be programmed can be calculated as follows:</p> $G = \frac{ P_e^2 - P^2 }{2 * Z_1} \text{ [mm/rev}^2\text{]}$ <p>The meanings are as follows: P_e: End pitch of thread [mm/rev] P: Start pitch of thread [mm/rev] Z₁: Thread length [mm]</p> <p>A larger pitch results in a larger distance between the thread turns on the workpiece.</p>	mm/rev ²
Machining U	<ul style="list-style-type: none"> ▽ (roughing) ▽▽▽ (finishing) ▽ + ▽▽▽ (roughing and finishing) 	
Infeed (only for ▽ and ▽ + ▽▽▽) U	<ul style="list-style-type: none"> Linear: Infeed with constant cutting depth Degressive: Infeed with constant cutting cross-section 	
Thread U	<ul style="list-style-type: none"> Internal thread External thread 	
X0	Reference point X Ø (abs, always diameter)	mm
Z0	Reference point Z (abs)	mm
X1 or X1α U	<p>End point X Ø (abs) or end point in relation to X0 (inc) or Thread taper</p> <p>Incremental dimension: The sign is also evaluated.</p>	mm or degrees

8.2 Rotate












Parameter	Description	Unit
Z1 	End point Z (abs) or end point in relation to Z0 (inc) Incremental dimension: 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) $\alpha > 0$: Infeed along the rear flank $\alpha < 0$: Infeed along the front flank $\alpha = 0$: 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.	mm 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. This means that you can increase the tool life. $\alpha > 0$: Start at the rear flank $\alpha < 0$: Start at the front flank	
D1 or ND  (only for ∇ and $\nabla + \nabla\nabla\nabla$)	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
U	Finishing allowance in X and Z – (only for ∇ and $\nabla + \nabla\nabla\nabla$)	mm
NN	Number of noncuts - (only for $\nabla\nabla\nabla$ and $\nabla + \nabla\nabla\nabla$)	
VR	Return distance (inc)	mm
Multiple threads 	No $\alpha 0$ Starting angle offset Yes N Number of thread turns The thread turns are distributed evenly across the periphery of the turned part; the 1st thread turn is always located at 0°.	 Degrees






Parameter	Description		Unit
	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: U	<ul style="list-style-type: none"> Complete, or from turn N1 N1 (1...4) start thread N1 = 1...N U or only thread NX NX (1...4) 1 from N threads U 	

Parameters, G code program (thread, face)			Parameters, ShopTurn program (thread, face)		
PL	Machining plane		T	Tool name	
			D	Cutting edge number	
			S / V U	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
P U	<ul style="list-style-type: none"> 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	<p>Change in thread pitch per revolution - (only for P = mm/rev or in/rev)</p> <p>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.</p> <p>If the start and end pitch of the thread are known, the pitch change to be programmed can be calculated as follows:</p> $G = \frac{ P_e^2 - P^2 }{2 * Z_1} \text{ [mm/rev}^2\text{]}$ <p>The meanings are as follows: P_e: End pitch of thread [mm/rev] P: Start pitch of thread [mm/rev] Z₁: Thread length [mm]</p> <p>A larger pitch results in a larger distance between the thread turns on the workpiece.</p>	mm/rev ²

8.2 Rotate

Parameter	Description	Unit
Machining 	<ul style="list-style-type: none"> ∇ (roughing) ∇∇∇ (finishing) ∇ + ∇∇∇ (roughing and finishing) 	
Infeed (only for ∇ and ∇ + ∇∇∇) 	<ul style="list-style-type: none"> Linear: Infeed with constant cutting depth Degressive: Infeed with constant cutting cross-section 	
Thread 	<ul style="list-style-type: none"> Internal thread External thread 	
X0	Reference point X Ø (abs, always diameter)	mm
Z0	Reference point Z (abs)	mm
X1 	End point of the thread Ø (abs) or thread length (inc) Incremental dimensions: The sign is also evaluated.	mm
LW  or LW2  or LW2 = LR 	<p>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.</p> <p>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)</p>	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	<p>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</p> <p>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.</p>	Degrees
  	<p>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. This means that you can increase the tool life.</p> <p>α > 0: Start at the rear flank α < 0: Start at the front flank</p>	

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	
U	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 across the periphery 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: 	<ul style="list-style-type: none"> • Complete, or • from turn N1 N1 (1...4) start thread N1 = 1...N  or • only thread NX NX (1...4) 1 from N threads  		

8.2.7 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.

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.

Procedure for thread chain












1. The part program or ShopTurn 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 "Thread chain" softkey.
The "Thread Chain" input window opens.

Parameters, G code program			Parameters, ShopTurn program		
PL	Machining plane		T	Tool name	
SC	Safety clearance	mm	D	Cutting edge number	
			S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
Machining 	<ul style="list-style-type: none"> • ∇ (roughing) • ∇∇∇ (finishing) • ∇ + ∇∇∇ (roughing and finishing) 	
Infeed (only for ∇ and ∇ + ∇∇∇) 	<ul style="list-style-type: none"> • Linear: Constant cutting depth infeed • Degressive: Constant cutting cross-section infeed 	
Thread 	<ul style="list-style-type: none"> • Internal thread • External thread 	
X0	Reference point X Ø (abs, always diameter)	mm
Z0	Reference point Z (abs)	mm
P0	Thread pitch 1	mm/rev in/rev turns/" MODULUS
X1 or X1α 	<ul style="list-style-type: none"> • Intermediate point 1 X Ø (abs) or • Intermediate point 1 in relation to X0 (inc) or • Thread taper 1 Incremental dimensions: The sign is also evaluated.	mm Degrees
Z1	<ul style="list-style-type: none"> • Intermediate point 1 Z (abs) or • Intermediate point 1 in relation to Z0 (inc) 	mm

8.2 Rotate

Parameter	Description	Unit	
P1	Thread pitch 2 (unit as parameterized for P0)	mm/rev in/rev turns/" MODULUS	
X2 or X2α 	<ul style="list-style-type: none"> Intermediate point 2 X Ø (abs) or Intermediate point 2 in relation to X1 (inc) or Thread taper 2 Incremental dimensions: The sign is also evaluated.	mm Degrees	
Z2 	<ul style="list-style-type: none"> 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 	<ul style="list-style-type: none"> End point X Ø (abs) or End point 3 in relation to X2 (inc) or Thread taper 3 	mm Degrees	
Z3 	<ul style="list-style-type: none"> End point Z Ø (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	
 	<ul style="list-style-type: none"> 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

8.2.8 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 or ShopTurn program to be processed 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.

Parameters, G code program			Parameters, ShopTurn program		
PL	Machining plane		T	Tool name	
SC	Safety clearance	mm	D	Cutting edge number	
F	Feedrate	*	F	Feedrate	mm/rev
			S / V	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
DIR	Direction of spindle rotation 	
S	Spindle speed	rev/min
V	Constant cutting rate	m/min
SV	Maximum speed limit - (only for constant cutting rate V)	rev/min
X0	Reference point in X \varnothing (abs, always diameter)	mm
Z0	Reference point in Z (abs)	mm
FS or R	Chamfer width or rounding radius	mm
X1	Depth for speed reduction \varnothing (abs) or depth for speed reduction in relation to X0 (inc)	mm
FR (only for ShopTurn)	Reduced feedrate	mm/rev
FR (only for G code)		*
SR	Reduced speed	rev/min
X2	Final depth \varnothing (abs) or final depth in relation to X1 (inc)	mm

* Unit of feedrate as programmed before the cycle call

8.3 Contour turning

8.3.1 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.

Programming

For example, the programming procedure for stock removal is as follows:

Note

When programming in G code, it must be ensured that the contours are located after the end of program identifier!

1. Enter the blank contour

If, when removing stock along the contour, you want to take into account a blank contour (and no cylinder or no allowance) as blank shape, then you must define the contour of the blank before you define the finished-part contour. Compile the blank 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 - only for G code program

8.3 Contour turning

4. Stock removal along the contour (roughing)

The contour is machined longitudinally, transversely or parallel to the contour.

5. Remove residual material (roughing)

When removing stock along the contour, ShopTurn automatically detects residual material that has been left. For G code programming, when removing stock, it must first be decided whether to 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.

8.3.2 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.


ShopTurn 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 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 active 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)

Graphic 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.

8.3.3 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 or ShopTurn program to be processed has been created and you are in the editor.
 2. Press the "Contour turning" softkey.


 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 Ø (abs)	mm
Transition to contour start 	Type of transition <ul style="list-style-type: none"> • 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:  <ul style="list-style-type: none"> • In the negative direction of the horizontal axis  <ul style="list-style-type: none"> • In the positive direction of the horizontal axis  <ul style="list-style-type: none"> • In the negative direction of the vertical axis  <ul style="list-style-type: none"> • In the positive direction of the vertical axis 	
Additional commands	<p>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.</p> <p>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.</p> <p>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.</p> <p>If you wish to avoid this, then it is possible to use additional commands when programming.</p> <p>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.</p> <p>Note: The additional commands are only effective for finishing!</p>	

8.3.4 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
- 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 the 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.

- Selecting a dialog box

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.

Producing exact contour transitions

The axis moves in the 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, 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.

- 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.

8.3 Contour turning



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 line (e.g. Z)" input window opens.

- OR



The "Straight line (e.g. X)" input window opens.

- OR



The "Straight line (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 $\alpha 2$ 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
$\alpha 1$	Starting angle to Z axis			Degrees
$\alpha 2$	Angle to the preceding element			Degrees
Transition to next element 	Type of transition <ul style="list-style-type: none"> • 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  <ul style="list-style-type: none"> •  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 Ø (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 <ul style="list-style-type: none"> • 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  <ul style="list-style-type: none"> •  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 \varnothing (abs) or end point X (inc)	mm
$\alpha 1$	Starting angle to Z axis	Degrees
$\alpha 2$	Angle to the preceding element	Degrees
Transition to next element 	Type of transition <ul style="list-style-type: none"> • Radius • Chamfer 	
Radius	R Transition to following element - radius	mm
Chamfer	FS Transition to following element - chamfer	mm
CA	Grinding allowance  <ul style="list-style-type: none"> •  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 	<ul style="list-style-type: none"> Clockwise direction of rotation  Counterclockwise direction of rotation  	
Z 	End point Z (abs or inc)	mm
X 	End point X \varnothing (abs) or end point X (inc)	mm
K 	Circle center point K (abs or inc)	mm
I 	Circle center point I \varnothing (abs) or circle center point I (inc)	mm
$\alpha 1$	Starting angle to Z axis	Degrees
$\beta 1$	End angle to Z axis	Degrees
$\beta 2$	Opening angle	Degrees
Transition to next element 	Type of transition <ul style="list-style-type: none"> Radius Chamfer 	
Radius	R Transition to following element - radius	mm
Chamfer	FS Transition to following element - chamfer	mm
CA	Grinding allowance  <ul style="list-style-type: none">  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.

8.3.5 Entering the master dimension

If you would like to finish your workpiece to an exact fit, you can input the master dimension directly into the parameter screen form during programming.

Specify the master dimension as follows:

F<Diameter/Length> <Tolerance class> <Tolerance quality>

"F" identifies that a master dimension follows, i.e. in this case, a hole.

Example: F20h7

Possible tolerance classes:

A, B, C, D, E, F, G, H, J, T, U, V, X, Y, Z

Upper-case characters: Holes

Lower case letters: Shafts

Possible tolerance qualities:

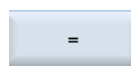
1 to 18, if they are not restricted by DIN standard 7150.

Fit calculator

A fit calculator supports you when making entries.

Procedure

1. Position the cursor on the desired entry field.
 2. Press the <=> key.
The calculator is displayed.
 3. Press the "Fit shaft" or "Fit hole" softkey.
"F" (for hole) or "f" (for shaft) is automatically inserted in front of the entry fields for diameter or length data, tolerance class and tolerance quality.
 4. Enter the diameter or length value in the first field.
 5. In the second field, select the tolerance class and in the third field, enter the tolerance quality.
 6. 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.



Press the "Accept" softkey.

The calculated value is accepted and displayed in the entry field of the window.

Rejecting entries



Press the "Delete" softkey to reject your entries.

8.3.6 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 or ShopTurn 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 ShopTurn 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.

8.3.7 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 part program to be executed has been created and you are in the editor.
2. Press the "Contour turning" softkey.
3. Press the "Contour" and "Contour call" softkeys.
The "Contour Call" input window opens.
4. Assign parameters to the contour selection.

8.3 Contour turning

Parameter	Description	Unit
Contour selection 	<ul style="list-style-type: none"> Contour name Labels Subprogram Labels in the subprogram 	
Contour name	CON: Contour name	
Labels	<ul style="list-style-type: none"> LAB1: Label 1 LAB2: Label 2 	
Subprogram	PRG: Subprogram	
Labels in the subprogram	<ul style="list-style-type: none"> PRG: Subprogram LAB1: Label 1 LAB2: Label 2 	

8.3.8 Stock removal (CYCLE952)

Function

For stock removal, the cycle takes into account a blank that can comprise a cylinder, an allowance on the finished-part contour or any blank contour. You must define a blank contour as a separate closed contour in advance of the finished-part 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").

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 with each cut (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 refer to the machine manufacturer's specifications.

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.

The percentage for the alternating cutting depth is saved in a machine data element.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

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.

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.

Residual material machining / naming conventions

G code program

For multi-channel systems, cycles attach a "_C" and a two-digit number of the specific channel to the names of the programs to be generated, e.g. for channel 1 "_C01".

This is the reason that 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 for 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.

See also



General information (Page 351)



Machining type











You can select the machining mode (roughing or 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.

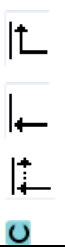

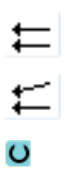


Procedure






1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Contour turning" softkey.

3. Press the "Stock removal" softkey.
The "Stock Removal" input window opens.

Parameters, G code program			Parameters, ShopTurn program		
PRG	Name of the program to be generated		T	Tool name	
PL	Machining plane		D	Cutting edge number	
RP	Retraction plane – (only for machining direction, longitudinal, inner)	mm	F	Feedrate	mm/rev
SC	Safety clearance	mm	S / V	Spindle speed or constant cutting rate	m/min
F	Feedrate	*			
Residual material 	With subsequent residual material removal <ul style="list-style-type: none"> • Yes • No 				
CONR	Name to save the updated blank contour for residual material removal				

Parameter	Description	Unit
Machining 	<ul style="list-style-type: none"> • ∇ (roughing) • ∇∇∇ (finishing) 	
Machining direction 	<ul style="list-style-type: none"> • Face  • Longitudinal  • Parallel to the contour  <div style="display: flex; flex-direction: column; align-items: center;"> <div style="display: flex; align-items: center; margin-bottom: 5px;"> • From inside to outside  </div> <div style="display: flex; align-items: center; margin-bottom: 5px;"> • From outside to inside  </div> <div style="display: flex; align-items: center; margin-bottom: 5px;"> • From end face to rear side  </div> <div style="display: flex; align-items: center;"> • From rear side to end face  </div> </div>	
	The machining direction depends on the stock removal direction and choice of tool.	
Position 	<ul style="list-style-type: none"> • 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

8.3 Contour turning

Parameter	Description	Unit
	Always round on the contour. Never round on the contour. Only round to the previous intersection.	
	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
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 ∇) <ul style="list-style-type: none"> • 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) <ul style="list-style-type: none"> • For blank description, cylinder <ul style="list-style-type: none"> – Version, absolute: Cylinder dimension \varnothing (abs) – Version incremental: Allowance (inc) to maximum values of the CYCLE62 finished part contour • For blank description, allowance <ul style="list-style-type: none"> – Allowance on the CYCLE62 finished part contour (inc) 	mm

Parameter	Description	Unit
ZD	<ul style="list-style-type: none"> - (only for ▽ machining) - (only for blank description, cylinder and allowance) • For blank description, cylinder <ul style="list-style-type: none"> – Version, absolute: Cylinder dimension (abs) – Version incremental: Allowance (inc) to maximum values of the CYCLE62 finished part contour • For blank description, allowance <ul style="list-style-type: none"> – Allowance on the CYCLE62 finished part contour (inc) 	mm
Allowance 	<p>Allowance for pre-finishing - (only for ▽▽▽)</p> <ul style="list-style-type: none"> • Yes U1 contour allowance • No 	
U1	<p>Compensation allowance in X and Z direction (inc) – (only for allowance)</p> <ul style="list-style-type: none"> • Positive value: Compensation allowance is kept • Negative value: Compensation allowance is removed in addition to finishing allowance 	mm
Set machining area limits 	<p>Set machining area limits</p> <ul style="list-style-type: none"> • Yes • No 	
XA XB  ZA ZB 	<p>with limited machining area only, yes:</p> <ol style="list-style-type: none"> 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 	<p>Machine relief cuts</p> <ul style="list-style-type: none"> • Yes • No 	
FR	Insertion feedrate, relief cuts	

* Unit of feedrate as programmed before the cycle call

8.3.9 Stock removal rest (CYCLE952)

Function

Using the "Stock removal residual" function, you 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 ShopTurn, the updated unmachined-part contour is automatically generated. For a C 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.



Software option

For stock removal of residual material, you require the option "residual material detection and machining".

Procedure








1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Contour turning" softkey.
3. Press the "Stock removal residual material" softkey.
The "Stock removal residual material" input window opens.

Parameters, G code program		Parameters, ShopTurn program	
PRG	Name of the program to be generated	T	Tool name
PL	Machining plane	D	Cutting edge number

Parameters, G code program			Parameters, ShopTurn program		
RP	Retraction plane	mm	F	Feedrate	mm/rev
SC	Safety clearance	mm	S / V	Spindle speed or constant cutting rate	rpm m/min
F	Feedrate	*			
CON	Name of the updated blank contour for residual material machining (without the attached character "_C" and double-digit number)				
Residual material	With subsequent residual material removal				
	<ul style="list-style-type: none"> • Yes • No 				
CONR	Name to save the updated unmachined-part contour for residual material removal - (only "Yes" for residual material removal)				

Parameter	Description	Unit
Machining	<ul style="list-style-type: none"> • ∇ (roughing) • ∇∇∇ (finishing) 	
Machining direction	<ul style="list-style-type: none"> • Face • Longitudinal • Parallel to the contour <ul style="list-style-type: none"> • From inside to outside • From outside to inside • From end face to rear side • From rear side to end face <p>The machining direction depends on the stock removal direction and choice of tool.</p>	
Position	<ul style="list-style-type: none"> • front • back • internal • external 	
D	Maximum depth infeed - (only for ∇)	mm
XDA	1. Grooving limit tool (abs) – (only for face machining direction)	mm
XDB	2. Grooving limit tool (abs) – (only for face machining direction)	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	
	only for align cut segmentation at the edge: Constant cutting depth alternating cutting depth	

8.3 Contour turning

Parameter	Description	Unit
Allowance 	Allowance for pre-finishing - (only for ∇∇∇) <ul style="list-style-type: none"> • Yes U1 contour allowance • No 	s
U1	Compensation allowance in X and Z direction (inc) – (only for allowance) <ul style="list-style-type: none"> • Positive value: Compensation allowance is kept • Negative value: Compensation allowance is removed in addition to finishing allowance 	mm
Set machining area limits 	Set machining area limits <ul style="list-style-type: none"> • 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 <ul style="list-style-type: none"> • Yes • No 	
FR	Insertion feedrate, relief cuts	

* Unit of feedrate as programmed before the cycle call

8.3.10 Plunge-cutting (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.

Precondition

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 unmachined part contour and the second call is the finished-part contour (also see Chapter "Programming").

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.

Feedrate interruption

To prevent the occurrence of excessively long chips during machining, you can program a feedrate interruption.

See also

General information (Page 351)

Machining type

You can freely select the machining type (roughing or finishing).

For more detailed information, please refer to section "Stock removal".





Procedure

1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Contour turning" softkey.
3. Press the "Grooving" softkey.
The "Grooving" input window opens.

8.3 Contour turning

Parameters, G code program			Parameters, ShopTurn program		
PRG	Name of the program to be generated		T	Tool name	
PL	Machining plane		D	Cutting edge number	
RP	Retraction plane – (only for machining direction, longitudinal, inner)	mm	F	Feedrate	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 <ul style="list-style-type: none"> • Yes • No 				
CONR	Name to save the updated unmachined-part contour for residual material removal - (only "Yes" for residual material removal)				

Parameter	Description	Unit
Machining	<ul style="list-style-type: none"> • ∇ (roughing) • ∇∇∇ (finishing) 	
Machining direction	<ul style="list-style-type: none"> • Face • Longitudinal 	
Position	<ul style="list-style-type: none"> • front • back • internal • external 	
D	Maximum depth infeed - (only for ∇)	mm
XDA	1. Grooving limit tool (abs) – (only for face machining direction)	mm
XDB	2. 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 (unmachined part) description (only for ∇) <ul style="list-style-type: none"> • 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) 	

Parameter	Description	Unit
XD	- (only for ▽ machining) - (only for unmachined part description, cylinder and allowance) <ul style="list-style-type: none"> For unmachined part description, cylinder <ul style="list-style-type: none"> Version, absolute: Cylinder dimension \varnothing (abs) Version incremental: Allowance (inc) to maximum values of the CYCLE62 finished part contour For unmachined part description, allowance <ul style="list-style-type: none"> Allowance on the CYCLE62 finished part contour (inc) 	mm
ZD	- (only for ▽ machining) - (only for unmachined part description, cylinder and allowance) <ul style="list-style-type: none"> For unmachined part description, cylinder <ul style="list-style-type: none"> Version, absolute: Cylinder dimension (abs) Version incremental: Allowance (inc) to maximum values of the CYCLE62 finished part contour For unmachined part description, allowance <ul style="list-style-type: none"> Allowance on the CYCLE62 finished part contour (inc) 	mm
Allowance 	Allowance for pre-finishing - (only for ▽▽▽) <ul style="list-style-type: none"> Yes U1 contour allowance No 	mm
U1	Compensation allowance in X and Z direction (inc) – (only for allowance) <ul style="list-style-type: none"> Positive value: Compensation allowance is kept Negative value: Compensation allowance is removed in addition to finishing allowance 	mm
Set machining area limits 	Set machining area limits <ul style="list-style-type: none"> Yes No 	
XA XB  ZA ZB 	with limited machining area only, yes: <ol style="list-style-type: none"> Limit XA \varnothing Limit XB \varnothing (abs) or 2nd limit referred to XA (inc) Limit ZA 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

8.3.11 Plunge-cutting rest (CYCLE952)

Function

The "Grooving residual material" function is used when you want to machine the material that remained after grooving along the contour.

During grooving, the cycle automatically detects any residual material and generates an updated blank contour. For a G code program, the function must have been previously selected. 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.



Software option

To machine residual material, you require the "Residual material detection and machining" option.

Procedure






1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Contour turning" softkey.
3. Press the "Grooving residual material" softkey.
The "Grooving residual material" input window is opened.

Parameters, G code program			Parameters, ShopTurn program		
PRG	Name of the program to be generated		T	Tool name	
PL	Machining plane		D	Cutting edge number	
RP	Retraction plane – (only for longitudinal machining direction)	mm	F	Feedrate	mm/rev
SC	Safety clearance	mm	S / V	Spindle speed or constant cutting rate	rpm m/min
F	Feedrate	*			
CON	Name of the updated blank contour for residual material machining (without the attached character "_C" and double-digit number)				
Residual material	With subsequent residual material removal				
	<ul style="list-style-type: none"> • Yes • No 				
CONR	Name to save the updated unmachined-part contour for residual material removal - (only "Yes" for residual material removal)				

Parameter	Description	Unit
Machining	<ul style="list-style-type: none"> • ∇ (roughing) • ∇∇∇ (finishing) 	
Machining direction	<ul style="list-style-type: none"> • Face • Longitudinal 	
Position	<ul style="list-style-type: none"> • front • back • internal • external 	
D	Maximum depth infeed - (only for ∇)	mm
XDA	1. Grooving limit tool (abs) – (only for face machining direction)	mm
XDB	2. 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
Allowance	Allowance for pre-finishing - (only for ∇∇∇) <ul style="list-style-type: none"> • Yes <ul style="list-style-type: none"> U1 contour allowance • No 	mm

8.3 Contour turning

Parameter	Description	Unit
U1	Compensation allowance in X and Z direction (inc) – (only for allowance) <ul style="list-style-type: none"> • Positive value: Compensation allowance is kept • Negative value: Compensation allowance is removed in addition to finishing allowance 	mm
Set machining area limits 	Set machining area limits <ul style="list-style-type: none"> • Yes • No 	
XA XB  ZA ZB 	with limited machining area only, yes: <ol style="list-style-type: none"> 1. Limit XA \varnothing 2. Limit XB \varnothing (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

8.3.12 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.

Blank

For plunge turning, 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.

Precondition

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 unmachined part contour and the second call is the finished-part contour (also see Chapter "Programming (Page 351)").

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.

Feedrate interruption

To prevent the occurrence of excessively long chips during machining, you can program a feedrate interruption.

Machining type

You can freely select the machining type (roughing or finishing).

For more detailed information, please refer to section "Stock removal".

Procedure








1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Contour turning" softkey.
3. Press the "Plunge turning" softkey.
The "Plunge turning" input window opens.

Parameters, G code program		Parameters, ShopTurn program		
PRG	Name of the program to be generated	T	Tool name	
PL	Machining plane	D	Cutting edge number	
RP	Retraction plane – (only for longitudinal machining direction)	mm	S / V	Spindle speed or constant cutting rate
				rpm m/min
SC	Safety clearance	mm		
Residual material	With subsequent residual material removal			
	<ul style="list-style-type: none"> • Yes • No 			
CONR	Name to save the updated unmachined-part contour for residual material removal - (only "Yes" for residual material removal)			

8.3 Contour turning

Parameter	Description	Unit
FX (only ShopTurn)	Feedrate in X direction	mm/rev
FZ (only ShopTurn)	Feedrate in Z direction	mm/rev
FX (only G Code)	Feedrate in X direction	*
FZ (only for G code)	Feedrate in Z direction	*
Machining U	<ul style="list-style-type: none"> ∇ (roughing) ∇∇∇ (finishing) 	
Machining direction U	<ul style="list-style-type: none"> Face Longitudinal 	
Position U	<ul style="list-style-type: none"> front back internal external 	
D	Maximum depth infeed - (only for ∇)	mm
XDA	1. Grooving limit tool (abs) – (only for face machining direction)	mm
XDB	2. 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 ∇)	mm
DI	For zero: Continuous cut - (only for ∇)	mm
BL U	Blank (unmachined part) description (only for ∇) <ul style="list-style-type: none"> 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 U	- (only for ∇ machining) - (only for unmachined part description, cylinder and allowance) <ul style="list-style-type: none"> For unmachined part description, cylinder <ul style="list-style-type: none"> Version, absolute: Cylinder dimension \varnothing (abs) Version incremental: Allowance (inc) to maximum values of the CYCLE62 finished part contour For unmachined part description, allowance Allowance on the CYCLE62 finished part contour (inc) 	mm

Parameter	Description	Unit
ZD 	<ul style="list-style-type: none"> - (only for ▽ machining) - (only for unmachined part description, cylinder and allowance) • For unmachined part description, cylinder <ul style="list-style-type: none"> – Version, absolute: Cylinder dimension (abs) – Version incremental: Allowance (inc) to maximum values of the CYCLE62 finished part contour • For unmachined part description, allowance <ul style="list-style-type: none"> – Allowance on the CYCLE62 finished part contour (inc) 	mm
Allowance 	<p>Allowance for pre-finishing - (only for ▽▽▽)</p> <ul style="list-style-type: none"> • Yes U1 contour allowance • No 	mm
U1	<p>Compensation allowance in X and Z direction (inc) – (only for allowance)</p> <ul style="list-style-type: none"> • Positive value: Compensation allowance is kept • Negative value: Compensation allowance is removed in addition to finishing allowance 	mm
Set machining area limits 	<p>Set machining area limits</p> <ul style="list-style-type: none"> • Yes • No 	
XA XB  ZA ZB 	<p>with limited machining area only, yes:</p> <ol style="list-style-type: none"> 1. Limit XA \emptyset 2. Limit XB \emptyset (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	mm

* Unit of feedrate as programmed before the cycle call

8.3.13 Plunge-turning rest (CYCLE952)

Function

The "Plunge turning residual material" function is used when you want to machine the material that remained after plunge turning.

For plunge turning, the cycle automatically detects any residual material and generates an updated blank contour. For a G code program, the function must have been previously selected 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.



Software option

To machine residual material, you require the "Residual material detection and machining" option.

Procedure

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



2. Press the "Contour turning" softkey.



3. Press the "Plunge turning residual material" softkey.
The "Plunge turning residual material" input window opens.

Parameters, G code program			Parameters, ShopTurn program		
PRG	Name of the program to be generated		T	Tool name	
PL	Machining plane		D	Cutting edge number	
RP	Retraction plane – (only for longitudinal machining direction)	mm	F	Feedrate	mm/rev
SC	Safety clearance	mm	S / V	Spindle speed or constant cutting rate	rpm m/min
CON	Name of the updated blank contour for residual material machining (without the attached character "_C" and double-digit number)				
Residual material	With subsequent residual material removal				
	<ul style="list-style-type: none"> • Yes • No 				
CONR	Name to save the updated unmachined-part contour for residual material removal - (only "Yes" for residual material removal)				

Parameter	Description	Unit
FX (only ShopTurn)	Feedrate in X direction	mm/rev
FZ (only ShopTurn)	Feedrate in Z direction	mm/rev
FX (only G Code)	Feedrate in X direction	*
FZ (only for G code)	Feedrate in Z direction	*
Machining U	<ul style="list-style-type: none"> ▽ (roughing) ▽▽ (finishing) 	
Machining direction U	<ul style="list-style-type: none"> Face Longitudinal 	
Position U	<ul style="list-style-type: none"> front back internal external 	
D	Maximum depth infeed - (only for ▽)	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 ▽)	mm
XDA	1. Grooving limit tool Ø (abs) – (end face or rear face only)	mm
XDB	2. Grooving limit tool Ø (abs) – (end face or rear face only)	mm
Allowance U	Allowance for prefinishing <ul style="list-style-type: none"> Yes <ul style="list-style-type: none"> U1 contour allowance No 	
DI	For zero: Continuous cut - (only for ▽)	mm
U1	Compensation allowance in X and Z direction (inc) – (only for allowance) <ul style="list-style-type: none"> Positive value: Compensation allowance is kept Negative value: Compensation allowance is removed in addition to finishing allowance 	mm
Set machining area limits U	Set machining area limits <ul style="list-style-type: none"> Yes No 	
XA XB U ZA ZB U	with limited machining area only, yes: <ol style="list-style-type: none"> Limit XA Ø Limit XB Ø (abs) or 2nd limit referred to XA (inc) Limit ZA 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

8.4 Milling

8.4.1 Face milling (CYCLE61)

Function

You can face mill any workpiece with the "Face milling" cycle.

A rectangular surface is always machined. The rectangle is obtained from corner points 1 and 2 - which for a ShopTurn program - are pre-assigned with the values of the blank part dimensions from the program header.

Workpieces with and without limits can be face-milled.

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.

See also

Clamping the spindle (Page 234)

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 ShopTurn 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.

8.4 Milling

Parameters, G code program			Parameters, ShopTurn program		
PL 	Machining plane		T	Tool name	
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 surface  (only for ShopTurn)	<ul style="list-style-type: none"> • Face Y • Peripheral surface Y 	
  (only for ShopTurn)	Clamp/release spindle The function must be set up by the machine manufacturer.	
Machining 	The following machining technologies can be selected: <ul style="list-style-type: none"> • ▽ (roughing) • ▽▽▽ (finishing) 	
Direction 	Same direction of machining <ul style="list-style-type: none"> •  •  Alternating direction of machining <ul style="list-style-type: none"> •  •  	
(only for G code) X0 Y0 Z0 X1  Y1  Z1 	The positions refer to the reference point: <ul style="list-style-type: none"> Corner point 1 in X Corner point 1 in Y Height of blank Corner point 2X (abs) or corner point 2X in relation to X0 (inc) Corner point 2Y (abs) or corner point 2Y in relation to Y0 (inc) Height of blank (abs) or height of blank in relation to Z0 (inc) 	mm mm mm mm mm mm

Parameter	Description	Unit
(only ShopTurn)	Face Y: The positions refer to the reference point:	
CP	Positioning angle for machining area - only for face Y	Degrees
X0	Corner point 1 in X	mm
Y0	Corner point 1 in Y	mm
Z0	Height of blank	mm
X1 	Corner point 2 in X (abs) or corner point 2X in relation to X0 (inc)	mm
Y1 	Corner point 2 in Y (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
(only ShopTurn)	Peripheral surface Y: The positions refer to the reference point:	
C0	Positioning angle for machining area - (only for peripheral surface Y)	Degrees
Y0	Corner point 1 in Y	mm
Z0	Corner point 1 in Z	mm
X0	Height of blank	mm
Y1 	Corner point 2 in Y (abs) or corner point 2X in relation to Y0 (inc)	mm
Z1 	Corner point 2 in Z (abs) or corner point 2Y in relation to Z0 (inc)	mm
X1 	Height of blank (abs) or height of blank in relation to X0 (inc)	mm
DXY 	Maximum plane infeed Alternately, you can specify the plane infeed in %, as a ratio → plane infeed (mm) to 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.

8.4.2 Rectangular pocket (POCKET3)

Function

You can use the "Mill rectangular pocket" cycle to mill any rectangular pockets on the face or peripheral surface. .

The following machining variants are available:

- Mill rectangular pocket from solid material.
- Pre-drill 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).

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.

See also

Clamping the spindle (Page 234)

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**
Roughing involves machining the individual planes of the pocket one after the other from the center out, until depth Z1 or X1 is reached.
- **Finishing**
During finishing, the edge is always machined first. The pocket edge is approached on the quadrant that joins the corner radius. During 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 (base finishing) is omitted.
- **Chamfering**
Chamfering involves edge breaking at the upper edge of the rectangular pocket.

Procedure

1. The part program or ShopTurn 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, G code program			Parameters, ShopTurn program		
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 (only for G code) 	<p>The following different reference point positions can be selected:</p> <ul style="list-style-type: none"> • (center) • (bottom left) • (bottom right) • (top left) • (top right) <p>The reference point (highlighted in blue) is displayed in the Help screen.</p>	
Machining surface (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y • Peripheral surface C • Peripheral surface Y 	

8.4 Milling

Parameter	Description	Unit
Position  (only for ShopTurn)	<ul style="list-style-type: none"> At the front (face) At the rear (face) Outside (peripheral surface) Inside (peripheral surface) 	
  (only for ShopTurn)	Clamp/release spindle (only for end face Y/peripheral surface Y) The function must be set up by the machine manufacturer.	
Machining 	The following machining technologies can be selected: <ul style="list-style-type: none"> ▽ (roughing) ▽▽ (finishing) ▽▽ edge (edge finishing) Chamfering 	
Machining position 	<ul style="list-style-type: none"> Single position Mill rectangular pocket at the programmed position (X0, Y0, Z0). Position pattern Position with MCALL 	
X0 Y0 Z0 (only for G code)	The positions refer to the reference point: Reference point X – (only for single position) Reference point Y – (only for single position) Reference point Z	mm mm mm
X0 or L0  Y0 or C0  Z0 (only for ShopTurn)	Face C: The positions refer to the reference point: Reference point X or reference point length polar – (only for single position) Reference point Y or reference point angle polar - (only for single position) Reference point Z - (only for single position)	mm mm or degrees mm
CP X0 or L0  Y0 or C0  Z0 (only for ShopTurn)	Face Y: The positions refer to the reference point: Positioning angle for machining area – (only single position) Reference point X or reference point length polar – (only for single position) Reference point Y or reference point angle polar - (only for single position) Reference point Z - (only for single position)	Degrees mm mm or degrees mm
Y0 or C0  Z0 X0 (only for ShopTurn)	Peripheral surface C: The positions refer to the reference point: Reference point Y or reference point angle polar - (only for single position) Reference point Z - (only for single position) Cylinder diameter \varnothing – (only for single position)	mm or degrees mm mm

Parameter	Description	Unit
C0	Peripheral surface Y: The positions refer to the reference point: Positioning angle for machining surface – (only for single position)	Degrees
Y0	Reference point Y – (only for single position)	mm
Z0	Reference point Z – (only for single position)	mm
X0 (only for ShopTurn)	Reference point X – (only for single position)	mm
W	Pocket width	mm
L	Pocket length	mm
R	Corner radius	mm
$\alpha 0$	Angle of rotation	Degrees
Z1	Pocket depth (abs) or depth relative to Z0 (inc) – (only for ∇ , $\nabla\nabla\nabla$ or $\nabla\nabla\nabla$ edge)	mm
		
DXY	<ul style="list-style-type: none"> Maximum plane infeed Maximum plane infeed as a percentage of the milling cutter diameter - (only for ∇ and $\nabla\nabla\nabla$)	mm %
		
DZ	Maximum depth infeed – (only for ∇ , $\nabla\nabla\nabla$ or $\nabla\nabla\nabla$ edge)	mm
UXY	Plane finishing allowance – (only for ∇ , $\nabla\nabla\nabla$ or $\nabla\nabla\nabla$ edge)	mm
UZ	Depth finishing allowance – (only for ∇ , $\nabla\nabla\nabla$)	mm
Insertion	The following insertion modes can be selected – (only for ∇ , $\nabla\nabla\nabla$ or $\nabla\nabla\nabla$ edge):	
	<ul style="list-style-type: none"> 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 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 remove the slope caused by insertion. Oscillating: Insert with oscillation along center axis of rectangular pocket(only for G code) 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 remove the slope caused by insertion. 	

8.4 Milling

Parameter	Description	Unit
  (only for ShopTurn)	Clamp/release spindle (only for end face C/peripheral surface C, if inserted vertically) The function must be set up by the machine manufacturer.	
FZ (only for G code)	Depth infeed rate – (for vertical insertion only)	*
 (only for ShopTurn)	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 milling cutter radius; otherwise, material will remain.	mm
EW	Maximum insertion angle – (for insertion with oscillation only)	Degrees
Solid machining (for roughing only) 	<ul style="list-style-type: none"> Complete machining The rectangular pocket is milled from the solid material. Remachining The size of any existing smaller rectangular pocket or hole is increased in 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	Insertion depth of tool tip (abs or inc) – (for chamfering only)	mm

* Unit of feedrate as programmed before the cycle call

8.4.3 Circular pocket (POCKET4)

Function

You can use the "Circular pocket" cycle to mill circular pockets on the face or peripheral surface.

The following machining variants are available:

- Mill circular pocket from solid material.
- Pre-drill 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).

For milling with the "Circular pocket" function two methods are available: the plane-by-plane method and the helical method.

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.

See also

Clamping the spindle (Page 234)

Approach/retraction during plane-by-plane 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 distance.
2. The tool is inserted into the material according to the method selected.
3. The circular pocket is always machined from inside out using the selected machining method.
4. The tool moves back to the safety distance at rapid traverse.

Approach/retraction during helical machining

In helical machining, 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 distance.
2. Infeed to the first machining diameter.
3. The circular pocket is machined to pocket depth using the selected machining method.
4. The tool moves back to the safety distance at rapid traverse.

Machining type: Plane by plane

When milling circular pockets, you can select these methods for the following machining types:

- Roughing

Roughing involves machining the individual planes of the circular pocket one after the other from the center out, until depth Z1 or X1 is reached.

- Finishing

During finishing, the edge is always machined first. The pocket edge is approached on the quadrant that joins the pocket radius. During 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 (base finishing) is omitted.

Machining type: Helical

When milling circular pockets, you can select these methods for the following machining types:

- Roughing

During roughing, the circular pocket is machined downward with helical movements.

A full circle is made at pocket depth to remove the residual material.

The tool is removed from the edge and base in the quadrant and retracted with rapid traverse to the safety distance.

This process is repeated layer by layer, from inside out, until the circular pocket has been completely machined.

- Finishing

During finishing, machining begins at the edge and continues in a helical movement until the bottom is reached.

A full circle is made at pocket depth to remove the residual material.

The base is milled from outside in, using a spiral movement.

The tool is retracted with rapid traverse from the center of the pocket to a safety distance.

- Edge finishing

During edge finishing, machining begins at the edge and continues in a helical movement until the bottom is reached.

A full circle is made at pocket depth to remove the residual material.

The tool is removed from the edge and base in the quadrant and retracted with rapid traverse to the safety distance.

Chamfering machining

Chamfering involves edge breaking at the upper edge of the circular pocket.

Procedure

1. The part program or ShopTurn 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, G code program			Parameters, ShopTurn program		
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 surface (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y • Peripheral surface C • Peripheral surface Y 	
Position (only for ShopTurn)	<ul style="list-style-type: none"> • At the front (face) • At the rear (face) • Outside (peripheral surface) • Inside (peripheral surface) 	
 (only for ShopTurn)	Clamp/release spindle (only for end face Y/peripheral surface Y) The function must be set up by the machine manufacturer.	
Machining 	<ul style="list-style-type: none"> • ∇ (roughing, plane-by-plane or helical) • ∇∇∇ (finishing, plane-by-plane or helical) • ∇∇∇ edge (edge finishing, plane-by-plane or helical) • Chamfering 	

8.4 Milling

Parameter	Description	Unit
Machining type U	<ul style="list-style-type: none"> Plane by plane Solid machine circular pocket plane-by-plane Helical Solid machine circular pocket helically 	
Machining position U	<ul style="list-style-type: none"> 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 (only for G code)	The positions refer to the reference point: Reference point X – (only for single position) Reference point Y – (only for single position) Reference point Z	mm mm mm
X0 or L0 U Y0 or C0 U Z0 (only for ShopTurn)	Face C: The positions refer to the reference point: Reference point X or reference point length polar – (only for single position) Reference point Y or reference point angle polar – (only for single position) Reference point Z – (only for single position)	mm mm or degrees mm
CP X0 or L0 U Y0 or C0 U Z0 (only for ShopTurn)	Face Y: The positions refer to the reference point: Positioning angle for machining area – (only single position) Reference point X or reference point length polar – (only for single position) Reference point Y or reference point angle polar – (only for single position) Reference point Z – (only for single position)	Degrees mm mm or degrees mm
Y0 or C0 U Z0 X0 (only for ShopTurn)	Peripheral surface C: The positions refer to the reference point: Reference point Y or reference point angle polar – (only for single position) Reference point Z – (only for single position) Cylinder diameter \varnothing – (only for single position)	mm or degrees mm mm
C0 Y0 Z0 X0 (only for ShopTurn)	Peripheral surface Y: The positions refer to the reference point: Positioning angle for machining surface – (only for single position) Reference point Y – (only for single position) Reference point Z – (only for single position) Reference point X – (only for single position)	Degrees mm mm mm
\varnothing	Diameter of pocket	mm

Parameter	Description	Unit
Z1 	Pocket depth (abs) or depth relative to Z0/X0 (inc) – (only for ▽, ▽▽▽ and ▽▽▽ edge)	mm
DXY 	<ul style="list-style-type: none"> 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 ▽▽▽ edge)	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 ▽, ▽▽▽ and ▽▽▽ edge): <ul style="list-style-type: none"> Predrilled (only for G code) Perpendicular: Insert vertically at 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.	
  (only for ShopTurn)	Clamp/release spindle (only for end face C/peripheral surface C, if inserted vertically) The function must be set up by the machine manufacturer.	
FZ (only for G code)	Depth infeed rate – (for vertical insertion only)	*
FZ  (only for ShopTurn)	Depth infeed rate – (for vertical insertion only)	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 cutter radius, otherwise material will remain. Also make sure the circular pocket is not violated	mm
Solid machining  (only for G code)	<ul style="list-style-type: none"> Complete machining The circular pocket must be milled from a solid workpiece (e.g. casting). Remachining A small pocket or hole has already been machined in the workpiece, which needs to be enlarged. Parameters AZ, and Ø1 must be programmed. 	

Parameter	Description	Unit
FS	Chamfer width for chamfering - (for chamfering only)	mm
ZFS	Insertion depth of tool tip (abs or inc) - (for chamfering only)	mm
AZ (only for G code)	Depth of premachining - (for remachining only)	mm
Ø1 (only for G code)	Diameter of premachining - (for remachining only)	mm

* Unit of feedrate as programmed before the cycle call

8.4.4 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:



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 spigot is machined using only one infeed. If you want to machine the spigot using multiple infeeds, you must program the "Rectangular spigot" function several times, with a continually decreasing finishing allowance.

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.

See also

Clamping the spindle (Page 234)

Approach/retraction

1. The tool approaches the starting point at rapid traverse at the height of the retraction plane and adjusts to the safety distance. The starting point is on the positive X axis rotated through $\alpha 0$.
2. The tool traverses the spigot contour sideways in a semicircle at the machining feedrate. The tool first executes the infeed at machining depth, followed by the movement in the plane. Depending on the machining direction that has been programmed (up-cut/synchronism), the spigot is machined in a clockwise or counterclockwise direction.
3. When the spigot has been circumnavigated once, the tool is removed from the contour in a semicircle; the infeed to the next machining depth is then executed.
4. The spigot is approached again in a semicircle and circumnavigated once. This process is repeated until the programmed spigot depth is reached.
5. The tool moves back to the safety distance 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.












Procedure




1. The part program or ShopTurn 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.

8.4 Milling

Parameters, G code program			Parameters, ShopTurn program		
 PL	Machining plane		T	Tool name	
	Milling direction		D	Cutting edge number	
RP	Retraction plane	mm	F	Feedrate	mm/min mm/tooth
			S / V	Spindle speed or constant cutting rate	rpm m/min
SC	Safety clearance	mm			
F	Feedrate	*			

Parameter	Description	Unit
FZ (only for G code)	Depth infeed rate (only for ▽ and ▽▽▽)	*
Reference point  (only for ShopTurn)	The following different reference point positions can be selected: <ul style="list-style-type: none">  (center)  (bottom left)  (bottom right)  (top left)  (top right) 	
Machining surface  (only for ShopTurn)	<ul style="list-style-type: none"> Face C Face Y Peripheral surface Y 	
Position  (only for ShopTurn)	<ul style="list-style-type: none"> At the front (face) At the rear (face) Outside (peripheral surface) Inside (peripheral surface) 	
  (only for ShopTurn)	Clamp/release spindle (only for end face Y/peripheral surface Y) The function must be set up by the machine manufacturer.	
Machining 	The following machining technologies can be selected: <ul style="list-style-type: none"> ▽ (roughing) ▽▽▽ (finishing) Chamfering 	

Parameter	Description	Unit
Machining position U	<ul style="list-style-type: none"> Single position Mill rectangular pocket at the programmed position (X0, Y0, Z0). Position pattern Position with MCALL 	
X0 Y0 Z0 (only for G code)	The positions refer to the reference point: Reference point X – (only for single position) Reference point Y – (only for single position) Reference point Z	mm mm mm
X0 or L0 U Y0 or C0 U Z0 (only for ShopTurn)	Face C: The positions refer to the reference point: Reference point X or reference point length polar – (only for single position) Reference point Y or reference point angle polar – (only for single position) Reference point Z – (only for single position)	mm mm mm
CP X0 or L0 U Y0 or C0 U Z0 (only for ShopTurn)	Face Y: The positions refer to the reference point: Positioning angle for machining area – (only single position) Reference point X or reference point length polar – (only for single position) Reference point Y or reference point angle polar – (only for single position) Reference point Z – (only for single position)	Degrees mm mm or degrees mm
Y0 or C0 U Z0 X0 (only for ShopTurn)	Peripheral surface C: The positions refer to the reference point: Reference point Y or reference point angle polar – (only for single position) Reference point Z – (only for single position) Cylinder diameter \varnothing – (only for single position)	mm or degrees mm mm
C0 Y0 Z0 X0 (only for ShopTurn)	Peripheral surface Y: The positions refer to the reference point: Positioning angle for machining surface – (only for single position) Reference point Y – (only for single position) Reference point Z – (only for single position) Reference point X – (only for single position)	Degrees mm mm mm
W	Width of spigot	mm
L	Length of spigot	mm
R	Corner radius	mm
$\alpha 0$	Angle of rotation	Degrees
Z1 U	Spigot depth (abs) or depth relative to Z0 or X0 (inc) - (only for ∇ and $\nabla\nabla\nabla$)	mm
DZ	Maximum depth infeed - (only for ∇ and $\nabla\nabla\nabla$)	mm

Parameter	Description	Unit
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

8.4.5 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.

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.

See also

Clamping the spindle (Page 234)

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.







Procedure









1. The part program or ShopTurn 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.

8.4 Milling

Parameters, G code program			Parameters, ShopTurn program		
 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 surface (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y • Peripheral surface C • Peripheral surface Y 	
 Position (only for ShopTurn)	<ul style="list-style-type: none"> • At the front (face) • At the rear (face) • Outside (peripheral surface) • Inside (peripheral surface) 	
  (only for ShopTurn)	Clamp/release spindle (only for end face Y/peripheral surface Y) The function must be set up by the machine manufacturer.	
 Machining	The following machining technologies can be selected: <ul style="list-style-type: none"> • ▽ (roughing) • ▽▽▽ (finishing) • Chamfering 	
 Machining position	<ul style="list-style-type: none"> • Single position Mill circular spigot at the programmed position (X0, Y0, Z0). • Position pattern • Position with MCALL 	

Parameter	Description	Unit
X0 Y0 Z0 (only for G code)	The positions refer to the reference point: Reference point X – (only for single position) Reference point Y – (only for single position) Reference point Z	mm mm mm
X0 or L0  Y0 or C0  Z0 (only for ShopTurn)	Face C: The positions refer to the reference point: Reference point X or reference point length polar – (only for single position) Reference point Y or reference point angle polar – (only for single position) Reference point Z – (only for single position)	mm mm or degrees mm
CP X0 or L0  Y0 or C0  Z0 (only for ShopTurn)	Face Y: The positions refer to the reference point: Positioning angle for machining area – (only single position) Reference point X or reference point length polar – (only for single position) Reference point Y or reference point angle polar – (only for single position) Reference point Z – (only for single position)	Degrees mm mm or degrees mm
Y0 or C0  Z0 X0 (only for ShopTurn)	Peripheral surface C: The positions refer to the reference point: Reference point Y or reference point angle polar - (only for single position) Reference point Z - (only for single position) Cylinder diameter \varnothing – (only for single position)	mm or degrees mm mm
C0 Y0 Z0 X0 (only for ShopTurn)	Peripheral surface Y: The positions refer to the reference point: Positioning angle for machining surface – (only for single position) Reference point Y – (only for single position) Reference point Z – (only for single position) Reference point X – (only for single position)	Degrees mm mm mm
\varnothing	Diameter of spigot	mm
Z1 	Spigot depth (abs) or depth relative to Z0 or X0 (inc) - (only for ∇ and $\nabla\nabla\nabla$)	mm
DZ	Maximum depth infeed - (only for ∇ and $\nabla\nabla\nabla$)	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 $\nabla\nabla\nabla$)	mm
UZ	Depth finishing allowance (tool axis) - (only for ∇ and $\nabla\nabla\nabla$)	mm
\varnothing 1	Diameter of blank spigot (important for determining approach position) - (only for ∇ and $\nabla\nabla\nabla$)	mm

Parameter	Description	Unit
FS	Chamfer width for chamfering - (for chamfering only)	mm
ZFS	Insertion depth of tool tip (abs or inc) - (for chamfering only) (ZFS for machining surface, face C/Y or XFS for peripheral surface C/Y)	mm

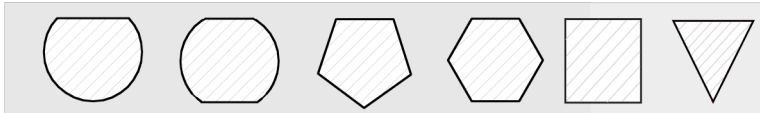
* Unit of feedrate as programmed before the cycle call

8.4.6 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:



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.

See also

Clamping the spindle (Page 234)

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 ShopTurn 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, G code program			Parameters, ShopTurn program		
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	*			

8.4 Milling

Parameter	Description	Unit
Machining surface  (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y 	
Position  (only for ShopTurn)	<ul style="list-style-type: none"> • Front • back 	
  (only for ShopTurn)	Clamp/release spindle (only for face Y) The function must be set up by the machine manufacturer.	
Machining 	<ul style="list-style-type: none"> • ∇ (roughing) • ∇∇∇ (finishing) • ∇∇∇ edge (edge finishing) • Chamfering 	
Machining position  (only for G code)	<ul style="list-style-type: none"> • Single position A multiple edge is milled at the programmed position (X0, Y0, Z0). • Position pattern Several multiple edges are milled at the programmed position pattern (e.g. pitch circle, grid, line). 	
X0 (only G code)	The positions refer to the reference point: Reference point X – (only for single position)	mm
Y0 (only G code)	Reference point Y – (only for single position)	mm
Z0	Reference point Z – (only for single position)	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 	<ul style="list-style-type: none"> • 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

Parameter	Description	Unit
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

8.4.7 Longitudinal groove (SLOT1)

Function

You can use the "Longitudinal groove" function to mill any longitudinal groove.

The following machining methods are available:

- Mill longitudinal groove from solid material.
- Pre-drill longitudinal groove in the center first if, for example, the milling cutter does not cut in the center (program the drilling, longitudinal groove and position program blocks in succession).

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.

Longitudinal slot with the width of the tool

When milling a longitudinal slot, which is located in parallel with the spindle axis, and which should be machined with the width of the tool, then the clamping remains active after insertion in order to achieve more accurate results.

The cycles identify this special case and do not cancel clamping after insertion if the following secondary conditions are fulfilled.

After machining, the clamping in the cycles is canceled again.

Supplementary conditions

- Finishing longitudinal slot with width = tool diameter
- Roughing longitudinal slot with (width - 2 * finishing allowance) = tool diameter

See also

Clamping the spindle (Page 234)

Approach/retraction

1. The tool approaches the center point of the slot at rapid traverse at the height of the retraction plane and adjusts to the safety distance.
2. The tool is inserted into the material according to the method selected.
3. The longitudinal slot is always machined from inside out using the selected machining method.
4. The tool moves back to the safety distance at rapid traverse.

Machining method





You can select any of the following machining methods for milling the longitudinal slot:












- **Roughing**
Roughing involves machining the individual planes of the slot one after the other from the inside out, until depth Z1 or X1 is reached.
- **Finishing**
During finishing, the edge is always machined first. The slot edge is approached on the quadrant that joins the corner radius. During 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 (base finishing) is omitted.
- **Chamfering**
Chamfering involves breaking the upper edge of the slot.

Procedure








1. The part program or ShopTurn 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, G code program			Parameters, ShopTurn program		
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  (only for G code)	Position of the reference point: <ul style="list-style-type: none">  (lefthand edge)  (inside left)  (center)  (inside right)  (righthand edge) 	
Machining surface  (only for ShopTurn)	<ul style="list-style-type: none"> Face C Face Y Peripheral surface C Peripheral surface Y 	
Position  (only for ShopTurn)	<ul style="list-style-type: none"> At the front (face) At the rear (face) Outside (peripheral surface) Inside (peripheral surface) 	
  (only for ShopTurn)	Clamp/release spindle (only for end face Y/peripheral surface Y) The function must be set up by the machine manufacturer.	
Machining 	The following machining technologies can be selected: <ul style="list-style-type: none"> ▽ (roughing) ▽▽▽ (finishing) ▽▽▽ edge (edge finishing) Chamfering 	

8.4 Milling

Parameter	Description	Unit
Machining position U	<ul style="list-style-type: none"> Single position Mill rectangular pocket at the programmed position (X0, Y0, Z0). Position pattern Position with MCALL 	
X0 Y0 Z0 (only for G code)	The positions refer to the reference point: Reference point X – (only for single position) Reference point Y – (only for single position) Reference point Z	mm mm mm
X0 or L0 U Y0 or C0 U Z0 (only for ShopTurn)	Face C: The positions refer to the reference point: Reference point X or reference point length polar – (only for single position) Reference point Y or reference point angle polar – (only for single position) Reference point Z – (only for single position)	mm mm or degrees mm
CP X0 or L0 U Y0 or C0 U Z0 (only for ShopTurn)	Face Y: The positions refer to the reference point: Positioning angle for machining area – (only single position) Reference point X or reference point length polar – (only for single position) Reference point Y or reference point angle polar - (only for single position) Reference point Z - (only for single position)	Degrees mm mm or degrees mm
Y0 or C0 U Z0 X0 (only for ShopTurn)	Peripheral surface C: The positions refer to the reference point: Reference point Y or reference point angle polar - (only for single position) Reference point Z - (only for single position) Cylinder diameter \varnothing – (only for single position)	mm or degrees mm mm
C0 Y0 Z0 X0 (only for ShopTurn)	Peripheral surface Y: The positions refer to the reference point: Positioning angle for machining surface – (only for single position) Reference point Y – (only for single position) Reference point Z – (only for single position) Reference point X – (only for single position)	Degrees mm mm mm
W	Slot width	mm
L	Slot length	mm
$\alpha 0$	Angle of rotation of slot Face: $\alpha 0$ refers to the X axis or to the position of C0 with a polar reference point Peripheral surface: $\alpha 0$ refers to the Y axis	Degrees
Z1 U	Slot depth (abs) or depth relative to Z0 or X0 (inc) - (only for ∇ and $\nabla\nabla\nabla$)	mm

Parameter	Description	Unit
DXY  (only ShopTurn)	<ul style="list-style-type: none"> 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 for the length (L) and width (W) of the slot. - (only for ▽ and ▽▽▽)	mm
UZ	Depth finishing allowance (tool axis) - (only for ▽ and ▽▽▽)	mm
Insertion 	The following insertion modes can be selected – (only for ▽, ▽▽▽ or ▽▽▽ edge): <ul style="list-style-type: none"> Predrilled (only for G code) Approach reference point shifted by the amount of the safety clearance with G0. Vertical Insert vertically in longitudinal groove center: 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. Helical(only for G code) insertion on 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 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. 	
  (only for ShopTurn)	Clamp/release spindle (only for end face C/peripheral surface C, if inserted vertically) The function must be set up by the machine manufacturer.	
FZ (only for G code)	Depth infeed rate – (for vertical insertion only)	*
 (only for ShopTurn)	Depth infeed rate - (only for insertion, predrilled and perpendicular)	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 milling cutter radius; otherwise, material will remain.	mm

Parameter	Description	Unit
EW	Maximum insertion angle – (for insertion with oscillation only)	Degrees
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

8.4.8 Circumferential groove (SLOT2)

Function

You can mill one or several circumferential grooves of equal size on a full or pitch circle with the "circumferential groove" cycle.

Tool size

Please note that there is a minimum size for the milling cutter used to machine the circumferential groove:

- Roughing:
1/2 groove width W – finishing allowance $UXY \leq$ milling cutter diameter
- Finishing:
1/2 groove width $W \leq$ milling cutter diameter
- Edge finishing:
Finishing allowance $UXY \leq$ milling cutter diameter

Annular groove

To create an annular groove, you must enter the following values for the "Number N" and "Aperture angle α_1 " parameters:

$$N = 1$$

$$\alpha_1 = 360^\circ$$

Approach/retraction

1. At the height of the retraction plane, the tool approaches the center point of the semicircle at the end of the groove at rapid traverse and adjusts to the safety distance.
2. It is then inserted into the workpiece at the machining feedrate, allowing for the maximum Z direction infeed (for face machining), X direction infeed (for peripheral machining), and the finishing allowance. Depending on the machining direction (up-cut or down-cut), the circumferential groove is machined in a clockwise or counterclockwise direction.
3. When the first circumferential groove is finished, the tool moves to the retraction plane at rapid traverse.
4. The next circumferential groove is approached along a straight line or circular path and then machined.
5. The rapid traverse feedrate for positioning on a circular path is specified in a machine data element.

Machining type

You can select the machining mode for milling the circumferential slot as follows:

- Roughing

During roughing, the individual planes of the slot are machined one after the other from center point of the semicircle at the end of the slot until depth Z1 is reached.

- Finishing

In "Finishing" mode, the edge is always machined first until depth Z1 is reached. The slot edge is approached on the quadrant, which joins the radius. In the last infeed, the base is finished from the center point of the semicircle to the end of the slot.

- Finishing of edge

Edge finishing is performed in the same way as finishing, except that the last infeed (finish base) is omitted.

- Chamfer





Chamfering involves edge breaking at the upper edge of the circumferential groove.






Procedure







1. The part program or ShopTurn 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.






8.4 Milling

Parameters, G code program			Parameters, ShopTurn program		
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 surface  (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y • Peripheral surface C • Peripheral surface Y 	
Position  (only for ShopTurn)	<ul style="list-style-type: none"> • At the front (face) • At the rear (face) • Outside (peripheral surface) • Inside (peripheral surface) 	
  (only for ShopTurn)	Clamp/release spindle (only for end face Y/peripheral surface Y) The function must be set up by the machine manufacturer.	
Machining 	<ul style="list-style-type: none"> • ∇ (roughing) • ∇∇∇ (finishing) • ∇∇∇ edge (edge finishing) • Chamfering 	
FZ (only for G code)	Depth infeed rate	*

Parameter	Description	Unit
Circular pattern 	<ul style="list-style-type: none"> 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 α_2. 	
X0 Y0 Z0 (only for G code)	The positions refer to the reference point: Reference point X – (only for single position) Reference point Y – (only for single position) Reference point Z – (only for single position)	mm mm mm
X0 or L0  Y0 or C0  Z0 (only for ShopTurn)	Face C: The positions refer to the reference point: Reference point X or reference point length polar – (only for single position) Reference point Y or reference point angle polar – (only for single position) Reference point Z – (only for single position)	mm mm or degrees mm
CP X0 or L0  Y0 or C0  Z0 (only for ShopTurn)	Face Y: The positions refer to the reference point: Positioning angle for machining area – (only single position) Reference point X or reference point length polar – (only for single position) Reference point Y or reference point angle polar - (only for single position) Reference point Z - (only for single position)	Degrees mm mm or degrees mm
Y0 or C0  Z0 X0 (only for ShopTurn)	Peripheral surface C: The positions refer to the reference point: Reference point Y or reference point angle polar - (only for single position) Reference point Z - (only for single position) Cylinder diameter \varnothing – (only for single position)	mm or degrees mm mm
C0 Y0 Z0 X0 (only for ShopTurn)	Peripheral surface Y: The positions refer to the reference point: Positioning angle for machining surface – (only for single position) Reference point Y – (only for single position) Reference point Z – (only for single position) Reference point X – (only for single position)	Degrees mm mm mm
N	Number of slots	
R	Radius of circumferential slot	mm
α_0	Starting angle	Degrees
α_1	Opening angle of the slot	Degrees

Parameter	Description	Unit
$\alpha 2$	Advance angle - (for pitch circle only)	Degrees
W	Slot width	mm
Z1 	Slot depth (abs) or depth relative to Z0 or X0 (inc) - (only for ∇ , $\nabla\nabla\nabla$)	mm
DZ	Maximum depth infeed - (only for ∇ , $\nabla\nabla\nabla$)	mm
UXY	Plane finishing allowance – (only for ∇ , $\nabla\nabla\nabla$)	mm
Positioning 	Positioning motion between the slots: <ul style="list-style-type: none"> • Straight line: Next position is approached linearly in rapid traverse. • Circular: Next position is approached along a circular path at the feedrate defined in a machine data code. 	
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

8.4.9 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 cutting

The following machining types are available to completely machine the slot:

- Roughing
- Rough-finishing
- Finishing
- Base finishing
- Edge finishing
- 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 cutting

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 plunge cutter, as it is known, ensures a longer service life due to less vibration for long overhangs.

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.

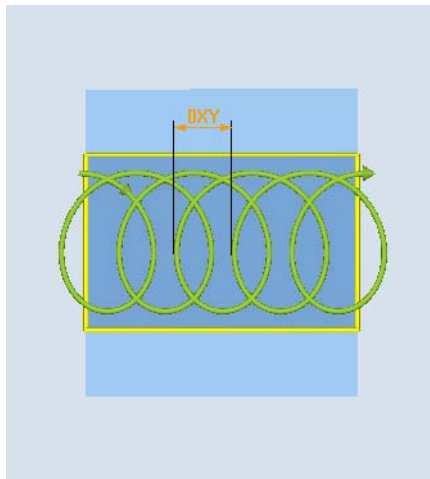
Approach/retraction for plunge cutting

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.

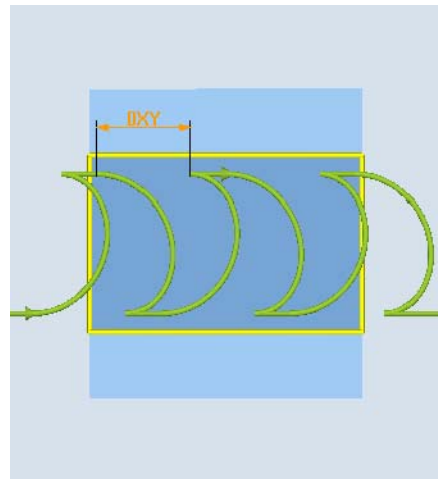
Machining type, roughing vortex milling

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.



Vortex milling: Climbing or conventional milling



Vortex milling: Climbing - conventional milling

Supplementary conditions for vortex milling

- Roughing
1/2 slot width W – finishing allowance $UXY \leq$ milling cutter diameter
- Slot width
minimum $1.15 \times$ milling cutter diameter + finishing allowance
maximum, $2 \times$ milling cutter diameter + $2 \times$ finishing allowance
- Radial infeed
minimum, $0.02 \times$ milling cutter diameter
maximum, $0.25 \times$ milling cutter diameter
- Maximum infeed depth \leq 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

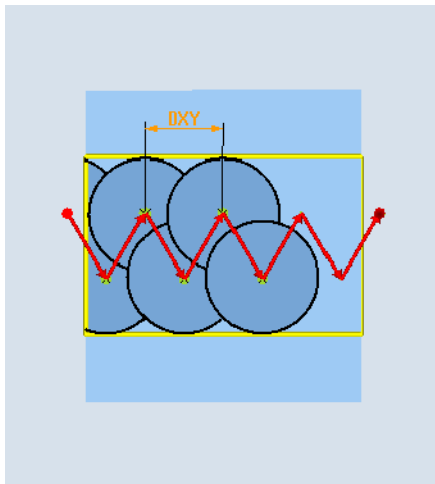
Roughing of the slot takes place sequentially along the length of the groove, with the milling cutter performing vertical insertions at the machining feedrate. The milling cutter is then retracted and repositioned at the next insertion point.

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 a 45° angle from the base 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 $UXY \leq$ 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.

8.4 Milling

- 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.

Additional supplementary conditions

- Finishing

$1/2 \text{ slot width } W \leq \text{milling cutter diameter}$

- Edge finishing

Finishing allowance $UXY \leq \text{milling cutter diameter}$

- Chamfering

The tip angle must be entered into the tool table.

Procedure




1. The part program or ShopTurn 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, G code program			Parameters, ShopTurn 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 surface (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y • Peripheral surface C • Peripheral surface Y 	
Position (only for ShopTurn)	<ul style="list-style-type: none"> • At the front (face) • At the rear (face) • Outside (peripheral surface) • Inside (peripheral surface) 	
 (only for ShopTurn)	Clamp/release spindle (only for end face Y/peripheral surface Y) The function must be set up by the machine manufacturer.	

8.4 Milling

Parameter	Description	Unit
Reference point U	Position of the reference point: <ul style="list-style-type: none">  (lefthand edge)  (center)  (righthand edge) 	
Machining U	<ul style="list-style-type: none"> ▽ (roughing) ▽▽ (pre-finishing) ▽▽▽ (finishing) ▽▽▽ base (base finishing) ▽▽▽ edge (edge finishing) Chamfering 	
Technology U	<ul style="list-style-type: none"> 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. 	
U	Milling direction: - (except plunge cutting). <ul style="list-style-type: none"> Climbing Conventional Climbing - conventional milling 	
Machining position U	<ul style="list-style-type: none"> 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). 	
X0 Y0 Z0 (only for G code)	The positions refer to the reference point: Reference point X – (only for single position) Reference point Y – (only for single position) Reference point Z – (only for single position)	mm mm mm
X0 or L0 U Y0 or C0 U Z0 (only for ShopTurn)	Face C: The positions refer to the reference point: Reference point X or reference point length polar – (only for single position) Reference point Y or reference point angle polar – (only for single position) Reference point Z – (only for single position)	mm mm mm
CP X0 or L0 U Y0 or C0 U Z0 (only for ShopTurn)	Face Y: The positions refer to the reference point: Positioning angle for machining area – (only single position) Reference point X or reference point length polar – (only for single position) Reference point Y or reference point angle polar - (only for single position) Reference point Z - (only for single position)	Degrees mm mm or degrees mm

Parameter	Description	Unit
Y0 or C0 	Peripheral surface C: The positions refer to the reference point: Reference point Y or reference point angle polar - (only for single position)	mm or degrees
Z0	Reference point Z - (only for single position)	mm
X0 (only for ShopTurn)	Cylinder diameter \varnothing - (only for single position)	mm
C0	Peripheral surface Y: The positions refer to the reference point: Positioning angle for machining surface - (only for single position)	Degrees
Y0	Reference point Y - (only for single position)	mm
Z0	Reference point Z - (only for single position)	mm
X0 (only for ShopTurn)	Reference point X - (only for single position)	mm
W	Slot width	mm
L	Slot length	mm
$\alpha 0$	Angle of rotation of slot	Degrees
Z1  (only for G code)	Slot depth (abs) or depth relative to Z0 (abs) - (only for ∇ , $\nabla\nabla$, $\nabla\nabla\nabla$ base and $\nabla\nabla$)	mm
Z1 or X1  (only for ShopTurn)	Slot depth (abs) or depth relative to Z0 or X0 (abs) - (only for ∇ , $\nabla\nabla$, $\nabla\nabla\nabla$ base and $\nabla\nabla$) (Z1 for machining surface, face C/Y or X1 for peripheral surface C/Y)	mm
DXY 	<ul style="list-style-type: none"> Maximum plane infeed Maximum plane infeed as a percentage of the milling cutter diameter - (only for ∇)	mm %
DZ	Maximum depth infeed - (only for ∇ , $\nabla\nabla$, $\nabla\nabla\nabla$ and $\nabla\nabla\nabla$ edge) - (only for vortex milling)	mm
UXY	Plane finishing allowance (slot edge) - (only for ∇ , $\nabla\nabla$ and $\nabla\nabla\nabla$ base)	mm
UZ	Depth finishing allowance (slot base) - (only for ∇ , $\nabla\nabla$ and $\nabla\nabla\nabla$ edge)	mm
FZ	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

8.4.10 Long hole (LONGHOLE) - only for G code program

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.




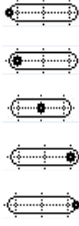


NOTICE
The cycle requires a milling cutter with a "face tooth cutting over center" (DIN 844).

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 to be executed 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 	<ul style="list-style-type: none"> • 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 	<ul style="list-style-type: none"> • 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). 	
X0	The positions refer to the reference point: 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
$\alpha 0$	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

8.4.11 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.

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.
3. Infeed to a starting point in the tool axis calculated internally in the controller with rapid traverse.
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. Retract on the thread center point and then to retraction plane in the tool axis in rapid traverse.



Please note that when milling an internal thread the tool must not exceed the following value:

Milling cutter diameter $< (\text{nominal diameter} - 2 \cdot \text{thread depth H1})$

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.
3. Infeed to a starting point in the tool axis calculated internally in the controller with rapid traverse.
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 $NT \geq 2$ 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 ShopTurn 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.



8.4 Milling

Table 8- 1

Parameters, G code program			Parameters, ShopTurn program		
PL 	Machining plane		T	Tool name	
	Milling direction		D	Cutting edge number	
RP	Retraction plane	mm	F 	Feedrate	mm/min mm/rev
SC	Safety clearance	mm	S / V 	Spindle speed or constant cutting rate	rpm m/min
F	Feedrate	mm/min			

Parameter	Description	Unit
Machining surface  (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y • Peripheral surface C • Peripheral surface Y 	
Position  (only for ShopTurn)	<ul style="list-style-type: none"> • At the front (face) • At the rear (face) • Outside (peripheral surface) • Inside (peripheral surface) 	
  (only for ShopTurn)	Clamp/release spindle (only for end face Y/peripheral surface Y) The function must be set up by the machine manufacturer.	
Machining	<ul style="list-style-type: none"> • ∇ (roughing) • ∇∇∇ (finishing) 	
	Machining direction: <ul style="list-style-type: none"> • Z0 → Z1 Machining from top to bottom • Z1 → Z0 Machining from bottom to top 	
	Direction of rotation of the thread: <ul style="list-style-type: none"> • Right-hand thread A right-hand thread is cut. • Left-hand thread A left-hand thread is cut. 	

Parameter	Description	Unit
	Position of the thread: <ul style="list-style-type: none"> • 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 tool cutting edge 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: <ul style="list-style-type: none"> • Single position • Position pattern (MCALL) 	
X0 Y0 Z0 (only for G code)	The positions refer to the center point: Reference point X – (only for single position) Reference point Y – (only for single position) Reference point Z	mm mm mm
Z1 	End point of the thread (abs) or thread length (inc)	mm
Table 	Thread table selection: <ul style="list-style-type: none"> • Without • ISO metric • Whitworth BSW • Whitworth BSP • UNC 	
Selection - (not for table "without")	Selection, table value: e.g. <ul style="list-style-type: none"> • M3; M10; etc. (ISO metric) • W3/4"; etc. (Whitworth BSW) • G3/4"; etc. (Whitworth BSP) • N1" - 8 UNC; etc. (UNC) 	
P	Display of the thread pitch for the parameter input in the input field "Table" and "Selection".	MODULUS turns/" mm/rev in/rev

Parameter	Description	Unit
P  - (selection option only for table selection "without")	Pitch ... <ul style="list-style-type: none"> In MODULUS: For example, generally used for worm gears that mesh with a gear wheel. 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. <ul style="list-style-type: none"> In mm/rev In inch/rev The tool used depends on the thread pitch.	MODULUS Turns/" mm/rev in/rev
∅	Nominal diameter Example: Nominal diameter of M12 = 12 mm	mm
H1	Thread depth	mm
DXY	Maximum plane infeed	mm
rev	Finishing allowance in X and Y - (only for ▽)	mm
αS	Starting angle	Degrees

8.4.12 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.

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

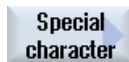


1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Milling" softkey.



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.



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> .



7. Press the "Variable" and "Time" softkeys if you want to engrave the current time.



The time is inserted in the European format (<TIME24>).

To have the time in the American format, change the format to <TIME12>.

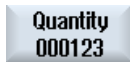
Example:

Text entry: Time: <TIME24> Execute: Time: 16.35

Time: <TIME12> Execute: Time: 04.35 PM



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.

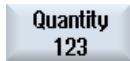


- Define the number of digits by adjusting the number of place holders (#) in the engraving field.
- If the specified number of positions (e.g. ##) is not sufficient to represent the unit quantity, then the cycle automatically increases the number of positions.

- OR



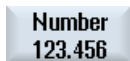
7. • Press the "Variable" and "Workpiece count 123" softkeys if you want to engrave a workpiece count without leading zeroes.
- The format text <#,_\$AC_ACTUAL_PARTS> is inserted and you return to the engraving field with the softkey bar.



- 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.



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.



- 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.

Input	Output	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	Places before and after the decimal point 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.

You can optionally use blanks to format before the decimal place.

Instead of _VAR_NUM you can use any other numeric 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



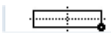
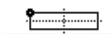

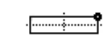



You can engrave the text mirrored on the workpiece.







Full circle

If you want to distribute the characters evenly around a full circle, enter the arc angle $\alpha_2=360^\circ$. The cycle then distributes the characters evenly around the full circle.

Parameters, G code program			Parameters, ShopTurn program		
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	mm/min			

Parameter	Description	Unit
FZ (only for G code)	Depth infeed rate	*
FZ  (only for ShopTurn)	Depth infeed rate	mm/min mm/tooth
Machining surface  (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y • Peripheral surface C • Peripheral surface Y 	
Position  (only for ShopTurn)	<ul style="list-style-type: none"> • At the front (face) • At the rear (face) • Outside (peripheral surface) • Inside (peripheral surface) 	
  (only for ShopTurn)	Clamp/release spindle (only for end face Y/peripheral surface Y) The function must be set up by the machine manufacturer.	
Alignment 	<ul style="list-style-type: none"> •  (linear alignment) •  (curved alignment) •  (curved alignment) 	

Parameter	Description	Unit
Reference point U	Position of the reference point <ul style="list-style-type: none"> •  bottom left •  bottom center •  bottom right •  top left •  top center •  top right •  left-hand edge •  center •  right-hand edge 	
Mirror writing	<ul style="list-style-type: none"> • 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 U Y0 or a0 U Z0 (only for G code)	The positions refer to the reference point: Reference point X or reference point length polar Reference point Y or reference point angle polar Reference point Z	mm mm or degrees mm
X0 or L0 U Y0 or C0 U Z0 (only ShopTurn)	Face C: The positions refer to the reference point: Reference point X or reference point length polar Reference point Y or reference point angle polar Reference point Z	mm mm or degrees mm
CP X0 or L0 U Y0 or C0 U Z0 (only ShopTurn)	Face Y: The positions refer to the reference point: Positioning angle for machining area Reference point X or reference point length polar Reference point Y or reference point angle polar Reference point Z	Degrees mm mm or degrees mm
Y0 or C0 U Z0 X0 (only ShopTurn)	Peripheral surface C: The positions refer to the reference point: Reference point Y or reference point angle polar – (only for single position) Reference point Z Cylinder diameter \varnothing	mm or degrees mm mm

Parameter	Description	Unit
C0	Peripheral surface Y: The positions refer to the reference point: Positioning angle for machining surface – (only for single position)	Degrees
Y0		mm
Z0		mm
X0 (only ShopTurn)		mm
Z1 	Engraving depth (abs) or referenced depth (inc)	mm
W	Character height	mm
DX1 or $\alpha 2$ 	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
$\alpha 1$	Text direction (for linear alignment only)	Degrees
XM or LM  (only G code)	Center point X (abs) or center point length polar – (for curved alignment only)	mm
YM or αM  (only G code)	Center point Y (abs) or center point angle polar – (for curved alignment only)	mm
YM or CM  (only ShopTurn)	Center point Y or C (abs.) – (for curved alignment only) - (only for machining surface, peripheral surface C/Y)	mm or degrees
ZM (only ShopTurn)	Center point Z (abs.) – (for curved alignment only) - (only for machining surface, peripheral surface C/Y)	mm

* Unit of feedrate as programmed before the cycle call

8.5 Contour milling

8.5.1 General information

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.

8.5.2 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.






ShopTurn 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 in 90° grid
Straight line down		Straight line in 90° grid
Straight line left		Straight line in 90° grid

Contour element	Symbol	Meaning
Straight line right		Straight line in 90° grid
Straight line in any direction		Straight line with any gradient
Arc right		Circle
Arc left		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 active 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)

Graphic 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.

8.5.3 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 ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Milling" softkey.

3. Press the "Contour milling" and "New contour" softkeys. The "New Contour" input window opens.


4. Enter a contour name.
5. 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
Machining surface (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y • Face B • Peripheral surface C • Peripheral surface Y 	
PL (only for G code)	Machining plane <ul style="list-style-type: none"> • G17 (XY) • G19 (YZ) 	
ϕ (only ShopTurn)	Cylinder diameter (only peripheral surface C)	mm
G17 or face C/Y/B	G19 or peripheral surface C/Y	
X Y	Cartesian: Starting point X or Y (abs) Starting point Y or Z (abs)	mm mm
X Y	Polar: Position pole (abs) Position pole (abs)	mm mm

8.5 Contour milling

Parameter	Description	Unit
Starting point		
L1	Distance to pole, end point (abs)	mm
φ1	Polar angle to the pole, end point (abs)	Degrees
Additional commands	<p>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.</p> <p>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.</p> <p>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 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.</p> <p>Note: The additional commands are only effective for path milling!</p>	

8.5.4 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 $X\alpha$, $Y\alpha$ and $I\alpha$, $J\alpha$.



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.












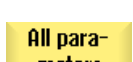
- Dialog box selection

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 or changing contour elements

- | | |
|---|---|
|  | <ol style="list-style-type: none"> The part program or ShopTurn program to be executed is created. Select the file type (MPF or SPF), enter the desired name of the program and press the "OK" softkey or the "Input" key.
This editor is opened. |
|  | |
|  | <ol style="list-style-type: none"> Select a contour element via softkey.
The input window "Straight (e.g. X)" opens. |
|  | <p>- OR</p> <p>The input window "Straight (e.g. Y)" opens.</p> |
|  | <p>- OR</p> <p>The input window "Straight (e.g. XY)" opens.</p> |
|  | <p>- OR</p> <p>The "Circle" input window opens.</p> |
|  | <p>- OR</p> <p>The "Pole Input" input window opens.</p> |
|  | |
|  | <ol style="list-style-type: none"> 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.). Press the "Accept" softkey.
The contour element is added to the contour. |
|  | <ol style="list-style-type: none"> 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 angle to the preceding element α_2 is set to 0°. The "tangential" selection appears in the parameter input field. |
|  | <ol style="list-style-type: none"> Repeat the procedure until the contour is complete. Press the "Accept" softkey.
The programmed contour is transferred to the machining plan (program view). |
|  | <ol style="list-style-type: none"> 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"

Parameters	Description	Unit
Machining surface  (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y • Face B • Peripheral surface C • Peripheral surface Y 	
X 	End point X (abs or inc)	mm
$\alpha 1$	Starting angle e.g. to the X axis	Degrees
$\alpha 2$	Angle to the preceding element	Degrees
Transition to next element 	Type of transition <ul style="list-style-type: none"> • 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"


Parameters	Description	Unit
Machining surface  (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y • Face B • Peripheral surface C • Peripheral surface Y 	
Y 	End point Y (abs or inc)	mm
$\alpha 1$	Starting angle to X axis	Degrees
Transition to next element 	Type of transition <ul style="list-style-type: none"> • 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. XY"


Parameters	Description	Unit
Machining surface U (only for ShopTurn)	<ul style="list-style-type: none"> Face C Face Y Face B Peripheral surface C Peripheral surface Y 	
X U	End point X (abs or inc)	mm
Y U	End point Y (abs or inc)	mm
L	Length	mm
α_1	Starting angle e.g. to the X axis	Degrees
α_2	Angle to the preceding element	Degrees
Transition to next element U	Type of transition <ul style="list-style-type: none"> 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"

Parameters	Description	Unit
Machining surface U (only for ShopTurn)	<ul style="list-style-type: none"> Face C Face Y Face B Peripheral surface C Peripheral surface Y 	
Direction of rotation U	<ul style="list-style-type: none"> Clockwise direction of rotation  Counterclockwise direction of rotation  	
R	Radius	mm
e.g. X U	End point X (abs or inc)	mm
e.g. Y U	End point Y (abs or inc)	mm
e.g. I U	Circle center point I (abs or inc)	mm
e.g. J U	Circle center point J (abs or inc)	mm
α_1	Starting angle to X axis	Degrees
α_2	Angle to the preceding element	Degrees

Parameters	Description	Unit
$\beta 1$	End angle to Z axis	Degrees
$\beta 2$	Opening angle	Degrees
Transition to next element 	Type of transition <ul style="list-style-type: none"> • 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"

Parameters	Description	Unit
Machining surface  (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y • Face B • Peripheral surface C • Peripheral surface Y 	
X	Position pole (abs)	mm (in)
Y	Position pole (abs)	Degrees

Contour element "End"

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

The values cannot be edited.

8.5.5 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 ShopTurn 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 ShopTurn 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.



8.5.6 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 part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Milling" and "Contour milling" 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 	<ul style="list-style-type: none"> • Contour name • Labels • Subprogram • Labels in the subprogram 	
Contour name	CON: Contour name	
Labels	<ul style="list-style-type: none"> • LAB1: Label 1 • LAB2: Label 2 	
Subprogram	PRG: Subprogram	
Labels in the subprogram	<ul style="list-style-type: none"> • PRG: Subprogram • LAB1: Label 1 • LAB2: Label 2 	

8.5.7 Path milling (CYCLE72)

Function

You can machine open or closed contours with the "Path milling" cycle. Before you can mill the contour, you must enter the contour. Machining can be performed in either direction, i.e. in the direction of the programmed contour or in the opposite direction.

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.

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.

See also

Clamping the spindle (Page 234)

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.

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.

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.

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.

Machining type

You can select the machining mode (roughing, finishing, or chamfer) for path 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.

It is also possible to choose between machining the contour with a cutter radius offset or traversing on the center-point path.

Slot side compensation

When you mill a contour on the peripheral surface (peripheral machining surface C), you can work with or without a slot wall compensation.

- Slot side compensation off
ShopTurn creates slots with parallel walls when the tool diameter is equal to the slot width. If the slot width is larger than the tool diameter, the slot walls will not be parallel.
- Slot side compensation on
ShopTurn creates slots with parallel walls also when the slot width is larger than the tool diameter. If you want to work with a slot wall compensation, you must not program the contour of the slot, but instead the imagined center path of a bolt inserted in the slot whereby the bolt touches both walls. Parameter D is used to specify the slot width.








Procedure












1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Milling" softkey.
3. Press the "Contour milling" and "Path milling" softkeys. The "Path Milling" input window opens.

Parameters, G code program			Parameters, ShopTurn 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 surface (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y • Peripheral surface C • Peripheral surface Y 	
Position (only for ShopTurn)	<ul style="list-style-type: none"> • At the front (face) • At the rear (face) • Outside (peripheral surface) • Inside (peripheral surface) 	
 (only for ShopTurn)	Clamp/release spindle (only for end face Y/peripheral surface Y) The function must be set up by the machine manufacturer.	
Machining 	<ul style="list-style-type: none"> • ∇ (roughing) • ∇∇∇ (finishing) • Chamfering 	

Parameter	Description	Unit
Machining direction 	Machining in the programmed contour direction <ul style="list-style-type: none"> Forward: Machining is performed in the programmed contour direction Backward: Machining is performed in the opposite direction to the programmed contour 	
Radius compensation 	<ul style="list-style-type: none"> Left (machining to the left of the contour)  Right (machining to the right of the contour)  off  <p>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.</p>	
Slot side compensation  (only ShopTurn)	Slot side compensation on or off (only for machining surface, peripheral surface C)	
D	Offset to programmed path - (only for slot side compensation on)	
CP	Positioning angle for machining area - (only for ShopTurn, machining surface, face Y)	Degrees
C0	Positioning angle for machining surface - (only for ShopTurn, machining surface, peripheral surface Y)	Degrees
Z0	Reference point Z	mm
Z1	Final drilling depth (abs) or final drilling depth referred to Z0 or X0 (inc)	mm
DZ	Maximum depth infeed - (only for machining ▽ and ▽▽▽)	mm
UZ	Depth finishing allowance - (only for machining ▽)	mm
UXY	Finishing allowance, plane	mm
Approach 	Planar approach mode: <ul style="list-style-type: none"> 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) Straight line: Slope in space Perpendicular: Perpendicular to the path (only with path milling on the center-point path) 	

8.5 Contour milling

Parameter	Description	Unit
Approach strategy 	<ul style="list-style-type: none"> axis-by-axis - (only for "quadrant, semi-circle or straight line" approach)  <ul style="list-style-type: none"> 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
FZ (only for G code)	Depth infeed rate	*
FZ (only for ShopTurn)	Depth infeed rate	mm/min mm/tooth
Retraction 	Planar retraction mode: <ul style="list-style-type: none"> 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) Straight line: 	
Retraction strategy 	<ul style="list-style-type: none"> axis-by-axis  <ul style="list-style-type: none"> spatial 	
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 <ul style="list-style-type: none"> No retraction to RP Z0 + safety clearance By the safety clearance 	
FS	Chamfer width for chamfering - (only for chamfering machining)	mm
ZFS 	Insertion depth of tool tip (abs or inc) - (for chamfering machining only)	mm

* Unit of feedrate as programmed before the cycle call

8.5.8 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.

Automatically calculating/Manually entering 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, which 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



Software option

For solid machining residual material, you require the option "residual material detection and machining".

Name convention

For multi-channel systems, cycles attach a "_C" and a two-digit number of the specific channel to the names of the programs to be generated, e.g. for channel 1 "_C01". This is the reason that 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 for 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.

8.5.9 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.

To prevent the drill slipping during drilling, you can center it first.

Before you predrill the pocket, you must enter the pocket contour. If you want to center before predrilling, you have to program the two machining steps in separate blocks.

The number and positions of the necessary predrilled holes depend on the specific circumstances (such as shape of contour, tool, plane infeed, finishing allowance) and are calculated by the cycle.

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. 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.

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.

See also





Clamping the spindle (Page 234)




Procedure when centering




1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Milling", "Mill contour", "Predrilling" and "Centering" softkeys. The "Centering" input window opens.

8.5 Contour milling

Parameters, G code program			Parameters, ShopTurn program		
PRG	Name of the program to be generated		T	Tool name	
PL	Machining plane		D	Cutting edge number	
					
Milling direction 	<ul style="list-style-type: none"> • Climbing • Conventional 		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 "solid machining" machining step. This is used to determine the plunge position.	
Machining surface  (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • End face Y (only when Y axis exists) • Face B • Peripheral surface C • Peripheral surface Y (only when Y axis exists) 	
  (only for ShopTurn)	Clamp/release spindle (only for end face Y/B and peripheral surface Y) The function must be set up by the machine manufacturer.	
Z0	Reference point in the tool axis Z	mm
Z1	Pocket depth \varnothing (abs) or depth referred to Z0	mm
CP	Positioning angle for machining area - (only for ShopTurn, machining surface, face Y)	Degrees
C0	Positioning angle for machining surface - (only for ShopTurn, machining surface, peripheral surface Y)	Degrees
DXY	<ul style="list-style-type: none"> • Maximum plane infeed • Maximum plane infeed as a percentage of the milling cutter diameter 	mm %

Parameter	Description	Unit
UXY	Finishing allowance, plane	mm
Lift mode 	Lift mode before new infeed If the machining operation requires several points of insertion, the retraction height can be programmed: <ul style="list-style-type: none"> To retraction plane Z0 + safety clearance When making the transition to the next insertion point, the tool returns to this height. 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

- The part program or ShopTurn program to be processed has been created and you are in the editor.
- Press the "Milling", "Contour milling", "Predrilling" and "Predrilling" softkeys.
The "Predrilling" input window opens.



Parameters, G code program			Parameters, ShopTurn program		
PRG	Name of the program to be generated		T	Tool name	
PL 	Machining plane		D	Cutting edge number	
Milling direction 	<ul style="list-style-type: none"> Climbing Conventional 		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			

8.5 Contour milling

Parameter	Description	Unit
TR	Reference tool Tool, which is used in the "solid machining" machining step. This is used to determine the plunge position.	
Machining surface  (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y • Face B • Peripheral surface C • Peripheral surface Y 	
  (only for ShopTurn)	Clamp/release spindle (only for end face Y/B and peripheral surface Y) The function must be set up by the machine manufacturer.	
Z0	Reference point in the tool axis Z	mm
Z1	Pocket depth (abs) or depth referred to Z0 or X0 (inc)	mm
CP	Positioning angle for machining area - (only for ShopTurn, machining surface, face Y)	Degrees
C0	Positioning angle for machining surface - (only for ShopTurn, machining surface, peripheral surface Y)	Degrees
DXY	<ul style="list-style-type: none"> • 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 	<p>Lift mode before new infeed</p> <p>If the machining operation requires several points of insertion, the retraction height can be programmed:</p> <ul style="list-style-type: none"> • To retraction plane • Z0 + safety clearance <p>When making the transition to the next insertion point, the tool returns to this height. 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.</p>	mm mm

8.5.10 Milling contour pocket (CYCLE63)

Function

You can use the "Mill pocket" function to mill a pocket on the face or peripheral surface.

Before you solid machine the pocket, you must first enter the contour of the pocket and, if applicable, the contour of an island. Stock is removed from the pocket parallel to the contour from the inside to the outside. The direction is determined by the machining direction (climbing or conventional). If an island is located in the pocket, the cycle automatically takes this into account during stock removal.

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.

See also

Clamping the spindle (Page 234)

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. If this occurs, please reduce the angle of insertion.





Procedure




1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Milling", "Contour milling" and "Pocket" softkeys.
The "Mill pocket" input window opens.




8.5 Contour milling

Parameters, G code program			Parameters, ShopTurn program		
PRG	Name of the program to be generated		T	Tool name	
PL	Machining plane		D	Cutting edge number	
			F	Feedrate	mm/min mm/tooth
Milling direction 	<ul style="list-style-type: none"> • Climbing • Conventional 			S / V	Spindle speed or constant cutting rate
RP	Retraction plane	mm			rpm m/min
SC	Safety clearance	mm			
F	Feedrate	mm/min			

Parameter	Description	Unit
Machining surface  (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y • Face B • Peripheral surface C • Peripheral surface Y 	
  (only for ShopTurn)	Clamp/release spindle (only for end face Y/B and peripheral surface Y) The function must be set up by the machine manufacturer.	
Machining 	The following machining technologies can be selected: <ul style="list-style-type: none"> • ▽ (roughing) • ▽▽ base (base finishing) • ▽▽ edge (edge finishing) • Chamfering 	
Z0	Reference point in the tool axis Z	mm
Z1	Pocket depth (abs) or depth referred to Z0	mm
CP	Positioning angle for machining area - (only for ShopTurn, machining surface, face Y)	Degrees
C0	Positioning angle for machining surface - (only for ShopTurn, machining surface, peripheral surface Y)	Degrees
DXY	<ul style="list-style-type: none"> • 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

Parameter	Description	Unit
Starting point	<ul style="list-style-type: none"> Manual Starting point is entered Automatic Starting point is automatically calculated 	
XS	Starting point X - (only for "manual" starting point)	mm
YS	Starting point Y - (only for "manual" starting point)	mm
Insertion 	<p>The following insertion modes can be selected – (only for ∇, ∇∇∇ base or ∇∇∇ edge):</p> <ul style="list-style-type: none"> 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 Insertion along a 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 remove the slope caused by insertion. Oscillating insertion Oscillating insertion at the center axis of the 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 remove the slope caused by insertion. 	
FZ  (only for ShopTurn)	Depth infeed rate – (for vertical insertion only)	mm/min mm/tooth
FZ (only for G code)	Depth infeed rate – (for vertical insertion only)	mm/min
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 milling cutter radius; otherwise, material will remain.	mm
EW	Note: 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. If this occurs, please reduce the angle of insertion.	Degrees
Lift mode 	<p>Lift mode before new infeed</p> <p>If the machining operation requires several points of insertion, the retraction height can be programmed:</p> <ul style="list-style-type: none"> To retraction plane Z0 + safety clearance <p>When making the transition to the next insertion point, the tool returns to this height. 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.</p>	mm mm

Parameter	Description	Unit
FS	Chamfer width for chamfering - (only for chamfering machining)	mm
ZFS 	Insertion depth of tool tip (abs or inc) - (for chamfering machining only)	mm

8.5.11 Contour pocket residual material (CYCLE63, option)

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. Material that remains as part of the finishing allowance is not residual material.

The residual material is calculated on the basis of the milling cutter used for stock removal.

If you mill several pockets and want to avoid unnecessary tool changeover, 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 ShopTurn program, additionally appears when you press the "All parameters" softkey. Then program as follows:

1. Contour pocket 1
2. Solid machining
3. Contour pocket 2
4. Solid machining
5. Contour pocket 1
6. Solid machine residual material
7. Contour pocket 2
8. Solid machine residual material



Software option

For solid machining residual material, you require the option "residual material detection and machining".

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.

See also

Clamping the spindle (Page 234)

Procedure

1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Milling", "Contour milling" and "Pocket resid. mat." softkeys.








The "Pocket Res. Mat." input window opens.



3. For the ShopTurn program, press the "All parameters" softkey if you want to enter additional parameters.

Parameters, G code program			Parameters, ShopTurn program		
PRG	Name of the program to be generated		T	Tool name	
PL	Machining plane		D	Cutting edge number	
	Milling direction	<ul style="list-style-type: none"> • Climbing • Conventional 	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			

8.5 Contour milling

Parameter	Description	Unit
Machining surface  (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y • Face B • Peripheral surface C • Peripheral surface Y 	
  (only for ShopTurn)	Clamp/release spindle (only for end face Y/B and peripheral surface Y) The function must be set up by the machine manufacturer.	
TR	Reference tool Tool, which is used in the "solid machining" 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 or X0 (inc)	mm
CP	Positioning angle for machining area - (only for ShopTurn, machining surface, face Y)	Degrees
C0	Positioning angle for machining surface - (only for ShopTurn, machining surface, peripheral surface Y)	Degrees
DXY	<ul style="list-style-type: none"> • Maximum plane infeed • Maximum plane infeed as a percentage of the milling cutter diameter 	mm %
DZ	Maximum depth infeed	
Lift mode 	<p>Lift mode before new infeed</p> <p>If the machining operation requires several points of insertion, the retraction height can be programmed:</p> <ul style="list-style-type: none"> • To retraction plane • Z0 + safety clearance <p>When making the transition to the next insertion point, the tool returns to this height. 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.</p>	mm mm

8.5.12 Milling contour spigot (CYCLE63)

Function

You can use the "Mill spigot" function to mill any spigots on the face or peripheral surface.

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. there, the tool moves with rapid traverse. Material is then removed between the blank contour and spigot contour.

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.

See also

Clamping the spindle (Page 234)

Machining type

You can select the machining type (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 at rapid traverse at the height of the retraction plane and is fed in 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 (climb/conventional) (see "Changing program settings").
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 retracts to the safety clearance at rapid traverse.

Procedure




1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Milling", "Contour milling" and "Spigot" softkeys.
The "Mill spigot" input window opens.



3. Select the "Roughing" machining type.

Parameters, G code program			Parameters, ShopTurn program		
PRG	Name of the program to be generated		T	Tool name	
PL	Machining plane		D	Cutting edge number	
	Milling direction	<ul style="list-style-type: none"> • Climbing • Conventional 	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 surface (only for ShopTurn)	<ul style="list-style-type: none"> • Face C • Face Y • Face B • Peripheral surface C • Peripheral surface Y 	
(only for ShopTurn)	Clamp/release spindle (only for end face Y/B and peripheral surface Y) The function must be set up by the machine manufacturer.	
Machining 	The following machining technologies can be selected: <ul style="list-style-type: none"> • ▽ (roughing) • ▽▽ base (base finishing) • ▽▽ edge (edge finishing) • Chamfering 	

Parameter	Description	Unit
Z0	Reference point in tool axis Z	mm
Z1	Pocket depth (abs) or depth referred to Z0 or X0 (inc)	mm
CP	Positioning angle for machining area - (only for ShopTurn, machining surface, face Y)	Degrees
C0	Positioning angle for machining surface - (only for ShopTurn, machining surface, peripheral surface Y)	Degrees
 DXY	<ul style="list-style-type: none"> 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	Finishing allowance, plane - (only for ▽, ▽▽▽ base or ▽▽▽ edge)	mm
UZ	Finishing allowance, depth - (only for ▽ or ▽▽▽ base)	mm
Lift mode 	<p>Lift mode before new infeed</p> <p>If the machining operation requires several points of insertion, the retraction height can be programmed:</p> <ul style="list-style-type: none"> To retraction plane Z0 + safety clearance <p>When making the transition to the next insertion point, the tool returns to this height. 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.</p>	mm mm mm
FS	Chamfer width for chamfering - (only for chamfering machining)	mm
ZFS 	Insertion depth of tool tip (abs or inc) - (for chamfering machining only)	mm

8.5.13 Contour spigot residual material (CYCLE63, option)

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. Material that remains as part of the finishing allowance is not residual material.

The residual material is calculated on the basis of the milling cutter used for clearing.

If you mill several spigots and want to avoid unnecessary tool changeover, 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 ShopTurn program, additionally appears when you press the "All parameters" softkey. Then program 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 solid machining residual material, you require the option "residual material detection and machining".

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.

See also

Clamping the spindle (Page 234)

Procedure

1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Milling", "Contour milling" and "Spigot resid. mat." softkeys.










The "Spigot Res. Mat." input window opens.

3. For the ShopTurn program, press the "All parameters" softkey if you want to enter additional parameters.

Parameters, G code program			Parameters, ShopTurn program		
PRG	Name of the program to be generated		T	Tool name	
PL	Machining plane		D	Cutting edge number	
	Milling direction	<ul style="list-style-type: none"> • Climbing • Conventional 	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			

8.5 Contour milling

Parameter	Description	Unit
Machining	The following machining technologies can be selected: <ul style="list-style-type: none"> ▽ (roughing) 	
Machining surface  (only for ShopTurn)	<ul style="list-style-type: none"> Face C Face Y Face B Peripheral surface C Peripheral surface Y 	
  (only for ShopTurn)	Clamp/release spindle (only for end face Y/B and peripheral surface Y) The function must be set up by the machine manufacturer.	
TR	Reference tool Tool, which is used in the "solid machining" machining step. This is used to determine the residual corners.	
D	Cutting edge number	
Z0	Reference point in tool axes Z	mm
 Z1	Pocket depth (abs) or depth referred to Z0	mm
CP	Positioning angle for machining area - (only for ShopTurn, machining surface, face Y)	Degrees
C0	Positioning angle for machining surface - (only for ShopTurn, machining surface, peripheral surface Y)	Degrees
 DXY	<ul style="list-style-type: none"> Maximum plane infeed Maximum plane infeed as a percentage of the milling cutter diameter 	mm %
DZ	Maximum depth infeed	
Lift mode 	Lift mode before new infeed If the machining operation requires several points of insertion, the retraction height can be programmed: <ul style="list-style-type: none"> To retraction plane Z0 + safety clearance When making the transition to the next insertion point, the tool returns to this height. 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 chamfering machining only)	mm

8.6 Further cycles and functions

8.6.1 Swiveling plane / aligning 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", "Align milling tool" and "Align turning tool" functions. When setting and aligning, contrary to swiveling, 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,"",0,57,0,0,0,0,0,0,0,0,0,0,0,0,1,0,1)) ;swivel ZERO to
;initial position of the
;machine kinematics
N5 WORKPIECE(,,,,,"BOX",0,0,50,0,0,0,100,100) ;blank declaration for
;simulation and
;simultaneous recording

```

For machines where swivel is set-up, each main program with a swivel should start in the initial position 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 ShopTurn 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

The purpose of the "Align turning tool" function is to support turning machines with a swivel-mounted B axis. The position and orientation of the turning tool can be changed by rotating swivel axis B (around Y) and the tool spindle.

In contrast to "Swivel plane", no rotation is operative in the active work offsets in the workpiece coordinate system in the case of "Align tool".

The maximum angular range for "Align milling tool" is limited by the traversing range of the participating rotary axes. Technological limits are also placed on the angular range depending on the tool used.

When aligning the tool, using the CUTMOD NC command, the tool data are calculated online based on the tool orientation (positions of the B axis and the tool spindle). For a turning tool, this involves the cutting edge position, the holder angle and the cut direction.

Name of swivel data set

Selecting the swivel data set or deselecting the swivel data set.

The selection can be hidden by the machine data.

For "Swivel plane" and "Swivel tool" / "Set tool", only the swivel data sets are available for selection where no B axis kinematics, turning technology has been set.

"Swivel tool" / "Align tool", only the swivel data sets are available for selection where B axis kinematics, turning technology has been set.



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 you can 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 refer to the machine manufacturer's specifications.



WARNING

Risk of collision

You must select a retraction position that avoids a collision between the tool and workpiece when swiveling.

Swivel plane (only for G code programming)

- **New**

Previous swivel frames and programmed frames are deleted and 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, 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 several swivel cycles are programmed in a program and programmable frames are also active between them (e.g., AROT ATRANS), these are taken into account in the swivel frame.

If the currently active WO 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 refer to the machine manufacturer's specifications.

- **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 projection angles around XY and YX are programmed, the new X-axis of the swiveled coordinate system lies in the old ZX plane.

When projection angles around XZ and ZX are programmed, the new Z-axis of the swiveled coordinate system lies in the old Y-Z plane.

When projection angles around YZ and ZY are programmed, 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

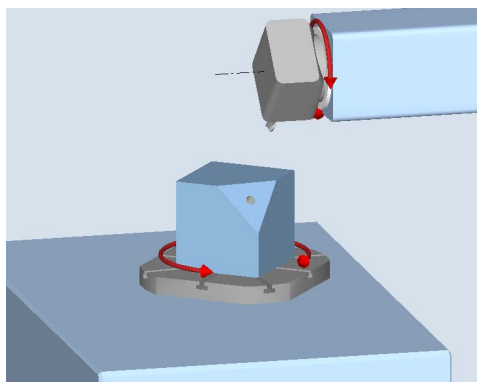
Please refer to the machine manufacturer's specifications.

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:

- Machine kinematics with swivel head and swivel table.
 - Swivel head with rotary axis 1 (B) rotates around machine axis Y.
- Angular traversing range of rotary axis B from -90 to +90 degrees.
- Swivel table with rotary axis 2 (C) rotates around machine axis Z.
- Angle traversing 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.

The machine in the basic setting (pole setting) of the kinematics (B = 0 C = 0) is shown in the following diagram.



- Direction "-" (minus)
 - Rotary axis B moves to -10 degrees in the negative direction (red arrow).
 - Rotary axis C moves to 90 degrees (rotation around X!).
- Direction "+" (plus)
 - Rotary axis B moves to +10 degrees in the positive direction (red arrow).
 - Rotary axis C moves to 270 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 C).

Tool

To avoid collisions, you can use the 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 during swiveling.



Machine manufacturer

Please refer to the machine manufacturer's specifications.




Procedure



1. The part program or ShopTurn 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.
This is done, for example, to swivel the coordinate system back to its original orientation.

Parameters, G code program			Parameters, ShopTurn program		
PL U	Machining plane		T	Tool name	
			D	Cutting edge number	
			U	Feedrate	mm/min mm/rev
			S / V U	Spindle speed or constant cutting rate	rpm m/min

Parameter	Description	Unit
TC U	Name of swivel data set	
Retract U- (only for G code)	<ul style="list-style-type: none"> No: No retraction before swiveling Z: Retraction in the direction of machine axis Z Z, X, Y: Move machining axis to retraction position before swiveling Tool direction, max.: Maximum retraction (up to the software end position) in the tool direction Tool direction, inc.: Retraction, incremental (up to the software end position) in the tool direction <p>When retracting in the tool direction, in the swiveled machine state, several axes can move (travel).</p>	
ZR	Retraction path - (only for incremental retraction in the tool direction)	mm
Swivel plane U- (only for G code)	<ul style="list-style-type: none"> New: New swivel plane Additive: Additive swivel plane 	
RP - (only for ShopTurn)	Retraction plane for face B	
C0 - (only for ShopTurn)	Position angle for machining surface	Degrees
X0	Reference point for rotation X	
Y0	Reference point for rotation Y	
Z0	Reference point for rotation Z	
Swivel mode U	<ul style="list-style-type: none"> axis-by-axis: Swivel coordinate system axis-by-axis Solid angle: Swivel via solid angle Proj. angle: Swiveling via projection angle Direct: Position rotary axes directly 	
Axis sequence U	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	
Z	Rotation around Z	

Parameter	Description	Unit
Projection position 	Position of the projection in space - (only for swivel mode, projection angle) X α , Y α , Z β or Y α , Z α , Z β or Z α , X α , Z β	
X α	Projection angle	- (only for projection position)
Y α	Projection angle	
Z β	Angle of rotation in the plane	
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  - (only for G code)	Preferred direction, rotary axis 1 - (not for swivel mode direct) <ul style="list-style-type: none"> • + • - 	
Tool  - (only for G code)	Tool tip when swiveling <ul style="list-style-type: none"> • Follow up The position of the tool tip is maintained during swiveling. • No follow up The position of the tool tip is not maintained during swiveling. 	

8.6.2 Swiveling tool (CYCLE800)

8.6.2.1 Aligning turning tools - only for G code program (CYCLE800)

Function

The purpose of the "Align milling tool" or "Align turning tool" function is to support combined milling machines - lathes with a B axis that can be swiveled.

This functionality is specifically addressing certain configurations of milling machines, which have been expanded by turning functionality. Tool orientation in the turning mode is realized using a swivel axis B (swiveling around Y) with an associated milling spindle (e.g. C1). It is designed to be compatible with both turning and milling tools.

In contrast to "Swivel plane", no rotation is operative in the active zero offsets in the workpiece coordinate system in the case of "Align tool". Only the offsets calculated by the NC and the corresponding tool orientation are effective.

The maximum angular range for "Align tool" is + 360 degrees or it is limited by the traversing range of the participating rotary axes. Technological limits are also placed on the angular range depending on the tool used. When aligning the tool, the data of the tool is calculated based on the tool orientation using the CUTMOD NC command. For a turning tool, this involves the cutting edge position, the holder angle and the cut direction.

The use of the "Align milling tool" function is restricted to milling operations in parallel with the axis (face, peripheral machining) at a machine with a B axis that can be swiveled. If milling is to be possible on any swiveled machining plane, then the "swivel plane" function must be used.



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. Select the "Miscellaneous" softkey.
3. Press the "Swivel tool" and "Align turning tool" softkeys. The "Align turning tool" input window opens.

Parameters	Description	Unit
TC	Name of the swivel data set	
Retract	<ul style="list-style-type: none"> • No: No retraction before swiveling • Z: Retraction in the direction of machine axis Z • Tool direction, max.: Maximum retraction in tool direction • Tool direction, inc.: Incremental retraction in tool direction 	
ZR	Retraction path - (only for incremental retraction in the tool direction)	
β	Rotation around the 3rd geometry axis (for G18 Y)	Degrees
γ	Rotation around the turning tool	Degrees
Tool	<ul style="list-style-type: none"> • Follow up The position of the tool tip is maintained during swiveling. • No follow up The position of the tool tip is not maintained during swiveling. 	

8.6.2.2 Aligning milling tools - only for G code program (CYCLE800)

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 milling tool" softkeys. The "Align milling tool" input window opens.



Parameter	Description	Unit
TC	Name of the swivel data record	
Retraction	<ul style="list-style-type: none"> No: No retraction before swiveling Z: Retraction in the direction of machine axis Z Tool direction, max.: Maximum retraction in tool direction Tool direction, inc.: Incremental retraction in tool direction 	
ZR	Retraction path - (only for incremental retraction in the tool direction)	
β	Rotation around the 3rd geometry axis (for G18 Y)	Degrees
Tool	Tool tip when swiveling <ul style="list-style-type: none"> Correct The position of the tool tip is maintained during swiveling. No correction The position of the tool tip is not maintained during swiveling. 	

8.6.2.3 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.

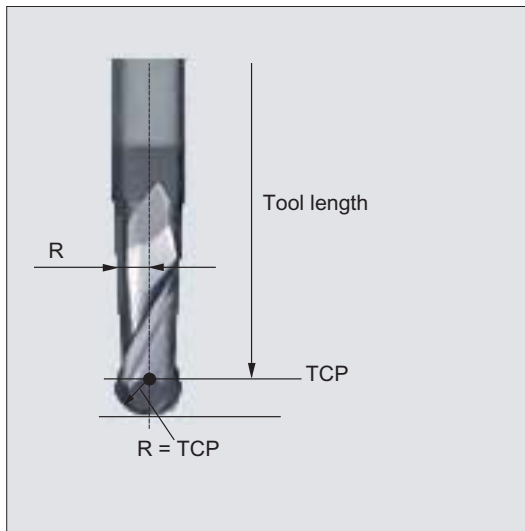






Figure 8-1 The length up to the TCP (Tool Center Point) must be entered as tool length of the radial cutter.

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
TC 	Name of the swivel data set	
Retraction 	<ul style="list-style-type: none"> • No: No retraction before swiveling • Z: Retraction in the direction of machine axis Z • Z, X, Y: Move machining axis to retraction position before swiveling • Tool direction, max.: Maximum retraction in tool direction • Tool direction, inc.: Incremental retraction in tool direction 	
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
Tool 	Tool tip when swiveling <ul style="list-style-type: none"> • Correct The position of the tool tip is maintained during swiveling. • No correction The position of the tool tip is not maintained during swiveling. 	

8.6.3 High-speed settings (CYCLE832)

Function

With the "High Speed Settings" function (CYCLE832), data for the machining of free-form surfaces is pre-assigned values so that optimum machining is possible.

The call of CYCLE832 contains three parameters:

- Tolerance
- Machining type (technology)
- Input of the orientation tolerance (for 5-axis machines)

Machining of free-form surfaces involves high requirements regarding velocity, precision and surface quality.

You can achieve optimum velocity control depending on the type of machining (roughing, semi-finishing, finishing) very simply with the "High Speed Settings" cycle.

Program the cycle in the technology program before the geometry program is called.

The "High Speed Settings" cycle is also in conjunction with the "Advanced Surface" function.



Software option

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

Machining methods

With the "High Speed Settings" function, you can select between four technological machining types:

- "Finishing"
- "Semi-finishing"
- "Roughing"
- "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.

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
Deselection	DYNNORM
Finishing	DYNFINISH
Semi-finishing	DYNSEMIFIN
Roughing	DYNROUGH

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 in conjunction with 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.

References

For additional information, please refer to the following documentation:

Commissioning Manual SINUMERIK Operate / SINUMERIK 840D sl





Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure



1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Select the "Miscellaneous" softkey.
3. Press the ">>" softkey.
4. Press the "High Speed Settings" softkey.
The "High Speed Settings" input window is opened.

Parameter	Description	Unit
Machining 	<ul style="list-style-type: none"> • ∇ (roughing) Plain text: _ROUGH • ∇∇ (semi-finishing) Plain text: _SEMIFIN • ∇∇∇ (finishing) Plain text entry: _FINISH • Deselection Plain text entry: _OFF 	
	For "Multi-axis programming yes", the following plain texts are generated in accordance with the machining type: <ul style="list-style-type: none"> • ∇ (roughing) with input of the orientation tolerance Plain text: _ORI_ROUGH • ∇∇ (semi-finishing) with input of the orientation tolerance Plain text entry: _ORI_SEMIFIN • ∇∇∇ (finishing) with input of the orientation tolerance Plain text entry: _ORI_FINISH 	
Tolerance	Tolerance of the machining axis	
Multi-axis program 	Multi-axis program for 5-axis machines <ul style="list-style-type: none"> • Yes The orientation tolerance >0 degrees can be entered here • No The value 1 is entered automatically <p>Note The field can be hidden. Please observe the machine manufacturer's instructions.</p>	

8.6.4 Subroutines

If you require the same machining steps in the programming of 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.

ShopTurn does not differentiate between main program and subprograms. This means that you can call a "standard" machining step or G code program in another machining step program as subprogram. In the subprogram, you can also call another subprogram. The maximum nesting depth is 8 subprograms.

Note

You cannot insert subprograms in linked blocks.

If you want to call a machining step program as 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.

The subprogram must always be stored in the NCK main memory (in a separate directory "XYZ" or in the "ShopTurn", "Part programs", and "Subprograms" directories).

If you want to call a subprogram located on another drive, you can use G code command "EXTCALL".

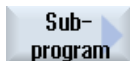
Program header

Please note that ShopTurn evaluates the settings from the program header of the subprogram, excluding the blank settings, when the subprogram is called. These settings also remain active even after the subprogram has been exited.

If you wish to activate the settings from the program header for the main program again, you can make the settings again in the main program after calling the subprogram.

Procedure

1. Create a ShopTurn or G code program that you would like 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 "Subprogram" softkeys.



4. Enter the path of the subprogram if the desired subprogram is not stored in the same directory as the main program.
5. Enter the name of the subprogram that you want to insert.
You only need to enter the file extension (*.mpf or *.spf) if the subprogram does not have the file extension specified for the directory in which the subprogram is stored.



6. 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.

8.7 Additional cycles and functions in ShopTurn

8.7.1 Drilling centric

Function

Using the "Drill centric" cycle, you can perform drilling operations at the center of a face surface.

You can choose between chip breaking during drilling or retraction from the workpiece for stock removal. During machining, either the main spindle or counterspindle rotates. You can use a drill, rotary drill or milling cutter as the tool.

The tool is moved with rapid traverse to the programmed position, allowing for the return plane and safety clearance.

Approach/retraction during chipbreaking

1. The tool drills at the programmed feedrate F as far as the first infeed depth.
2. For chipbreaking, the tool retracts by the retraction value $V2$ and drills as far as the next infeed depth that can be reduced by the factor DF .
3. Step 2 is repeated until final drilling depth $Z1$ has been reached and dwell time DT has expired.
4. The tool retracts to the safety clearance with rapid traverse.

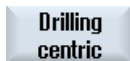
Approach/retraction during stock removal

1. The tool drills at the programmed feedrate F as far as the first infeed depth.
2. The tool is retracted from the workpiece with rapid traverse to the safety clearance for swarth removal and is then re-inserted at the 1st infeed depth in the automatic mode reduced by an anticipation distance calculated by the control system.
3. The tool then drills down to the next infeed depth that can be reduced by the factor DF and the tool retracts again to $Z0 +$ safety clearance for stock removal.
4. Step 3 is repeated until final drilling depth $Z1$ has been reached and dwell time DT has expired.
5. The tool retracts to the safety clearance with rapid traverse.

For example, if you want to drill very deep holes, you can also employ a rotating tool spindle. First specify the required tool and tool spindle speed under "Straight/Circle" → "Tool". Then program the "Drill centered" function.

Procedure

1. The ShopTurn program to be edited has been created and you are in the editor.
2. Press the "Drilling" and "Drill centric" softkeys.
The "Drilling centered" input window opens.



Parameters	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
Machining	<ul style="list-style-type: none"> • Chipbreaking • Stock removal 	
Z0	Reference point Z	
Drilling depth	Referred to <ul style="list-style-type: none"> • Shank The drill is inserted until the drill shank reaches the value programmed for Z1. The angle entered in the tool list is taken into account. • Tip The drill is inserted until the drill tip reaches the value programmed for Z1. 	
Z1	Final drilling depth X (abs) or final drilling depth in relation to Z0 (inc)	
D	Maximum depth infeed	
FD1	Percentage for the feedrate for the first infeed	%
DF	<ul style="list-style-type: none"> • Percentage for each additional infeed or • Amount for each additional infeed DF = 100: Infeed increment remains constant DF < 100: Infeed increment is reduced in direction of final drilling depth. Example: DF = 80 Last infeed was 4 mm; 4 x 80% = 3.2; next infeed increment is 3.2 mm 3.2 x 80% = 2.56; next infeed increment is 2.56 mm, etc.	% mm
V1	Minimum depth infeed Parameter V1 is available only if DF<100% has been programmed. A minimum infeed is programmed using parameter V1.	
V2	Retraction distance after each machining step – (only for "chipbreaking" operation)	

Parameters	Description	Unit
Clearance distance U	- (only for "stock removal" operation) <ul style="list-style-type: none"> • Manual • Automatic 	
V3	Clearance distance – (for "manual" clearance distance only)	
DT U	<ul style="list-style-type: none"> • Dwell time in seconds • Dwell time in revolutions 	s rev

8.7.2 Thread centered

Function

Using the "Centered tapping" cycle, tap a righthand or lefthand thread at the center of the face surface.

During machining, either the main spindle or counterspindle rotates. You can alter the spindle speed via the spindle override; feedrate override is not operative.

You can select drilling in one cut, chipbreaking or retraction from the workpiece for stock removal.

The tool is moved with rapid traverse to the programmed position, allowing for the retraction plane and safety clearance.

Approach/retraction in one cut

1. The tool drills in the direction of the longitudinal axis at the programmed spindle speed S or cutting rate V as far as the final drilling depth Z1.
2. The direction of rotation of the spindle reverses and the tool retracts to the safety clearance at the programmed spindle speed SR or cutting rate VR.

Approach/retraction for stock removal

1. The tool drills in the direction of the longitudinal axis at the programmed spindle speed S or feedrate V as far as the first infeed depth (maximum infeed depth D).
2. The tool retracts from the workpiece to the safety clearance at spindle speed SR or cutting rate VR for stock removal.
3. Then the tool is inserted again at spindle speed S or feedrate V and drills to the next infeed depth.
4. Steps 2 and 3 are repeated until the programmed final drilling depth Z1 is reached.
5. The direction of rotation of the spindle reverses and the tool retracts to the safety clearance at spindle speed SR or cutting rate VR.

Approach/retraction for chipbreaking

1. The tool drills in the direction of the longitudinal axis at the programmed spindle speed S or feedrate V as far as the first infeed depth (maximum infeed depth D).
2. The tool retracts by the retraction clearance V2 for chipbreaking.
3. The tool then drills to the next infeed depth at spindle speed S or feedrate V.
4. Steps 2 and 3 are repeated until the programmed final drilling depth Z1 is reached.
5. The direction of rotation of the spindle reverses and the tool retracts to the safety clearance at spindle speed SR or cutting rate VR.

The machine manufacturer may have made specific settings for centered tapping in a machine data element.





Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure

1. The ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Drilling" and "Drill centric" and "Thread centric" softkeys. The "Centric tapping" input window opens.



Parameters	Description	Unit
T	Tool name	
D	Cutting edge number	
F 	Feedrate	mm/min mm/rev
Table 	Thread table selection: <ul style="list-style-type: none"> • without • ISO metric • Whitworth BSW • Whitworth BSP • UNC 	

Parameters	Description	Unit
Selection U	Selection, table value: <ul style="list-style-type: none"> • M1 - M68 (ISO metric) • W3/4"; etc. (Whitworth BSW) • G3/4"; etc. (Whitworth BSP) • 1" - 8 UNC; etc. (UNC) 	
P U - (selection only possible for table selection "without")	Pitch ... <ul style="list-style-type: none"> • in MODULUS: MODULUS = Pitch/π • in mm/rev • in inch/rev • in turns per inch: Used with pipe threads, for example. <p>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.</p> <p>The pitch is determined by the tool used.</p>	MODULUS mm/rev in/rev turns/"
S / V U	Spindle speed or constant cutting rate	rpm m/min
SR	Spindle speed for retraction	rev/min
VR	Constant cutting rate for retraction	m/min
Machining U	<ul style="list-style-type: none"> • 1. Cut The thread is drilled in one cut without interruption. • Chipbreaking The drill is retracted by the retraction distance V2 for chipbreaking. • Stock removal The drill is retracted from the workpiece for stock removal. 	
Z0	Reference point Z)	mm
Z1 U	End point of the thread (abs) or thread length (inc)	mm
D	Maximum depth infeed - (for stock removal or chipbreaking only)	mm
Retraction U	- (only for "chipbreaking" operation) Retraction distance <ul style="list-style-type: none"> • Manual • Automatic 	
V2	Retraction distance (only for "manual" retraction) Distance through which the tap is retracted for chipbreaking. V2 = automatic: The tool is retracted by one revolution.	mm

8.7.3 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 the following transformations:

- Offset
- Rotation
- Scaling
- Mirroring
- Rotation C axis

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 TRAF00F.

Procedure for work offset, offset, rotation, scaling, mirroring or rotation C axis.

1. The ShopTurn 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 -






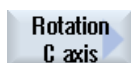
Press the "Scaling" softkey.
The "Scaling" input window opens.

- OR -



Press the "Mirroring" softkey.
The "Mirroring" input window opens.

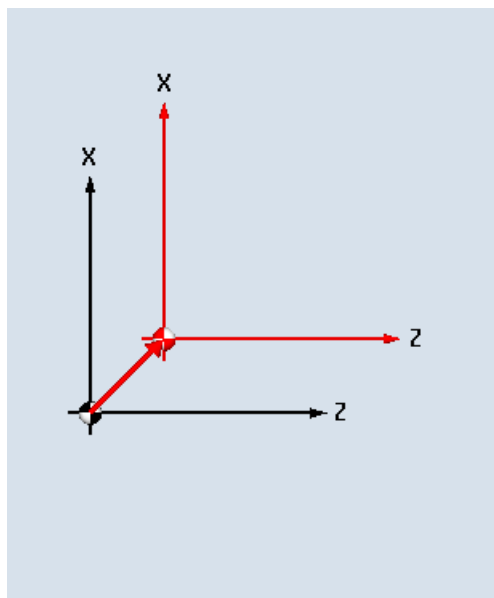
- OR -



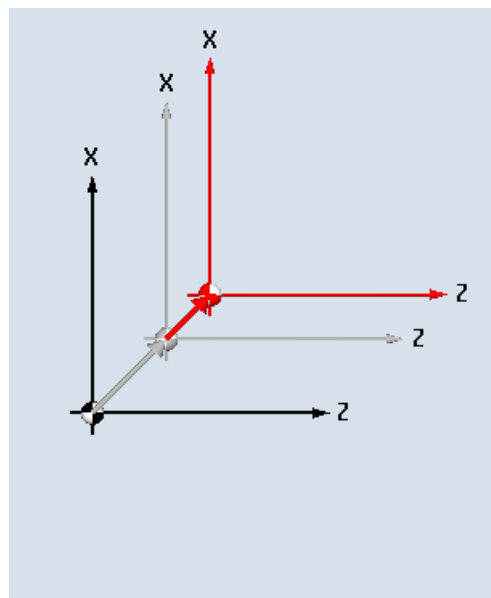
Press the "Rotation C axis" softkey.
The "Rotation C axis" input window opens.

8.7.4 Translation


For each axis, you can program an offset of the zero point.



New offset

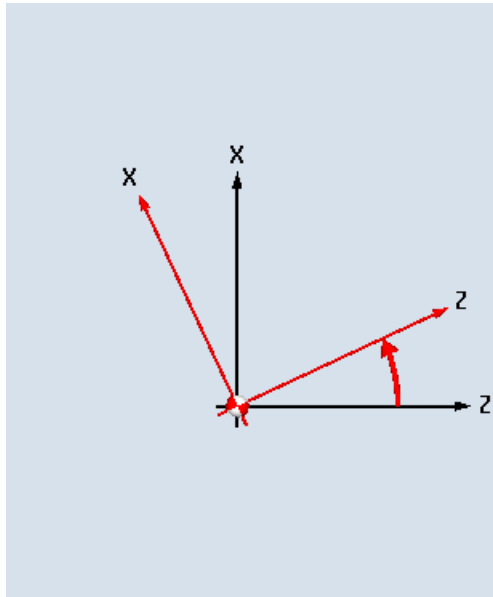


Additive offset

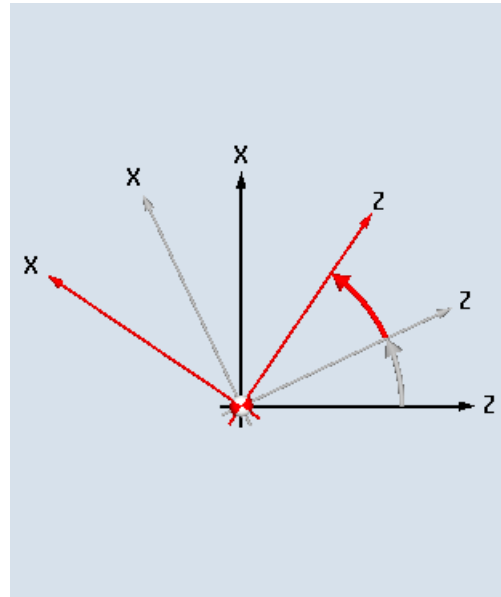
Parameters	Description	Unit
Offset 	<ul style="list-style-type: none"> New New offset Additive Additive offset 	
Z	Offset Z	mm
X	Offset X	mm
Y	Offset Y	mm

8.7.5 Rotation


You can rotate every axis through a specific angle. A positive angle corresponds to counterclockwise rotation.



New rotation

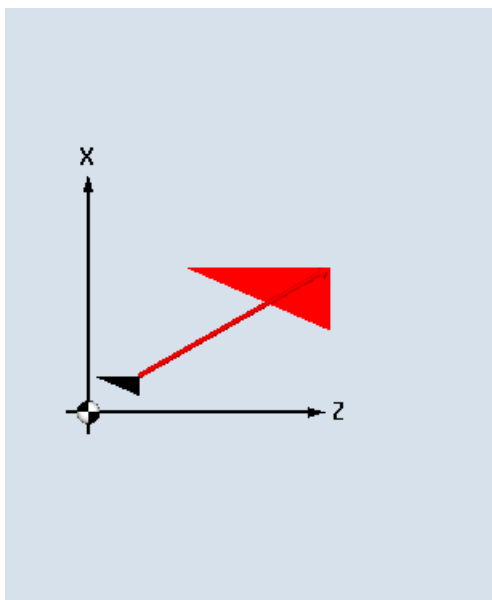


Additive rotation

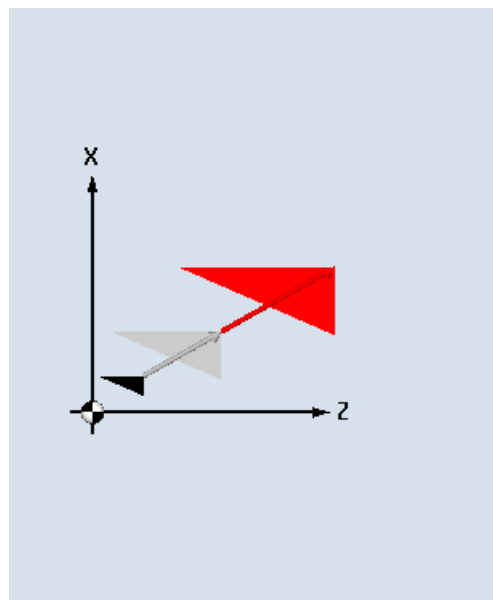
Parameters	Description	Unit
Rotation 	<ul style="list-style-type: none"> • New • New rotation 	
Z	Rotation around Z	Degrees
X	Rotation around X	Degrees
Y	Rotation around Y	Degrees

8.7.6 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.



New scaling



Additive scaling

Parameters	Description	Unit
Scaling 	<ul style="list-style-type: none"> New New scaling Additive Additive scaling 	
ZX	Scale factor ZX	
Y	Scale factor Y	

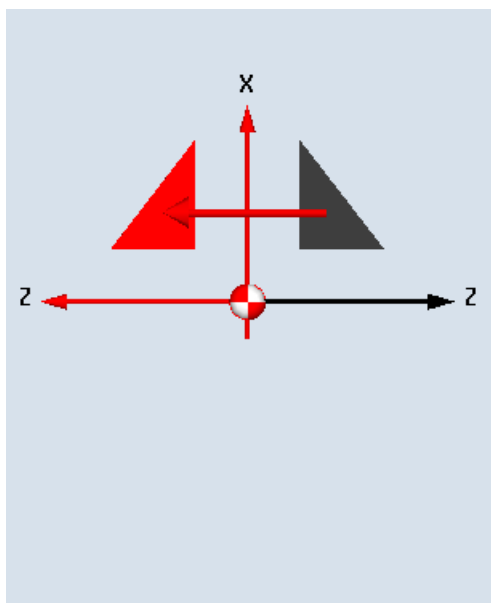
8.7.7 Mirroring

Furthermore, you can mirror all axes. Enter the axis to be mirrored in each case.

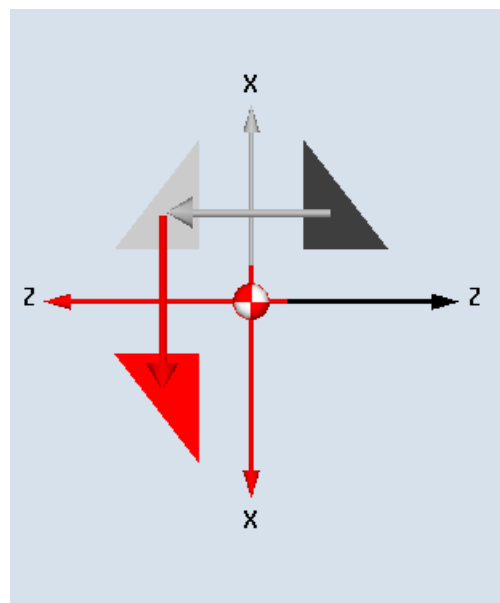
Note

Travel direction of the milling cutter

Note that with mirroring, the travel direction of the cutting tool (conventional/climb) is also mirrored.



New mirroring



Additive mirroring

Table 8- 2

Parameters	Description	Unit
Mirroring <input type="checkbox"/>	<ul style="list-style-type: none"> New New mirroring Additive Additive mirroring 	
Z <input type="checkbox"/>	Mirroring of the Z axis, on/off	
X <input type="checkbox"/>	Mirroring of the X axis, on/off	
Y <input type="checkbox"/>	Mirroring of the Y axis, on/off	

8.7.8 Rotation C

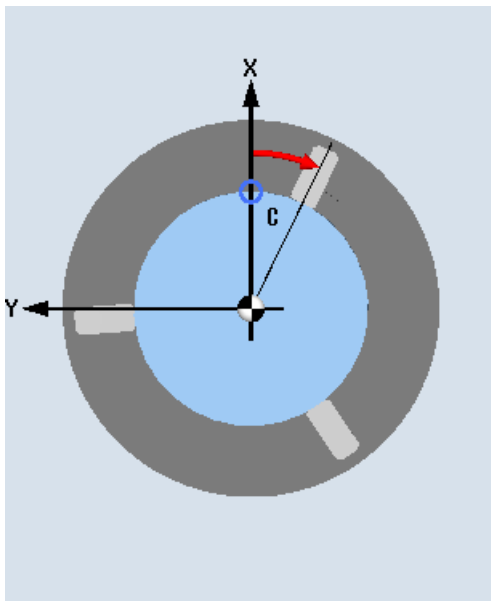
You can rotate the C axis through a specific angle to enable subsequent machining operations to be performed at a particular position on the face or peripheral surface.

The direction of rotation is set in a machine data element.

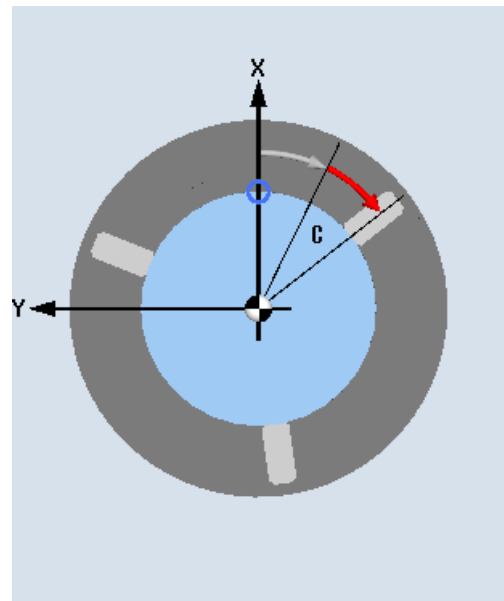


Machine manufacturer


Please refer to the machine manufacturer's specifications.



New C axis rotation



Additive C axis rotation

Parameters	Description	Unit
Rotation 	<ul style="list-style-type: none"> • New New rotation • Additive Additive rotation 	
C	Rotation C	Degrees

8.7.9 Straight and circular machining

If you want to perform simple, i.e. straight or circular path movements or machining without defining a complete contour, you can use the functions "Straight" or "Circle" respectively.

General sequence

To program simple machining operations, proceed as follows:

- Specify the tool and the spindle speed
- Program the machining operations

Machining options

The following machining options are available:

- Straight line
- Circle with known center point
- Circle with known radius
- 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 retract the tool to the retraction area defined in the program header using either straight or circular path motion, then you must carefully ensure that a collision cannot occur as a result of the normal retraction logic.
--

To be on the safe side, you should also move the tool back out of the retraction area again.
--

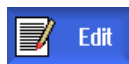
8.7.10 Selecting a tool and machining plane

Before you can program a line or circle, you have to select the tool, spindle, spindle speed and machining plane.

If you program a sequence of different straight or circular path motions, the settings for the tool, spindle, spindle speed and machining plane remain active until you change them again.

If you change the selected machining plane subsequently, the coordinates of the programmed path motion are automatically adjusted to the new machining plane. The originally programmed coordinates remain unchanged only for a straight motion (right-angled, not polar).

Procedure



1. The ShopTurn program to be processed 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 "Tool" window is opened.

4. Enter a tool into parameter field "T".

- OR -



Press the "Select tool" softkey if you want to select a tool from the tool list, position the cursor on the tool that you wish to use for the machining operation 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. In the lefthand input field of the Spindle parameter, select main spindle, tool spindle or counterspindle.
7. Enter the spindle speed or cutting rate.
8. In the selection box "Plane selection", select between the machining planes.
9. Enter the cylinder diameter if you selected the machining plane peripheral surface C.

- OR -

Enter the positioning angle for the CP machining area if you selected machining plane face Y.

- OR -

Enter reference point C0 if you selected the machining plane peripheral surface Y.

- OR -



Choose whether the spindle should be clamped or released or whether there should be no change (input field left blank).

Press the "Accept" softkey.

The values are saved and the window is closed. The process plan is displayed and the newly generated program block is marked.

Parameters	Description	Unit
T	Tool name	
D	Cutting edge number	
S1 / V1	Spindle speed or constant cutting rate	rpm m/min
Plane selection	Select between the following machining surfaces: <ul style="list-style-type: none"> • Peripheral surface/Peripheral C • Peripheral surface Y - only if there is a Y axis • Face/Face C • Face Y - only if there is a Y axis • Turning 	
∅	Diameter of the cylinder (for peripheral surface/peripheral surface C)	mm
C0	Positioning angle for machining area (for peripheral surface Y)	Degrees
CP	Positioning angle for machining area (for face Y)	Degrees

8.7.11 Programming a straight line

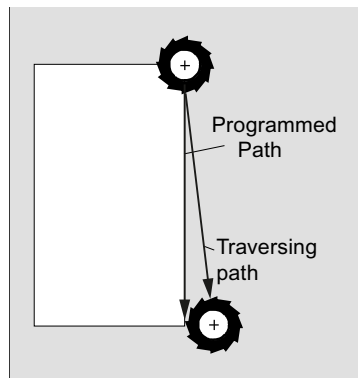
When you want to program a straight line in right-angled coordinates, you can use the "Straight" function.

The tool moves along a straight line at the programmed feedrate or at rapid traverse from its actual position to the programmed end position.

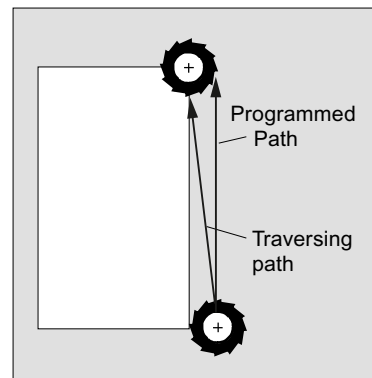
Radius compensation

Alternately, 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 select radius compensation only 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.



Straight line when selecting radius compensation



Straight line when deselecting radius compensation

















If you want to prevent deviation from the programmed path, you can program the first straight line with radius compensation or with deactivated radius compensation outside the workpiece. Programming without coordinate data is not possible.

Procedure

1. The ShopTurn program to be processed has been created and you are in the editor.
2. Press the menu forward key and the "Straight Circle" softkey.



3. Press the "Straight" softkey.
4. Press the "Rapid traverse" softkey if you want to use rapid traverse instead of a programmed machining feedrate.

Parameters	Description	Unit
X 	Target position X Ø (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
U 	Target position (abs) or target position referred to the actual position (inc)	mm
C 	Target angle (abs) or target angle referred to the actual position (inc)	Degrees
C1 	Target position of C axis of main spindle (abs or inc)	mm
C3 	Target position of C axis of counterspindle (abs or inc)	mm
Z3 	Target position of special axis (abs or inc) Note: Incremental dimension: The sign is also evaluated.	mm
AWZ 	Target angle (abs) or target angle referred to the actual position (inc)	Degrees
GS 	Target angle (abs) or target angle referred to the actual position (inc)	Degrees
F 	Machining feedrate Alternatively, rapid traverse	mm/rev mm/min mm/tooth
Radius compensation 	Input defining which side of the contour the cutter travels in the traversing 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.

8.7.12 Programming a circle with known center point

To program a circle or arc with a known center point, use the "Circle center point" function.








The tool traverses a circular path from its actual position to the programmed target position at the machining feedrate. The system calculates the radius of the circle/arc on the basis of the entered interpolation parameter settings I and K.








Procedure

1. The ShopTurn program to be processed 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.

Parameters	Description	Unit
Direction of rotation	Direction of rotation in which the tool travels from the circle starting point to the circle end point:	
	 Direction of rotation clockwise (right)	
	 Direction of rotation counterclockwise (left)	
Y  Z  J K	Machining plane, peripheral surface C Target position Y (abs) or target position X referred to the last programmed position (inc) Target position Z (abs) or target position Y referred to the last programmed position (inc) Circle center point J (ink). Circle center point K (inc). Note: Incremental dimension: The sign is also evaluated.	mm mm mm mm
Y  Z  J K	Machining plane, peripheral surface Y Target position Y (abs) or target position X referred to the last programmed position (inc) Target position Z (abs) or target position Y referred to the last programmed position (inc) Circle center point J (ink). Circle center point K (inc) Note: Incremental dimension: The sign is also evaluated.	mm mm mm mm

Parameters	Description	Unit
X  Y  I J	<p>Machining plane face C</p> <p>Target position X Ø (abs) or target position X referred to the last programmed position (inc)</p> <p>Target position Y (abs) or target position Y referred to the last programmed position (inc)</p> <p>Circle center point I (inc)</p> <p>Circle center point J (inc)</p> <p>Note: Incremental dimension: The sign is also evaluated.</p>	mm mm mm mm
X  Y  I J	<p>Machining plane face Y</p> <p>Target position X (abs) or target position X referred to the last programmed position (inc)</p> <p>Target position Y (abs) or target position Y referred to the last programmed position (inc)</p> <p>Circle center point I (inc).</p> <p>Circle center point J (inc).</p> <p>Note: Incremental dimension: The sign is also evaluated.</p>	mm mm mm mm
X  Z  I K	<p>Machining plane rotation</p> <p>Target position X Ø (abs) or target position Y referred to the last programmed position (inc)</p> <p>Target position Z (abs) or target position X referred to the last programmed position (inc)</p> <p>Circle center point I (inc).</p> <p>Circle center point K (inc)</p> <p>Note: Incremental dimension: The sign is also evaluated.</p>	mm mm mm mm
F 	Machining feedrate	mm/rev mm/min mm/tooth

8.7.13 Programming a circle with known radius

To program a circle or arc with a known radius, use the "Circle radius" function.

The tool traverses a circular arc with the programmed radius from its actual position to the programmed target position at the machining feedrate. To do this, the system calculates the position of the circle center point.

You can choose to traverse the arc in the clockwise or anticlockwise direction. Depending on the direction of rotation, there are two options for approaching the target position from the current position via an arc of the specified radius.

You can select the arc of your choice by entering a positive or a negative sign for the radius.

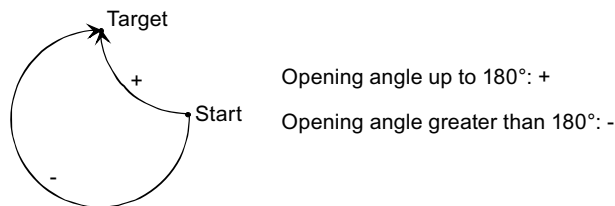


Figure 8-2 Opening angle















Procedure

1. The ShopTurn program to be processed 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.



Parameters	Description	Unit
Direction of rotation	Direction of rotation in which the tool travels from the circle starting point to the circle end point	
	 Direction of rotation clockwise (right)	
	 Direction of rotation counterclockwise (left)	
Y  Z 	Machining plane peripheral surface/peripheral surface C Target position Y (abs) or target position X referred to the last programmed position (inc) Target position Z (abs) or target position Y referred to the last programmed position (inc) Note: Incremental dimension: The sign is also evaluated	mm mm
Y  Z 	Machining plane, peripheral surface Y Target position Y (abs) or target position X referred to the last programmed position (inc) Target position Z (abs) or target position Y referred to the last programmed position (inc) Note: Incremental dimension: The sign is also evaluated.	mm mm
X  Y 	Machining plane face/face C Target position X (abs) or target position X referred to the last programmed position (inc) Target position Y (abs) or target position Y referred to the last programmed position (inc) Note: Incremental dimension: The sign is also evaluated	mm mm
X  Y 	Machining plane face Y Target position X (abs) or target position X referred to the last programmed position (inc) Target position Y (abs) or target position Y referred to the last programmed position (inc) Note: Incremental dimension: The sign is also evaluated.	mm mm
X  Z 	Machining plane rotation Target position X \varnothing (abs) or target position Y referred to the last programmed position (inc) Target position Z (abs) or target position X referred to the last programmed position (inc) Note: Incremental dimension: The sign is also evaluated.	mm mm
R	Radius of circular arc The sign determines the type of arc traversed.	mm mm
F 	Machining feedrate.	mm/rev mm/min mm/tooth

8.7.14 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 dimensions as polar coordinates.

Before you program a straight line or circle in polar coordinates, you must define the pole, i.e. the reference point, of the polar coordinate system.











Procedure

1. The ShopTurn program to be processed 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 "Pole" softkeys.



Parameters	Description	Unit
Y  Z 	Machining plane peripheral surface/peripheral surface C Pole Y (abs) Pole Z (abs) or pole Z referred to the last programmed position (inc) Note: Incremental dimension: The sign is also evaluated.	mm mm
Y  Z 	Machining plane, peripheral surface Y Pole Y (abs) Pole Z (abs) or pole Z referred to the last programmed position (inc) Note: Incremental dimension: The sign is also evaluated.	mm mm
X  Y 	Machining plane face/face C Pole X \varnothing (abs) Pole Y (abs) or pole Y referred to the last programmed position (inc) Note: Incremental dimension: The sign is also evaluated.	mm mm
X  Y 	Machining plane face Y Pole X (abs) Pole Y (abs) or pole Y referred to the last programmed position (inc) Note: Incremental dimension: The sign is also evaluated.	mm mm
X  Z 	Machining plane rotation Pole X (abs) or pole X referred to the last programmed position (inc) Z position pole (abs) Note: Incremental dimension: The sign is also evaluated.	mm mm

8.7.15 Straight line polar

When you want to program a straight line in polar coordinates, you can use the "Straight Polar" function.

A straight line in the polar coordinate system is defined by the length L and the angle α . Depending on the selected machining plane, the angle refers to another axis. The direction in which a positive angle points also depends on the machining plane.

Machining plane	Turning	Face	Peripheral
Reference axis for angle	Z	X	Y
Positive angle in direction of the axis	X	Y	Z

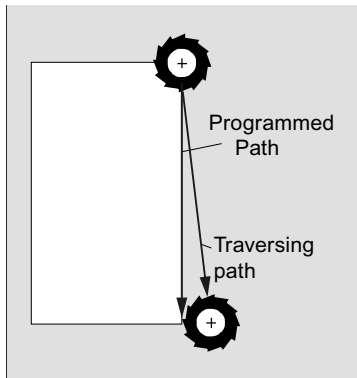
The tool traverses a straight line from its current position to the programmed end point at the machining feedrate or at rapid traverse.

The 1st line in polar coordinates entered after the pole must be programmed in absolute dimensions. You can program any additional lines or arcs also in incremental dimensions.

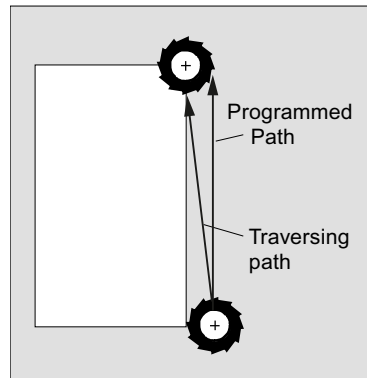
Radius compensation

Alternately, 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 select radius compensation only in the first program block.

For the first straight line with radius compensation, the tool approaches the starting point without radius compensation and the end point with radius compensation, i.e. if a vertical path is programmed, a slope will be traversed. The compensation does not act over the entire traverse path until the second programmed straight line with radius compensation. The reverse effect occurs when radius compensation is deactivated.



Straight line with selected radius compensation










Straight line with deselected radius compensation

If you want to prevent deviation from the programmed path, you can program the first straight line with radius compensation or with deactivated radius compensation outside the workpiece. Programming without coordinate data is not possible.

Procedure



1. The ShopTurn program to be processed 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 if you want to use rapid traverse instead of a programmed machining feedrate.

Parameters	Description	Unit
L	Distance to the pole, end point	mm
α 	Polar angle to the pole, end point (abs) or Polar angle change to the pole, end point (inc) The sign specifies the direction.	Degrees
F 	Machining feedrate	mm/rev mm/min mm/tooth
Radius compensation 	Input defining which side of the contour the cutter travels in the traversing direction:	
	 Radius compensation to left of contour	
	 Radius compensation to right of contour	
	 Radius compensation off	
	 The set radius compensation remains as previously set	

8.7.16 Circle polar

If you want to program a circle or arc using polar coordinates, you can use the "Circle Polar" function.

A circle in the polar coordinate system is defined by the angle α . Depending on the selected machining plane, the angle refers to another axis. The direction in which a positive angle points also depends on the machining plane.

Machining plane	Rotate	Face	Peripheral
Reference axis for angle	Z	X	Y
Positive angle in direction of the axis	X	Y	Z

The tool traverses a circular path from its actual position to the programmed end point (angle) at the machining feedrate. The radius is obtained from the distance between the actual tool position and the defined pole, i.e. the circle start and end point positions are at the same distance from the pole.

The 1st arc in polar coordinates entered after the pole must be programmed in absolute dimensions. You can program any additional lines or arcs also in incremental dimensions.






Procedure

1. The ShopTurn program to be processed 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.



Parameters	Description	Unit
Direction of rotation 	Direction of rotation in which the tool travels from the circle starting point to the circle end point	
	 Direction of rotation clockwise (right)	
	 Direction of rotation counterclockwise (left)	
α 	Polar angle to the pole, end point (abs) or Polar angle change to the pole, end point (inc) The sign specifies the direction.	Degrees
F 	Machining feedrate	mm/rev mm/min mm/tooth

8.7.17 Machining with the counterspindle

If your lathe has a counterspindle, you can machine workpieces using turning, drilling and milling functions on the front and rear faces without reclamping the workpiece manually.

Before machining commences on the rear face, the counterspindle must grip the workpiece, remove it from the main spindle and move it to the new machining position. You can program these operations with the "Counterspindle" function.

Operations

The following steps are available to program the operations:

- Gripping: Gripping the workpiece with the counterspindle (with limit stop if necessary)
- Withdrawing: Withdrawing the workpiece out of the main spindle with the counterspindle
- Rear side: Moving the workpiece with the counterspindle to the new machining position
- Complete: Gripping, withdrawing (optionally with cutting-off) and rear face
- Front side: Work offset for machining the next front face (for bars)

If you start to execute a program containing a counterspindle machining operation, the counterspindle is first retracted to the return position defined in a machine data element.









Machine manufacturer

Please refer to the machine manufacturer's specifications.

Teaching in the parking position and angle offset

Teaching the park position is possible only if you have selected the machine coordinate system (MCS).

1. Manually rotate the counterspindle chuck to the desired position, and move the tool to the desired position.
2. Press the "Various" and "Counterspindle" softkeys.


3. Select the "Gripping" or "Complete" programming step.

4. Select "MCS" tool under the park position.

5. Press the "Teach park pos." softkey.

The actual tool park position is saved.
6. Press the "Teach angl. offset" softkey.

The actual angular difference between the main and counter spindles will be saved.

8.7.17.1 Programming example: Machining main spindle – Transfer workpiece – Machining counterspindle

The programming for this operation might look like this:

Programming steps - alternative 1:

- Machining, main spindle
- Gripping
- Withdrawing
- Rear face
- Machining, counterspindle

Programming steps - alternative 2:

- Machining, main spindle
- Complete (gripping, withdrawing and rear face)
- Machining, counterspindle

8.7.17.2 Programming example: Machining, counterspindle - without previous transfer

Programming steps

- Rear face
 - Work offset
Work offset is only activated
 - ZV:
Parameter is not evaluated.
- Machining, counterspindle

Note

Special feature regarding "rear face":

The work offset that you choose in the parameter screen is only activated and not calculated. This means that the workpiece zero for counterspindle machining should be stored in the work offset. In addition, parameter ZV is not evaluated.

8.7.17.3 Programming example: Machining bar material

If you use bars to produce your workpieces, you can machine several workpieces on the front and rear face by starting the program just once.








Programming steps - alternative 1:



- Program header specifying the work offset in which the workpiece zero is stored
- Machining, main spindle
- Complete (withdraw blank: yes; cut-off cycle: yes)
- Cutting-off
- Machining, counterspindle
- End of program with number of workpieces to be machined




Programming steps - alternative 2:



- Start marker
- Machining, main spindle
- Complete (withdraw blank: yes; cutting-off cycle: yes)
- Cutting-off
- Machining, counterspindle
- Front face
- End marker
- Repeat from start to end marker


8.7.17.4 Parameters, counterspindle


Parameters	Description	Unit
Function 	You can select between 5 different functions: <ul style="list-style-type: none"> • Complete • Gripping • Withdrawing • Rear face • Front face 	
Function, complete	Gripping	
Coordinate system 	<ul style="list-style-type: none"> • Machine coordinate system The park position is specified in the machine coordinate system. Teaching in the park position and angular offset is only possible in the machine coordinate system. • Workpiece coordinate system The park position is specified in the workpiece coordinate system. 	
XP	Park position of tool in X direction (abs)	mm
ZP	Park position of tool in Z direction (abs)	mm
Flush chuck 	Flush counterspindle chuck <ul style="list-style-type: none"> • Yes • No 	
DIR 	Direction of rotation <ul style="list-style-type: none"> •  Spindle rotates clockwise •  Spindle rotates counterclockwise •  Spindle does not rotate 	
S	Spindle speed – (only when the spindle rotates)	rev/min
α1	Angular offset	Degrees
Z1	Transfer position (abs.)	

Parameters	Description	Unit
ZR 	Position, feedrate reduction (abs or inc) Position from which a reduced feedrate is used.	
FR	Reduced feedrate	mm/rev
Fixed stop	Travel to fixed stop <ul style="list-style-type: none"> • Yes The counterspindle stops at a defined distance away from transfer position Z1 and then traverses with a defined feedrate up to the fixed stop. • No The counterspindle traverses to the transfer position Z1. 	
Function, complete	Withdrawing	
Withdraw blank	Withdraw complete plank: <ul style="list-style-type: none"> • Yes • No 	
F	Feedrate	mm/min
Cutting-off cycle	Cutting-off cycle in the following block <ul style="list-style-type: none"> • Yes • No 	
Function, complete	Rear face	
Work offset 	Work offset in which the coordinate system, which was shifted according to ZW and by ZV as well as mirrored in Z, must be saved: <ul style="list-style-type: none"> • Basic reference • G54 • G55 • G56 • G57 • ... 	
ZW	Machining position for special axis (abs.); machine coordinate system	mm
ZV	Offset Z = 0 Workpiece zero is offset in Z direction (inc, the sign is also evaluated)	mm

Parameters	Description	Unit
Function, gripping	Teaching in the park position and angular offset is possible	
Coordinate system U	<ul style="list-style-type: none"> Machine coordinate system The park position is specified in the machine coordinate system. Teaching in the park position and angular offset is only possible in the machine coordinate system. Workpiece coordinate system The park position is specified in the workpiece coordinate system. 	
XP	Park position of tool in X direction (abs)	mm
ZP	Park position of tool in Z direction (abs)	mm
Flush chuck U	Flush counterspindle chuck <ul style="list-style-type: none"> Yes No 	
DIR U	Direction of rotation <ul style="list-style-type: none">  Spindle rotates clockwise  Spindle rotates counterclockwise  Spindle does not rotate 	
S	Spindle speed – (only when the spindle rotates)	rev/min
α1	Angular offset	Degrees
Z1	Transfer position (abs.)	
ZR U	Position, feedrate reduction (abs or inc) Position from which a reduced feedrate is used.	
FR	Reduced feedrate	mm/rev
Fixed stop	Travel to fixed stop <ul style="list-style-type: none"> Yes The counterspindle stops at a defined distance away from transfer position Z1 and then traverses with a defined feedrate up to the fixed stop. No The counterspindle traverses to the transfer position Z1. 	

Parameters	Description	Unit
Function, withdrawing		
Also take zero point 	Also take zero point <ul style="list-style-type: none"> • Yes • No 	
Work offset 	Work offset in which the coordinate system offset by Z1 must be saved. <ul style="list-style-type: none"> • Basic reference • G54 • G55 • G56 • G57 • ... 	
Z1	Amount by which the workpiece is withdrawn from the main spindle (inc)	
F	Feedrate	mm/min

Parameters	Description	Unit
Function, rear face		
Work offset 	Work offset in which the coordinate system, which was shifted according to ZW and by ZV as well as mirrored in Z, must be saved: <ul style="list-style-type: none"> • Basic reference • G54 • G55 • G56 • G57 • ... 	
ZW	Machining position for special axis (abs.); machine coordinate system	mm
ZV	Offset Z = 0 Workpiece zero is offset in Z direction (inc, the sign is also evaluated)	mm

Parameters	Description	Unit
Function, front face		
Work offset 	Work offset for machining the next front face: <ul style="list-style-type: none"> • Basic reference • G54 • G55 • G56 • G57 • ... 	

Multi-channel view

9.1 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

9.2 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 refer to the machine manufacturer's specifications.

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" operating mode 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, for your multi-channel machine, you always only wish to monitor one channel, 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.

See also

Setting the multi-channel view (Page 535)

9.3 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	<p>The following windows are displayed one above the other for each channel:</p> <ul style="list-style-type: none"> • Actual Value window • T,F,S window • Block Display window <p>Selecting functions</p> <ul style="list-style-type: none"> • The T,F,S window is overlaid by pressing one of the vertical softkeys.
4-channel view	<p>The following windows are displayed one above the other for each channel:</p> <ul style="list-style-type: none"> • 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 <p>Selecting functions</p> <ul style="list-style-type: none"> • The window showing the G codes is overlaid if you press one of the vertical softkeys.

Toggleing 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.

9.4 Setting the multi-channel view

Setting	Meaning
View	Here, you specify how many channels are displayed. <ul style="list-style-type: none"> • 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.

Multi-channel support

10.1 Working with several channels

Multi-channel support

SINUMERIK Operate supports you when generating the program, the simulation and when running-in a program on multi-channel machines.



Software options

For the multi-channel functionality and support, i.e. for generating and editing synchronized programs in the multi-channel editor as well as the block search, you require the "programSYNC" option.



Software options

You require the "ShopMill/ShopTurn" option to generate and edit ShopTurn machining step programs.

Note

Execution and simulation

The execution and simulation for multi-channel programming does not function if the programs and the job list are on an external storage medium, e.g. on the local drive.

Multi-channel view

With the multi-channel view, you have the option of viewing several channels in parallel on the display. This means that for multi-channel machines, the execution of several programs - simultaneously started - can be monitored and controlled.

View of the channels

In the window "Settings for multi-channel view" or "Settings for multi-channel functionality", you set which channels are important for the program execution and which channels are displayed simultaneously. In so doing, you also define the channel sequence.

Note

Hidden channels

Hidden channels still belong to the group of channels that are handled together. They are only temporarily excluded from the multi-channel view.

10.2 Multi-channel editor

In the multi-channel editor, you have the option of simultaneously opening several programs and editing them. In this case, the multi-channel editor supports you regarding program synchronization from a time perspective.



Software options

You require the "programSYNC" option to generate and edit synchronized programs in the multi-channel editor.

10.2.1 Creating a multi-channel program

All of the programs involved in a multi-channel machining operation are combined in one workpiece.

In a job list, enter the program names, define the program type - G code or ShopTurn program - and assign these to a channel.



Software options

You require the "ShopMill/ShopTurn" option to generate and edit ShopTurn programs.



Machine manufacturer

If you only program G code programs, then you can switch-out the multi-channel view.

Please refer to the machine manufacturer's specifications.

Requirement

- "programSYNC" option

Procedure



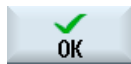
1. Select the "Program Manager" operating area.



2. Press the "NC" softkey and select the "Workpieces" folder.



3. Press the "New" and "programSYNC multi-channel" softkeys.
The "New job list" window opens.



4. Enter the required name and press the "OK" softkey.
The "Job list *.JOB" window opens.
For each channel that has been set-up, the window has one line to input or select the assigned program.



5. Position the cursor on the required channel line, enter the required program name and select the program type (G code or ShopTurn).
6. Press the "OK" softkey.
The "Multi-channel data" parameter screen opens in the editor.

10.2.2 Entering multi-channel data

In the parameter screen "multi-channel data", enter the following data, which apply for all channels for G code and ShopTurn programs:

- Measurement unit
- Zero offset (e.g. G54)
- Z value of the zero offset (optional)
- Blank
- Speed limitation
- if required, data for the counter spindle







The data defined here are saved as cycle call in the job list.




Machine manufacturer

If you are working with pure G code programming, it is possible that the parameter screen "multi-channel data" does not open.

Please refer to the machine manufacturer's instructions.

Parameters	Description	Unit
Measurement unit 	Selecting the measurement unit	mm inch
Main spindle		
Zero offset	Selecting the zero offset	
Write to the zero offset 	<ul style="list-style-type: none"> • Yes Parameter ZV is displayed • No Parameter ZV is not displayed 	
ZV	Z value of the zero offset For G54, the Z value is entered into the zero offset. Note: Please refer to the machine manufacturer's instructions.	
Blank 	<ul style="list-style-type: none"> • Tube • Cylinder • Polygon • Centered cuboid 	
XA	Outside diameter \varnothing – for tube and cylinder	mm
XI	Inside diameter (abs) or wall thickness (inc) – only for tube	mm
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)	
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
S	Speed limitation of the main spindle	rev/min
Counterspindle		
Zero offset	Selecting the zero offset	
Write to the zero offset	<ul style="list-style-type: none"> • Yes Parameter ZV is displayed • No Parameter ZV is not displayed 	
ZV	Z value of the zero offset For G54, the Z value is entered into the zero offset.	

Parameters	Description	Unit
Blank 	<ul style="list-style-type: none"> • Tube • Cylinder • Polygon • Centered 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
S	Speed limitation of the counterspindle	rev/min

Procedure

1. You have created programs for the multi-channel machining in the job list and the parameter screen "multi-channel data" is open in the editor.
2. Enter the data for the cross-channel data.
3. Press the "Accept" softkey.



The multi-channel editor is opened and displays the programs that have been created.

The cursor is positioned on an empty line before the cycle for the job list (CYCLE208). This means that you have the option to enter initializations for the G code program that are immediately required.

10.2.3 Editing the multi-channel program

10.2.3.1 Changing the job list

You now have the option to change the composition of the programs and/or the assignment of the channel and program in a job list.

Precondition

- "programSYNC" option

Procedure



1. Select the "Program Manager" operating area.



2. Select where the multi-channel program should be archived



3. Position the cursor in the "Workpieces" folder on a job list and press the "Open" softkey.

The window "Job list * JOB" is opened and the program assignment to the channels is displayed.



4. Select the channel to which you wish to assign a new program and press the softkey "Select program".

The "Program" window is opened and displays all of the programs created in the workpiece.

- OR -



Press the "Open job list" softkey.

10.2.3.2 Editing a G code multi-channel program

Editing a G code multi-channel program

Precondition

- The "programSYNC" option is set.
- In order to display the machining at the counterspindle at the correct position in the simulation, the linear axis of the counterspindle must be positioned before CYCLE208 (multi-channel data).



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure



1. Position the cursor in the "Workpieces" folder on a job list and press the "Open" softkey.

Note:

If the cursor is located on a workpiece, then a search is made for a job list with the same name.

The "Job list ..." window opens and the program assignment to the channels is displayed.



2. Press the "OK" softkey.

The programs are displayed next to one another in the editor.



3. Position the cursor on the first block of the program (multi-channel data) and press the <Cursor right> key.

The parameter screen "Multi-channel data" is opened.

4. Enter the required values if you wish to change cross-channel data.

Adding multi-channel data in a G code program





You have the possibility of adding the multi-channel cycle (CYCLE208) subsequently.

Procedure



1. The double editor is opened and the cursor is positioned in the G code program.
2. Press the "Misc." and "Multi-channel data" softkeys.
The "Call multi-channel data" input window opens.
A field for specifying the job list appears.
This field is read-only.
3. Press the "Accept job list" softkey.
The name of the job list is entered in the field.
4. Press the "Accept" softkey.
CYCLE208 is taken over into the program. The name of the job list is indicated in brackets.

Modify blank

Parameter	Description	Unit
Data for	Here, you specify the spindle selected for the blank. <ul style="list-style-type: none"> • Main spindle • Counterspindle 	
Blank 	The following blanks can be selected: <ul style="list-style-type: none"> • Tube • Cylinder • Polygon • Centered cuboid • Delete 	
W	Width of the blank - (only for centered cuboid)	mm
L	Length of the blank - (only for centered cuboid)	mm
N	Number of edges – (only for polygon)	
SW or L 	Width across flats or edge length – (only for polygon)	
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)	

Parameter	Description	Unit
XA	Outside diameter – (only for tube and cylinder)	mm
XI	Inside diameter (abs) or wall thickness (inc) – for tube only	mm

Procedure



1. The double editor is opened and the cursor is positioned in the G code program.
2. Press the "Misc." and "Blank" softkeys.
The "Blank Input" window opens.
3. Select the desired blank and enter the corresponding values.
4. Press the "Accept" softkey.

10.2.3.3 Editing a ShopTurn multi-channel program

Precondition

The "programSYNC" option is set.

Procedure










1. Position the cursor in the "Workpieces" folder on a job list and press the "Open" softkey.
Note:
If the cursor is located on a workpiece, then a search is made for a job list with the same name.
The "Job list ..." window opens and the program assignment to the channels is displayed.
2. Press the "OK" softkey.
The programs are displayed next to one another in the editor.
3. Open the program header if you wish to define cross-program or cross-channel entries.

Program header with multi-channel data


In the program header, set the following parameters, which are effective for the complete program.

In the parameter screen, you have the following options to save cross-program data:

- You can enter values in a common data set for the main and counterspindle
- You can enter values for the main and counter spindle
- You can enter values for the counterspindles






Parameter	Description	Unit
Multi-channel data	Yes Name of the job list in which the channel data are saved.	
Data for 	<ul style="list-style-type: none"> • Main+counterspindle All values for the main and counterspindle are saved in one data set • Main spindle Data set for the main spindle • Counterspindle Data set for the counterspindle <p>Note: If the machine does not have a counterspindle, then the entry field "Data for" is not applicable.</p>	
Retraction 	<p>The retraction area indicates the area outside of which collision-free traversing of the axes must be possible.</p> <ul style="list-style-type: none"> • simple • Extended • all 	
XRA 	Retraction plane X external \emptyset (abs) or retraction plane X referred to XA (inc)	
XRI 	- not for "basic" retraction retraction plane X internal \emptyset (abs) or retraction plane X referred to XI (inc) - not for "pipe" blank	
ZRA 	Retraction plane Z front (abs) or retraction plane Z referred to ZA (inc)	
ZRI 	Retraction plane Z rear – only for retraction "all"	
Tailstock 	<ul style="list-style-type: none"> • Yes • No 	
XRR	Retraction plane tailstock – only "Yes" for tailstock For "Main+counterspindle", the tailstock only refers to the main spindle (tailstock on the counterspindle side)	















Parameter	Description	Unit
Tool change point U	Tool change point, which must be approached by the revolver with its zero point. <ul style="list-style-type: none"> Work (Workpiece Coordinate System) Machine (Machine Coordinate System) Notes <ul style="list-style-type: none"> The tool change point must be far enough outside the retraction area that it is not possible for any tool to protrude into the retraction area while the revolver is moving. Ensure that the tool change point is relative to the zero point of the revolver and not the tool tip. 	
XT	Tool change point X Ø	
ZT	Tool change point Z	
Data for U	<ul style="list-style-type: none"> Main spindle Counterspindle <p>If several spindles have been set-up, the program can operate at both spindles</p> <ul style="list-style-type: none"> Empty <p>The program only operates at one spindle</p>	
Retraction U	The retraction area indicates the area outside of which collision-free traversing of the axes must be possible. <ul style="list-style-type: none"> simple extended – (not for a pipe blank) all 	
XRA U	Retraction plane X external Ø (abs) or retraction plane X referred to XA (inc)	
XRI U	- only for a pipe blank retraction plane X internal Ø (abs) or retraction plane X referred to XI (inc)	
ZRA U	Retraction plane Z front (abs) or retraction plane Z referred to ZA (inc)	rev/min
ZRI U	Retraction plane Z rear – only for retraction "all"	
Tailstock U	<ul style="list-style-type: none"> Yes No 	
XRR	Retraction plane tailstock – only "Yes" for tailstock	
Tool change point U	Tool change point, which must be approached by the revolver with its zero point. <ul style="list-style-type: none"> Work (Workpiece Coordinate System) Machine (Machine Coordinate System) Notes <ul style="list-style-type: none"> The tool change point must be far enough outside the retraction area that it is not possible for any tool to protrude into the retraction area while the revolver is moving. Ensure that the tool change point is relative to the zero point of the revolver and not the tool tip. 	
XT	Tool change point X Ø	










Parameter	Description	Unit
ZT	Tool change point Z	
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.	
Mach. direction of rotation 	Milling direction <ul style="list-style-type: none"> • Conventional • Climbing 	



Program header without multi-channel data

If a program is to be executed through one channel, then deselect multi-channel data. You then have the option of entering cross-program values into the program header as usual.

Parameter	Description	Unit
Multi-channel data	<ul style="list-style-type: none"> • No <p>This is only possible if you are not using a job list.</p>	
Unit of measurement 	The setting of the measurement unit in the program header only refers to the position data in the actual program. All other data, such as feedrate or tool offsets, are entered in the unit of measure that you have set for the entire machine.	mm inch
Data for 	<ul style="list-style-type: none"> • Main+counterspindle <p>All values for the main and counterspindle are saved in one data set</p> <ul style="list-style-type: none"> • Main spindle <p>Data set for the main spindle</p> <ul style="list-style-type: none"> • Counterspindle <p>Data set for the counterspindle</p> <p>If the machine does not have a counterspindle, then the entry field "Data for" is not applicable.</p>	
Work offset 	The work offset in which the zero point of the workpiece is saved. You can also delete the default value of the parameter if you do not want to specify a work offset.	
write to 	<ul style="list-style-type: none"> • Yes <p>Parameter ZV is displayed</p> <ul style="list-style-type: none"> • No <p>Parameter ZV is not displayed</p>	
ZV	Z value of the work offset For G54, the Z value is entered into the work offset. Note: Please observe the machine manufacturer's data	
Blank 	Define the form and dimensions of the workpiece:	



Parameter	Description	Unit
	<ul style="list-style-type: none"> Cylinder 	
XA	Outer diameter \varnothing	mm
	<ul style="list-style-type: none"> Polygon 	
N	Number of edges	
SW / L	Width across flats Edge length	
	<ul style="list-style-type: none"> Centered cuboid 	
W	Width of blank	mm
L	Length of blank	mm
	<ul style="list-style-type: none"> Tube 	
XA	Outer diameter \varnothing	mm
	Inner diameter \varnothing (abs) or wall thickness (inc)	mm
ZA	Initial dimension	mm
ZI 	Final dimension (abs) or final dimension in relation to ZA (inc)	mm
ZB 	Machining dimension (abs) or machining dimension in relation to ZA (inc)	
Retraction 	The retraction area indicates the area outside of which collision-free traversing of the axes must be possible.	
	<ul style="list-style-type: none"> simple 	
XRA 	Retraction plane X external \varnothing (abs) or retraction plane X referred to XA (inc)	
XRI 	- only for "pipe" blank Retraction plane X internal \varnothing (abs) or retraction plane X referred to XI (inc)	
ZRA 	Retraction plane Z front (abs) or retraction plane Z referred to ZA (inc)	
	<ul style="list-style-type: none"> extended - not for a "pipe" blank 	
XRA 	Retraction plane X external \varnothing (abs) or retraction plane X referred to XA (inc)	
XRI 	Retraction plane X internal \varnothing (abs) or retraction plane X referred to XI (inc)	
ZRA 	Retraction plane Z front (abs) or retraction plane Z referred to ZA (inc)	
	<ul style="list-style-type: none"> all 	
XRA 	Retraction plane X external \varnothing (abs) or retraction plane X referred to XA (inc)	
XRI 	Retraction plane X internal \varnothing (abs) or retraction plane X referred to XI (inc)	
ZRA 	Retraction plane Z front (abs) or retraction plane Z referred to ZA (inc)	
ZRI 	Retraction plant Z rear	



Parameter	Description	Unit
Tailstock 	<ul style="list-style-type: none"> • Yes • No 	
XRR	Retraction plane tailstock – only "Yes" for tailstock	
Tool change point 	<p>Tool change point, which must be approached by the revolver with its zero point.</p> <ul style="list-style-type: none"> • WCS (Workpiece Coordinate System) • MCS (Machine Coordinate System) <p>Notes</p> <ul style="list-style-type: none"> • The tool change point must be far enough outside the retraction area that it is not possible for any tool to protrude into the retraction area while the revolver is moving. • Ensure that the tool change point is relative to the zero point of the revolver and not the tool tip. 	
XT	Tool change point X \emptyset	
ZT	Tool change point Z	
S	Spindle speed	rev/min
Data for 	<ul style="list-style-type: none"> • Main spindle • Counterspindle. • Empty <p>If several spindles have been set-up, the program can operate at both spindles.</p>	
Retraction 	<p>The retraction area indicates the area outside of which collision-free traversing of the axes must be possible.</p> <ul style="list-style-type: none"> • simple • extended – not for a "pipe" blank • all 	
XRA 	Retraction plane X external \emptyset (abs) or retraction plane X referred to XA (inc)	
XRI 	- for "basic" retraction, only for a "pipe" blank retraction plane X internal \emptyset (abs) or retraction plane X referred to XI (inc)	
ZRA 	Retraction plane Z front (abs) or retraction plane Z referred to ZA (inc)	
ZRI 	Retraction plane Z rear – only for retraction "all"	
Tailstock 	<ul style="list-style-type: none"> • Yes • No 	
XRR	Retraction plane tailstock – only "Yes" for tailstock	

Parameter	Description	Unit
Tool change point 	Tool change point, which must be approached by the revolver with its zero point. <ul style="list-style-type: none"> WCS (Workpiece Coordinate System) MCS (Machine Coordinate System) Notes <ul style="list-style-type: none"> The tool change point must be far enough outside the retraction area that it is not possible for any tool to protrude into the retraction area while the revolver is moving. Ensure that the tool change point is relative to the zero point of the revolver and not the tool tip. 	
XT	Tool change point X \emptyset	
ZT	Tool change point Z	
S	Spindle speed	rev/min
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.	
Mach. direction of rotation 	Milling direction <ul style="list-style-type: none"> Conventional Climbing 	

Changing program settings

Under settings, the settings for the main and/or counterspindle can be changed while the program is being executed.

Parameter	Description	Unit
Data for	You define the spindle selection for processing the data here - (this is only available if the machine has a counterspindle) <ul style="list-style-type: none"> Main spindle Data set for the main spindle Counterspindle Data set for the counterspindle Main+counterspindle All values for the main and counterspindle are saved in one data set 	
Retraction 	Lift mode <ul style="list-style-type: none"> simple Extended all Empty 	
XRA 	Retraction plane X external \emptyset (abs) or retraction plane X referred to XA (inc)	mm

Parameter	Description	Unit
XRI 	Retraction plane X internal \varnothing (abs) or retraction plane X referred to XI (inc) - (only for retraction "extended" and "all")	mm
ZRA 	Retraction plane Z front (abs) or retraction plane Z referred to ZA (inc)	mm
ZRI	Retraction plane Z rear – (only for retraction "all")	mm
Tailstock	Yes <ul style="list-style-type: none"> Tailstock is displayed for simulation / simultaneous recording When approaching/retracting, the retraction logic is taken into account No	
XRR	Retraction plane – (only "Yes" for tailstock)	mm
Tool change point	Tool change point <ul style="list-style-type: none"> Work (Workpiece Coordinate System) Machine (Machine Coordinate System) Empty 	
XT	Tool change point X	mm
ZT	Tool change point Z	mm
SC	Safety clearance (inc) Acts in relation to the reference point. The direction in which the safety clearance is active is automatically determined by the cycle.	mm
S1	Maximum speed, main spindle	rev/min
Machining direction	Milling direction: <ul style="list-style-type: none"> Climbing Conventional Empty 	

Procedure

1. The ShopTurn program has been created.
2. Position the cursor on the program header.
3. Press the "Various" and "Settings" softkeys.
The "Settings" input window opens.

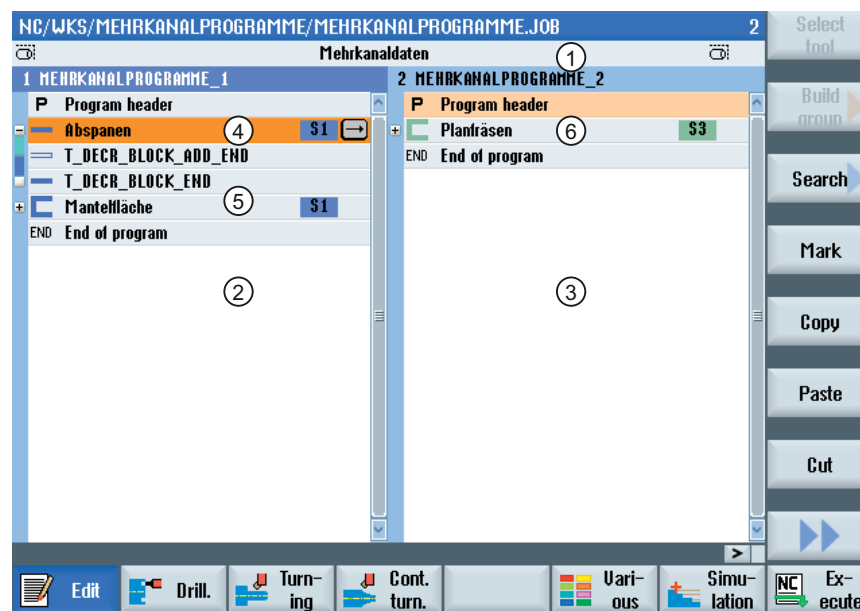


10.2.3.4 Creating a program block

In order to structure programs in order to achieve a higher degree of transparency when preparing for the synchronized view, you have the possibility of combining several blocks (G code and/or ShopTurn machining steps) to form program blocks.

Structuring programs

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



- 1) Cross-channel data from the "Multi-channel data" window.
- 2) "Multi-channel programs" program opened in channel 1.
- 3) "Multi-channel programs" program opened in channel 2.
- 4) Actual program with block name "Stock removal".
The program block has been opened and an Addit. run-in code has been activated.
The program block is assigned to the main spindle.
- 5) Program block with block name "Peripheral surface".
The program block is closed. In order to identify whether an Addit. run-in code is activated or automatic retraction is activated, open the block using the <Cursor right> key.
- 6) Program block with block name "Face milling"
The program block is assigned to the counterspindle. The spindle assignment is color coded in order to make a distinction.

Figure 10-1 Structured programs in the multi-channel editor

Settings for a program block

Display	Meaning
Text	Block designation
Spindle	<ul style="list-style-type: none"> • S1 • S2 <p>Spindle assignment. Defines at which spindle a program block is to be executed.</p>
Addit. run-in code	<ul style="list-style-type: none"> • Yes <p>For the case that the block is not executed, as the specified spindle should not be processed, then it is possible to temporarily activate a so-called "Addit. run-in code".</p> <ul style="list-style-type: none"> • No
Automat. retraction	<ul style="list-style-type: none"> • Yes <p>Block start and block end are moved to the tool change point, i.e. the tool is brought into a safe range.</p> <ul style="list-style-type: none"> • No

Procedure



1. Select the "Program manager" operating area.



2. Select the storage location and create a program or open a program. The program editor is opened.



3. Select the required program blocks, which you wish to combine to form a block.



4. Press the "Build group" softkey. The "Build group" window is opened.



5. Enter a designation for the block, assign the spindle, if required, select the Addit. run-in code and the automatic retraction and then press the "OK" softkey.

Opening and closing blocks



...



...



1. Position the cursor on the desired program block.
2. Press the <+> key or the <Cursor right> key.
The block is opened.

3. Press the <-> key or the <Cursor left> key.
The block is closed again.

Shifting blocks

You have the option of using "Select", "Copy", "Cut-out" and "Paste" softkeys to move individual or several blocks within the program.

10.2.4 Synchronizing programs

Using the synchronized view, you have the possibility of obtaining an overview of the time sequence of a program. In this case, program instructions are evaluated to coordinate channels and are arranged in parallel in the editor view.

As a result of the synchronized view of the programs, you can easily identify at which locations the programs are synchronized in the various channels.

Synchronizing commands



Commands	Meaning
START	Starts another program
WAITM	Sets a mark and waits for the specified channels (with exact stop)
WAITMC	Sets a mark and waits for the specified channels (without exact stop)
WAITE	Waits for the end of program of the specified channels
SETM	Sets a mark
CLEARM	Clears a mark
GET	Fetches an axis
RELEASE	Releases an axis

Note

Identifying program errors

The error can only be identified if the appropriate program is displayed. If a WAIT mark involves a channel to which a program is not assigned in the job list, then this is marked as having an error.

If you have enabled the synchronized view, then the following symbols are displayed at the top right in the program title bar:

Symbol	Meaning
	Synchronized view: Error Errors were identified after enabling "synchronized view" or new "synchronization" (e.g. WAIT marks not found in other programs).
	Synchronized view: checked No error was detected after enabling "synchronized view" or new "synchronization".

Wait marks can also be used within blocks.

- Closed block
 - If there is a WAIT mark within a closed block, the clock of this WAIT mark is displayed in front of the block name.
In the synchronized view, the closed block is synchronized.
 - If there are several WAIT marks within a closed block, then a clock is displayed in front of the block name. It is yellow, if all clocks of the WAIT marks in the block are yellow, otherwise it is red.
In the synchronized view, the closed block is synchronized to the last WAIT mark in the block.
- Opened block
 - If there is a WAIT mark, the clock is displayed in front of the WAIT mark.
In front of the WAIT mark, the clock is displayed in either yellow or red. The program is synchronized at the WAIT mark.
 - If there are several WAIT marks, the clocks are displayed in either yellow or red at the WAIT marks. The program is synchronized at the WAIT marks.

Determining program runtimes

After a simulation, the required processing time for the program blocks is displayed in the editor. For multi-channel programs, the delay time is displayed at the wait points.

Spindle transfer between the channels

If you use the spindles alternating in several channels (e.g. main and counterspindle), then it may be necessary to exit the active plane "Face C" (TRANSMIT) or "Peripheral C" (TRACYL):

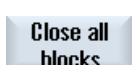


1. Press the menu forward key and the "Straight/Circle" softkey.



2. Press the "Tool" softkey.
3. In the "Plane selection" field, select the "Turning" setting (TRAFOOF).

Procedure



1. Select the required job list.
2. Press the "Open" softkey.
The job list is opened in the editor.
3. Press the ">>" and "View" softkeys.
4. Press the "Synchron. view" softkey.
5. Press the "Synchronizing" softkey if you wish to update the view after changes.
6. Press the "Open all blocks" softkey if you wish to view all of the program blocks in the synchronized display.
7. Press the "Close all blocks" softkey, if you wish to close the blocks to achieve a higher degree of transparency.
8. Select the required program.
9. Press the "Full screen" softkey.
The two-channel display is changed into a single-channel display and the selected program is displayed using the complete editor window.

See also

Optimizing the machining time (Page 559)





10.2.5 Optimizing the machining time

After simulating a program, the machining time is displayed for the blocks.

For a multi-channel display, the wait times that occur are displayed at the wait points (wait marks). This provides you with an overview of the time sequence of the program and you can perform the first optimization runs.

Shifting blocks

For instance, you have the option of shifting program blocks to longer wait points - assuming that the machining technology permits this - therefore reducing the machining time.

1. Select the block that you wish to shift.
2. Press the "Mark" softkey.

3. Press the "Copy" softkey if you wish to repeat the machining step at another position.

- OR -
Press the "Cut-out" softkey if you wish to execute the machining step at another position.

4. Position the cursor at the required program position and press the "Paste" softkey.

Insert the program block at the required position.

Time-related view

The wait time at the wait marks is displayed in seconds.

The total runtime is displayed at the end of the program.

If you change the program, time data is still displayed at the corresponding wait marks or the corresponding blocks dimmed.

The determined times are kept if you change, shift or copy the program blocks.

The determined times are lost if you exit the editor using the "Close" softkey or you open or select another program. The data must be determined again through simulation.

Time bars

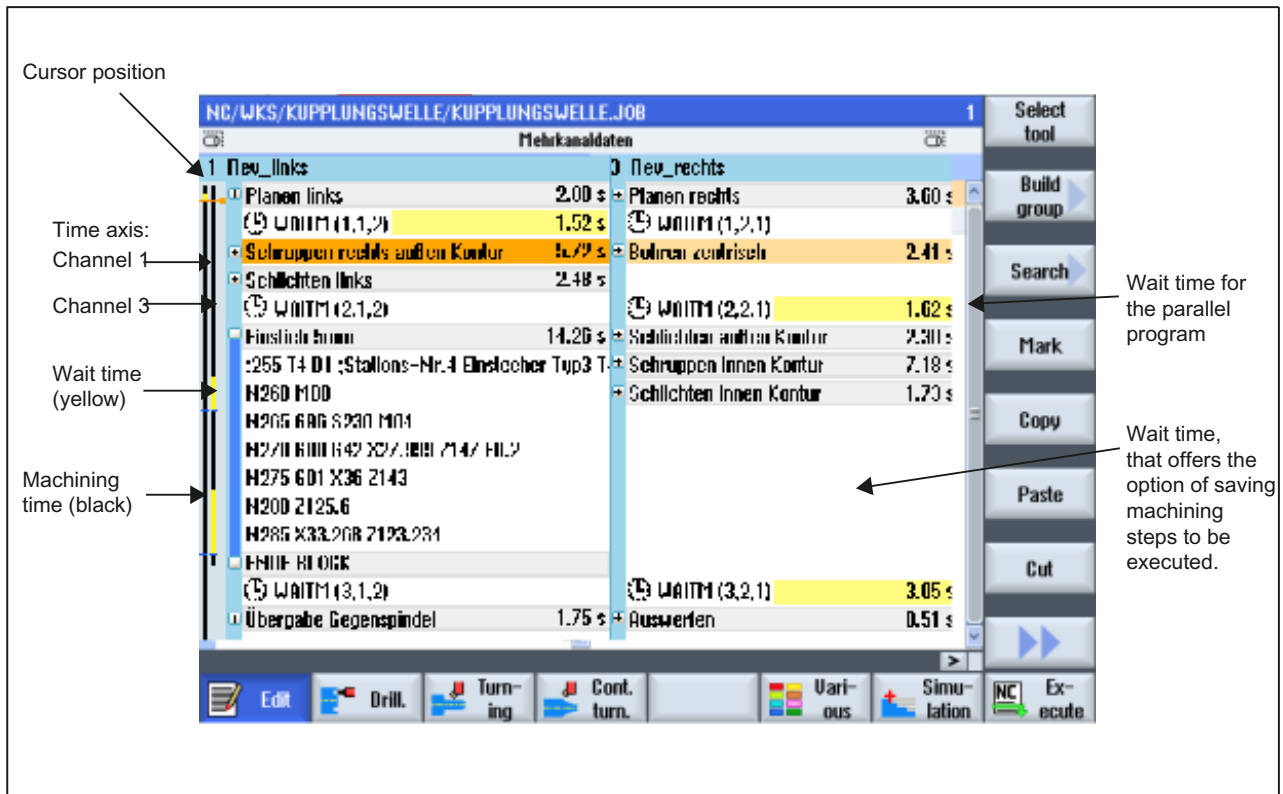


Figure 10-2 Time synchronous view

10.3 Simulating machining

10.3.1 Simulation

For classic lathes with main and counterspindle, up to two channels can be simultaneously simulated.

You have the option of executing the programs together before the actual machining. In this case, start, stop and reset - as well as the functions to control the program, act simultaneously on all of the simulated channels.



Software options

You require the "programSYNC" option to simultaneously simulate several channels.

Without this option, all of the channels are considered to be an autonomous system and are individually simulated.

Correction of the zero offsets for multi-channel data

If you use multi-channel data in the simulation, then the zero offsets are corrected so that they match the blank and the spindle chuck data.

Requirements

- The function of the individual spindles and special axes, must be specified in the channel-specific display machine data set-up for the purpose.
- The zero point of the counter-spindle must be the same as that of the main spindle.










Machine manufacturer

Please refer to the machine manufacturer's specifications.

Tool paths

Only the tool paths of the presently selected channel are displayed.

Procedure

1. Start the simulation.
 2. Press the "Main spindle" softkey if you wish to view the main spindle.

 - OR -
 3. Press the "Counterspindle" softkey if you wish to view the counterspindle.

 4. If you have pressed the "Main spindle" and "Counterspindle" softkeys, the simulation view is split into two and you can see the main spindle and counterspindle simultaneously.


 5. To toggle between the various channels, press the ">>" softkey and the "Channel +" or "Channel -" softkey.



- The status line informs you about the channel presently being processed.







10.3.2 Different workpiece views for multi-channel support

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:

- Side view
- Half section
- Front view
- 3D view
- 2-window

Procedure

1. Start the simulation.
2. Press the "Views" softkey.

3. Press the "Side view" softkey if you wish to view the workpiece in the Z-X plane.

- OR -
Press the "Half section" softkey if you wish to view the workpiece cut in the Z-X plane.

- OR -
Press the "Front view" softkey if you wish to view the workpiece in the X-Y plane.

- OR -
Press the "3D view" softkey if you wish to view the workpiece as a three-dimensional model.

- OR -
Press the 2-window softkey if you wish to simultaneously view the side view (left-hand window) and front view (right-hand window) of the workpiece.


Note

The 2-window view cannot be activated if you have simultaneously selected the main spindle and the counterspindle.

10.4 Display/edit the multi-channel functionality in the "Machine" operating area

10.4.1 Running-in a program

You have various options to run-in programs.

Running-in channel-by-channel

Select the channels that you wish to process using the "Running-in" function in the "Program control" window. The channels not selected here are brought into the "Program test (PRT)" state. As a consequence, the channels are only calculated, but are not processed.

No M and auxiliary functions or tool functions are output. Spindle commands are only output for selected spindles.

Running-in spindle-by-spindle

Only machining operations are executed for a spindle selected under program control / running-in. When programming, using the block formation, allocate the corresponding machining operations to a spindle.

When generating a block, a complete block can be assigned to a spindle. For the case that the block is not executed, as the specified spindle cannot be processed, then save a so-called "running-in – supplementary code"



Software options

You require the "programSYNC" option for cross-channel program control.

Precondition

- Multi-channel machine
- You have selected the multi-channel view via "Settings for channel functionalities".

Procedure



1. Select the "AUTO" operating area.
2. Press the "Prog. ctrl." softkey.



The "Program control - General" window appears on the screen.



3. Press the "Run-in" softkey.
The "Program control - Running-in" window is displayed.
4. Select the channels and the associated spindles to run-in the program.

10.4.2 Block search and program control

You define a group of channels that belong to one another from the "Settings for Multi-channel Functionality" window. Here, you specify which channel numbers should be displayed for a multi-channel view.

This group results in a common behavior for a block search and for program control.

Vertical softkeys for a block search

- The "Block search" and "Search mode" functions act on all channels, which you configured for the multi-channel view.
- All other vertical softkeys (e.g. "Search text", "Interrupt position", etc.) act on the current program.

If you selected the single-channel view in the multi-channel function settings, then all of the actions only act on the actual channel.



Software options

You require the "programSYNC" option for the multi-channel block search and the multi-channel program control in the multi-channel editor.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure



1. Press the "Block search" softkey.



Press the "Search mode" softkey.
The "Search Mode" window is displayed.



2. Select the required mode for the channel group.
3. Press the "OK" softkey to confirm the setting.



4. Press the "Search text" softkey if you wish to enter the search target using a text.

- OR -



Press the "Interrupt position" softkey if you wish to search for the search target using a program interrupt.

- OR -



Press the "Search pointer" softkey if you wish to enter a search target that you cannot enter using the editor (e.g. no interrupt position, search target is located in the subprogram) and enter the program path.



5. Press the "Start search" softkey.

The search starts.

All channels of the group are started corresponding to the search mode that has been set.

During the block search, the search states are displayed in a message window (e.g. "Block search running").

You receive a message if the search target has been reached - or an error message if the search target was not found.

Note

Search target via "Search pointer"

The "Search Pointer" window is displayed - as usual - over the complete lower part of the screen. The upper contents of the window remain multi-channel.

The actual channel is displayed in the title bar of the "Search Pointer" window.

The "Delete search pointer" also only acts on this channel.

Note

Search target via "Search text"

If, for a block search, you approach the program position using "Search text", please note that a search is only made in the selected channel column.

10.5 Multi-channel functionality for large operator panels

For the large OP15, OP019 operator panels as well as at the PC, there is more space in the "Machine", "Program" and "Parameter" operating areas – as well as in all lists – to display NC blocks, tools etc.

Further, you also have the option of simultaneously displaying more than 2 channels.

This makes it easier for you to identify the machine situation for machines with 3 and more channels. Further, it makes it simpler for you to generate and run-in three or four-channel programs.



Software options

If you require the option "programSYNC" for the views described here.

Supplementary conditions

- OP15, OP19 or PC with a display of at least 1280x1024 pixels
- For operating an OP019, at least one NCU720.2 or 730.2 with 1 GB of RAM or a PCU50 is required

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

If you have selected 3 channels via settings, then 3 or 4 channel columns are displayed next to one another.

Channel view	Display in the "Machine" operating area
3-channel view	The following windows are displayed one above the other for each channel: <ul style="list-style-type: none"> • Actual value window • T,F,S window • Block display window
4-channel view	The following windows are displayed one above the other for each channel: <ul style="list-style-type: none"> • Actual value window • T,S,F window • G functions (the "G functions" softkey is omitted) • Block display window

Displaying functions

Channel view	Display in the "Machine" operating area
	Selection using vertical softkeys:
3-channel view	<ul style="list-style-type: none"> The T,F,S window is overlaid by pressing one of the vertical softkeys.
4-channel view	<ul style="list-style-type: none"> The window showing the G codes is overlaid if you press one of the vertical softkeys.
	Selection using horizontal softkeys:
3-channel view / 4 channel view	<ul style="list-style-type: none"> The block display is overlaid if you press the "Overstore" horizontal softkey
	<ul style="list-style-type: none"> The block display is overlaid by pressing the softkey "block search".
	<ul style="list-style-type: none"> The window is shown as a pop-up if you press the "Prog. control" softkey.
	<ul style="list-style-type: none"> If you press one of the horizontal softkeys in the "JOG" operating mode (e.g. "T,S,M", "Meas. tool", "Positions" etc.), then you change into the single-channel view.

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

Contrary to the smaller operator panels, in the "Machine" operating area, for a 2-channel view, the TFS window is visible.

Program operating area

In the editor, just as many programs are displayed next to one another as in the "Machine" operating area.

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 display the column with the active program wider.

Simulation

In the simulation window, actual values are displayed for a maximum of 4 channels simultaneously as well as the actual block.

You can toggle between displaying the traversing paths and the channel zero point using the "Channel+" and "Channel-" softkeys.

Axes, which are located in several channels, are displayed grayed-out if the setpoint comes from a different channel.

Channel status

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



Machine manufacturer

Please refer to the machine manufacturer's specifications.

10.6 Setting the multi-channel function

Setting	Meaning
View	Here, you define whether one or two channels are displayed. <ul style="list-style-type: none"> • 1 channel • 2 channels • 3 channels • 4 channels
Channel selection and sequence (for "2 - 4 channels" view)	Here, you create the channel group, i.e. you specify which channels and 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 two-channel view.

Precondition



Software options

You require the "programSYNC" option to generate and edit synchronized programs in the multi-channel editor as well as for the multi-channel functions in the "Machine" operating area.

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).

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.

Procedure



1. Select the "Machine" operating area.



2. Select the operating mode "JOG", "MDA" or "AUTO".



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



4. Press the "Multi-channel function" softkey.
The "Settings for the multi-channel functionality" window is opened.



10.7 Synchronizing a counterspindle

For multi-channel machines, the counterspindle steps must be synchronized across all channels.

You program handling the counterspindle in one channel. This channel controls the motion of the counterspindle and adapts the zero offset of the channel.









In the synchronization step, the other channels park their tools in order to avoid collisions. The zero offset is also accepted.


NC/WKS/GEGENSP_2KAN/GEGENSP_2KAN.JOB		1
Mehrkanaldaten		
CHAN1	GEGENSP_2KAN_1	CHAN2
P	N10 Program header	P N10 Program header
	N20 Stock removal T=SCHRI	N11 waitm(1, 1, 2)ff
✓	N21 waitm(1, 1, 2)ff	N30 Counterspindle ② Complete
⊠	N30 Counterspindle ① Synchron	N40 Cutoff T=EINSTI
✓	WAITM(1, 1, 2) ff	N50 Drilling centric T=BOHRE
END	End of program	waitm(1, 1, 2)ff
		END
		End of program

- 1 Synchronization step
- 2 Counterspindle step

The following counterspindle steps, which implicitly contain WAIT marks, are identified using a symbol:



- Pulling (take zero point = yes)
- Rear face
- Front face
- Complete
- Synchronizing


Parameters	Description	Unit
Function 	You can select one of the following functions: <ul style="list-style-type: none"> • Synchronizing • Complete • Gripping • Withdrawing • Rear face • Front face 	
Synchronization function	Synchronizes with the counterspindle in the other channel.	
Coordinate system	<ul style="list-style-type: none"> • MCS The park position is specified in the machine coordinate system. Teaching in the park position and angular offset is only possible in the machine coordinate system. • Workpiece coordinate system The park position is specified in the workpiece coordinate system. 	
XP	Park position of tool in X direction (abs)	mm
ZP	Park position of tool in Z direction (abs)	mm
Function, complete	Gripping	
Coordinate system 	<ul style="list-style-type: none"> • MCS The park position is specified in the machine coordinate system. Teaching in the park position and angular offset is only possible in the machine coordinate system. • WCS The park position is specified in the workpiece coordinate system. 	
XP	Park position of tool in X direction (abs)	mm
ZP	Park position of tool in Z direction (abs)	mm
Flush chuck 	Flush counterspindle chuck <ul style="list-style-type: none"> • Yes • No 	
DIR 	Direction of rotation <ul style="list-style-type: none"> •  Spindle rotates clockwise •  Spindle rotates counter-clockwise •  Spindle does not rotate 	
S	Spindle speed – (only when the spindle rotates)	rev/min
$\alpha 1$	Angular offset	Degrees
Z1	Transfer position (abs.)	
ZR 	Position, feedrate reduction (abs or inc) Position from which a reduced feedrate is used.	
FR	Reduced feedrate	mm/rev


Parameters	Description	Unit
Fixed stop	Travel to fixed stop <ul style="list-style-type: none"> • Yes The counterspindle stops at a defined distance away from transfer position Z1 and then traverses with a defined feedrate up to the fixed stop. • No The counterspindle traverses to the transfer position Z1. 	
Function, complete	Withdrawing	
Withdraw blank	Withdraw complete blank: <ul style="list-style-type: none"> • Yes • No 	
F	Feedrate	mm/min
Cutting-off cycle	Cutting-off cycle in the following block <ul style="list-style-type: none"> • Yes • No 	
Function, complete	Rear face	
Zero offset 	Zero offset in which the coordinate system, which was shifted according to ZW and by ZV as well as mirrored in Z, must be saved: <ul style="list-style-type: none"> • Basic reference • G54 • G55 • G56 • G57 • ... 	
ZW	Machining position for special axis (abs.); machine coordinate system	mm
ZV	Offset Z = 0 Workpiece zero is offset in Z direction (inc, the sign is also evaluated)	mm

Parameters	Description	Unit
Function, gripping	Teaching in the park position and angular offset is possible	
Coordinate system U	<ul style="list-style-type: none"> MCS The park position is specified in the machine coordinate system. Teaching in the park position and angular offset is only possible in the machine coordinate system. WCS The park position is specified in the workpiece coordinate system. 	
XP	Park position of tool in X direction (abs)	mm
ZP	Park position of tool in Z direction (abs)	mm
Flush chuck U	Flush counterspindle chuck <ul style="list-style-type: none"> Yes No 	
DIR U	Direction of rotation <ul style="list-style-type: none"> ↻ Spindle rotates clockwise ↺ Spindle rotates counter-clockwise ⊗ Spindle does not rotate 	
S	Spindle speed – (only when the spindle rotates)	rev/min
α1	Angular offset	Degrees
Z1	Transfer position (abs.)	
ZR U	Position, feedrate reduction (abs or inc) Position from which a reduced feedrate is used.	
FR	Reduced feedrate	mm/rev
Fixed stop	Travel to fixed stop <ul style="list-style-type: none"> Yes The counterspindle stops at a defined distance away from transfer position Z1 and then traverses with a defined feedrate up to the fixed stop. No The counterspindle traverses to the transfer position Z1. 	

10.7 Synchronizing a counterspindle

Parameters	Description	Unit
Function, withdrawing		
Also take zero point 	Also take zero point <ul style="list-style-type: none"> • Yes • No 	
Zero offset 	Zero offset in which the coordinate system offset by Z1 must be saved. <ul style="list-style-type: none"> • Basic reference • G54 • G55 • G56 • G57 • ... 	
Z1	Amount by which the workpiece is withdrawn from the main spindle (inc)	
F	Feedrate	mm/min

Parameters	Description	Unit
Function, rear face		
Zero offset 	Zero offset in which the coordinate system, which was shifted according to ZW and by ZV as well as mirrored in Z, must be saved: <ul style="list-style-type: none"> • Basic reference • G54 • G55 • G56 • G57 • ... 	
ZW	Machining position for special axis (abs.); machine coordinate system	mm
ZV	Offset Z = 0 Workpiece zero is offset in Z direction (inc, the sign is also evaluated)	mm

Parameters	Description	Unit
Function, front face		
Zero offset 	Zero offset for machining the next front face: <ul style="list-style-type: none"> • Basic reference • G54 • G55 • G56 • G57 • ... 	

User variables

11.1 Overview

The defined user data may be displayed in lists.

The following variables can be defined:

- Data parameters (R parameters)
- Global user data (GUD) is valid in all programs
- Local user data (LUD) is valid in one program
- Program-global user data (PUD) is valid in one program and the called subroutines.

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 also refer to the machine manufacturer's instructions.

Note

Reading and writing variables protected

Reading and writing of user data are protected via a keyswitch and protection levels.

Searching for user data

You may search for user data within the lists using any character string.

Refer to the "Defining and activating user data" section to learn how to edit displayed user data.

11.2 R parameters

R parameters (arithmetic parameters) are channel-specific variables that you can use within a G code program. G code programs can read and write R parameters.

These values are retained after the controller is switched off.

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).

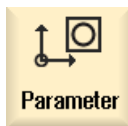
There are no gaps in the numbering within the range.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure



1. Select the "Parameter" operating area.



2. Press the "User variable" softkey.



3. Press the "R variables" softkey.
The "R Parameters" window appears.

Delete R variables



1. Press the ">>" and "Delete" softkeys.
The "Delete R parameters" window appears.



2. Enter the R parameter(s) whose channel-specific values you would like to delete and press the "OK" softkey.

A value of 0 is assigned to the selected R parameters or to all R parameters.

11.3 Displaying global user data (GUD)

Global user variables

Global GUDs are NC global user data (**Global User Data**) 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)

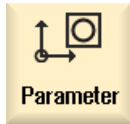
Example

```
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.

11.4 Displaying channel GUDs

Channel-specific user variables

Like the GUDs, channel-specific user variables are applicable in all programs for each channel. However, unlike GUDs, they have specific values.

Definition

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

- Keyword DEF
- Range of validity CHAN
- Data type
- Variable names
- Value assignment (optional)

Example

```
DEF CHAN REAL X_POS = 100.5
```

Procedure



1. Select the "Parameter" operating area.



2. Press the "User variable" softkey.



3. Press the "Channel GUD" and "GUD selection" softkeys.



A new vertical softkey bar appears.



4. Press the "SGUD" ... "GUD6" softkeys if you want to display the SGUD, MGUD, UGUD as well as GUD4 to GUD 6 of the channel-specific user variables.



- OR -



Press the "Continue" softkey and the "GUD7" to "GUD9" softkeys if you want to display GUD 7 to GUD 9 of the channel-specific user variables.



11.5 Displaying local user data (LUD)

Local user variables

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

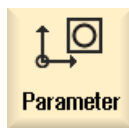
The controller displays the LUDs after the start of program processing. The display is available until the end of program processing.

Definition

A local user variable is defined with the following:

- Keyword DEF
- Data type
- Variable names
- Value assignment (optional)

Procedure



1. Select the "Parameter" operating area.



2. Press the "User variable" softkey.



3. Press the "Local LUD" softkey.

11.6 Displaying program user data (PUD)

Program-global user variables

PUDs are global part program variables (**Program User Data**). 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.

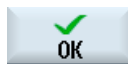
11.7 Searching for user variables

You can search for R parameters and user variables.

Procedure



1. Select the "Parameter" operating area.



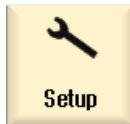
2. Press the "R parameters", "Global GUD", "Channel GUD", "Local GUD" or "Program PUD" softkeys to select the list in which you would like to search for user variables.
3. Press the "Search" softkey.
The "Search for R Parameters" or "Search for User Variables" window opens.
4. Enter the desired search term and press "OK".

The cursor is automatically positioned on the R parameter or user variables you are searching for, if they exist.

11.8 Defining and activating user variables

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.



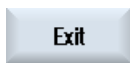
- OR -
Press the "Open" softkey.



- 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.

Teaching in a program

12.1 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 "AUTO" teach-in mode, the selected program is "taught".

In "MDA" teach-in mode, you teach to the MDA buffer.

External programs, which may have been rendered offline, can therefore be adjusted and modified according to need.

12.2 General sequence

General sequence

Select the desired program block, press the relevant softkey "Teach position", "Rap. tra. G01", "Straight line G1" or "Circ. interm. pos. CIP", and "Circ. end pos. CIP" and traverse the axes to change the program block.

You can only overwrite a block with a block of the same type.

- OR -

Position the cursor at the desired point in the program, press the relevant softkey "Teach position", "Rap. tra. G01", "Straight line G1" or "Circ. interp. pos. CIP", and "Circ. end pos. CIP" and traverse the axes to insert a new program block.

In order for the block to be inserted, the cursor must be positioned in an empty line using the cursor key and input key.

Press the "Accept" softkey to teach-in the modified or new program block.

Note

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.

Operating mode or operating area switchover

If you switch to another operating mode or operating area in teach-in mode, the position changes will be canceled and teach-in mode will be cleared.

12.3 Inserting a block

You have the option of traversing the axes and writing the current actual values directly to a new position block.

Requirement

"AUTO" mode: The program to be edited is selected.

Proceed as follows



1. Select the "Machine" operating area.



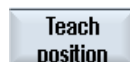
2. Press the <AUTO> or <MDA> key.



3. Press the <TEACH IN> key.



4. Press the "Teach prog." softkey.




5. Traverse the axes to the relevant position.

6. Press the "Teach position" softkey.

A new program block with the current actual position values will be created.

12.3.1 Input parameters for teach-in blocks

Parameters for teach-in of position and teach-in of G0, G1, and circle end position CIP

Parameter	Description
X	Approach position in X direction
Y	Approach position in Y direction
Z	Approach position in Z direction
F	Feedrate (mm/r; mm/min) - only for teach-in of G1 and circle end position CIP
	

Parameters for teach-in of circle intermediate position CIP

Parameter	Description
I	Coordinate of the circle center point in the X direction
J	Coordinate of the circle center point in the Y direction
K	Coordinate of the circle center point in the Z direction

Transition types for teach-in of position and teach-in of G0 and G1, and ASPLINE

The following parameters are offered for the transition:

Parameter	Description
G60	Exact stop
G64	Corner rounding
G641	Programmable corner rounding
G642	Axis-specific corner rounding
G643	Block-internal corner rounding
G644	Axis dynamics corner rounding

Motion types for teach-in of position and teach-in of G0 and G1

The following motion parameters are offered:

Parameter	Description
CP	Path-synchronous
PTP	Point-to-point
PTPG0	Only G0 point-to-point

Transition behavior at the beginning and end of the spline curve

The following motion parameters are offered:

Parameter	Description
Start	
BAUTO	Automatic calculation
BNAT	Curvature is zero or natural
BTAN	Tangential
End	
EAUTO	Automatic calculation
ENAT	Curvature is zero or natural
ETAN	Tangential

12.4 Teach-in via Windows

12.4.1 General

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 were already traversed before the window was selected.

All data from the input/output fields are transferred to the program via the "Accept" softkey.

Precondition

"AUTO" mode: The program to be edited is selected.

Procedure



1. Select the "Machine" operating area.



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



3. Press the <TEACH IN> key.



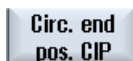
4. Press the "Teach prog." softkey.

5. Use the cursor and input keys to position the cursor at the desired point in the program.

If an empty row is not available, insert one.



6. 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.

7. Traverse the axes to the relevant position.



8. Press the "Accept" softkey.
A new program block will be inserted at the cursor position.

- OR -



Press the "Cancel" softkey to cancel your input.

12.4.2 Teach in rapid traverse G0

You traverse the axes and teach-in a rapid traverse block with the approached positions.

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.

12.4.3 Teach in straight G1

You traverse the axes and teach-in a machining block (G1) with the approached positions.

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.

12.4.4 Teaching in circle intermediate and circle end point 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.

Note

Selection of axes for teach in

You can select the axes to be included in the teach-in block in the "Settings" window.

12.4.5 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.

Note

The relevant option bit must be set to enable you to program a spline interpolation.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

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.



5. Press the ">>" and "ASPLINE" softkeys.
The "Akima-spline" window opens with the input fields.



6. Traverse the axes to the required position and if necessary, set the transition type for the starting point and end point.



7. Press the "Accept" softkey.
A new program block will be inserted at the cursor position.

- OR -



Press the "Cancel" softkey to cancel your input.

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.

12.5 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 alphanumeric input.

Requirement

The program to be processed is selected.

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.



5. Click the program block to be edited.



6. Press the relevant softkey "Teach position", "Rap. tra. G0", "Straight line G1", or "Circ. interm. pos. CIP", and "Circ. end pos. CIP".

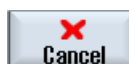
The relevant windows with the input fields are displayed.



7. 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.

12.6 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.

Requirement

The program to be processed is selected.

Proceed as follows



1. Select the "Machine" operating area.



2. Press the <AUTO> key.



3. Press the <TEACH IN> key.



4. Press the "Teach prog." softkey.



5. Place the cursor on the desired program block.
6. Press the "Block selection" softkey.

12.7 Deleting a block

You have the option of deleting a program block entirely.

Requirement

"AUTO" mode: The program to be processed is selected.

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.



5. Click the program block to be deleted.



6. Press the ">>" and "Delete block" softkeys.

The program block on which the cursor is positioned is deleted.

12.8 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.

Proceed as follows



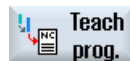
1. Select the "Machine" operating area.



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



3. Press the <TEACH IN> key.



4. Press the "Teach prog." softkey.



5. Press the ">>" and "Settings" softkeys.
The "Settings" window appears.



6. Under "Axes to be taught" and "Parameters to be taught", select the check boxes for the relevant settings and press the "Accept" softkey to confirm the settings.

Working with Manual Machine

13.1 Manual Machine

"Manual Machine" offers a 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 "ShopMill/ShopTurn" option for working with "Manual Machine"

Machining

Machining before the center of rotation is typical.

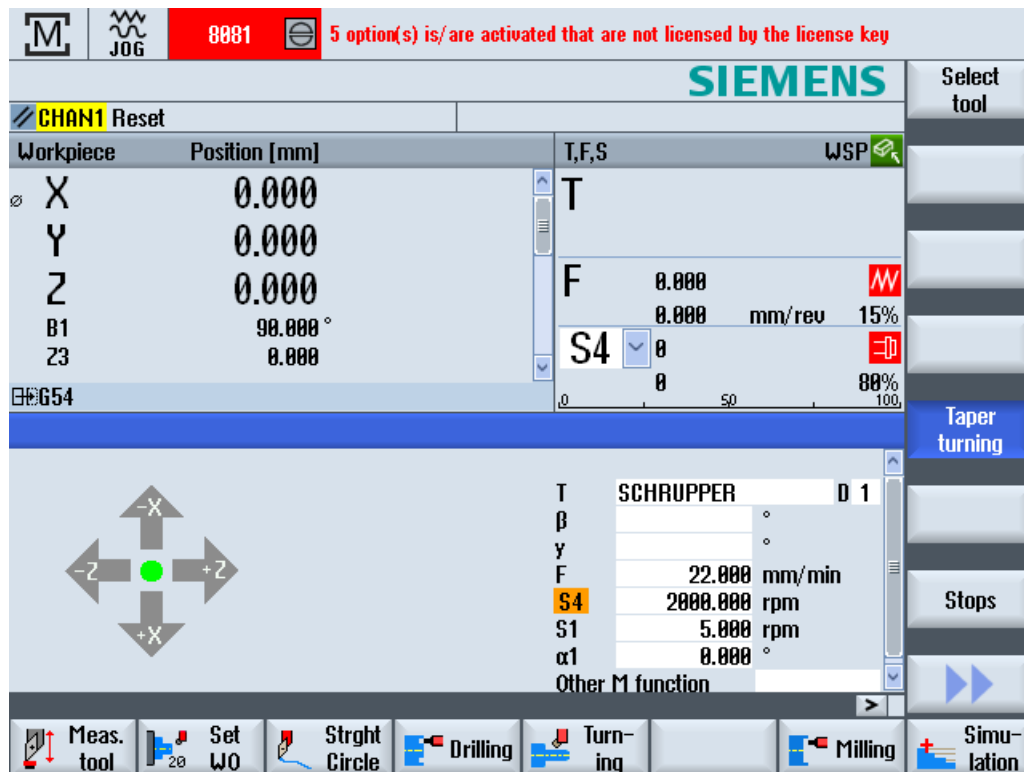


Machine manufacturer

Please refer to the machine manufacturer's specifications.

Main screen

After power-up of the control, the basic screen appears for "Manual Machine".



Machining options

You have the following options for machining the workpieces:

- Manual mode
- Single-cycle machining

13.2 Measuring the tool

All the options of the manual and automatic measurement are available to determine the tool offset data (see also Section "Measuring the tool (Page 75)").

Procedure



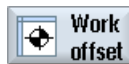
1. "Manual Machine" is active.
2. Press the "Meas. tool" softkey.
3. Select the the required measuring function via the vertical softkey bar and press the appropriate softkey.

13.3 Setting the zero offset



As an alternative to the "Set zero offset" function, you can select the values of the zero offset directly in the zero offset list.

Procedure



1. "Manual Machine" is active.
2. Press the "Zero offset" softkey in the "Parameters" operating area.



The "Zero Offset G54 ...599" window is opened.

See also

Setting the zero offset (Page 73)

13.4 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

Entered and activated limit stops remain active after switching from JOG mode to the MDA or AUTO mode.

Procedure

1. "Manual Machine" is active
2. Press the "Limit stops" softkey.
The "Limit Stops" window appears.

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.


13.5 Simple workpiece machining

In "Manual Machine", you machine workpieces directly without creating a program.

Functions

The following functions are available to you for machining in manual mode:

- Axis movements
- Taper turning
- Straight (face and longitudinal turning) and circle

Note




Tool, spindle speed and direction of spindle rotation are activated with <CYCLE START>.

A change in feedrate immediately becomes active.

13.5.1 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.
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.

13.5.2 Taper turning

The basic effective direction can be selected via the axis direction keys or via the cross-switching lever. In addition, a taper angle (α_1) can also be entered.

Procedure




1. "Manual Machine" is active.
2. Press the "Taper turning" softkey.
3. Select the tool, spindle, and spindle direction and specify the machining feedrate.
4. Enter the desired angle α_1 .

Note

Selecting/deselecting the taper turning and changing the angle α_1 is only possible in the reset state.

Parameter	Description	Unit
T	Tool name	
D	Cutting edge number	
F	Feedrate	mm/min mm/rev







Parameter	Description	Unit
S / V 	Spindle speed or constant cutting rate	rev/min m/min
$\alpha 1$	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.	







13.5.3 Straight and circular machining

13.5.3.1 Straight turning

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 "All axes" softkey

 - OR -
 - Press the softkey "Straight X α "

 - OR -
 - Press the softkey "Straight Z α "

 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
	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
C 	Target position of C axis of main spindle (abs or inc)	mm
Z2 	Target position of an added axis, if it exists (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

13.5.3.2 Circular turning

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.

Parameters

Parameter	Description	Unit
F 	Feedrate	mm/min mm/rev
Circle input 	<ul style="list-style-type: none"> • End point + center point • End point + radius 	
Direction of rotation 	 Clockwise direction of rotation  Counter-clockwise direction of rotation	
X 	Target position in the X direction (abs and inc)	mm
Z 	Target position in the Z direction (abs and inc)	mm
I	Circle center point I (inc) - only if circle input via end point and center point	mm
K	Circle center point K (inc) - only if circle input via end point and center point Note: Incremental dimensions: The sign is also evaluated.	mm
R	Radius - only if circle input via end point and radius	mm

13.6 More complex machining

The following functions are available to you for comprehensive machining in manual mode:

- Drilling (center drilling, centering, drilling, reaming, deep-drilling, threads, positions)
- Turning (stock removal, groove, undercut, threads, tapping)
- Milling (face milling, pocket, spigot, multiple edge, groove, thread milling, engraving)

General sequence

For more complex machining processes, proceed in the following order:

- Select the desired function via the corresponding softkey.
- Select the tool and enter the desired values in the parameter screen.
- Press the "Accept" softkey to save the values.

The input screen form 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 form at any time to check and correct the inputs.

Press the <"Cursor right"> key to jump back to the input screen form.

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, enter the desired values in the parameter screen and press the "Accept" softkey to confirm the technology block.

The input screen is closed and the line with the technology data is displayed in the main screen.

- Press the "Positions" softkey and select the desired position pattern (e.g. any positions) via softkey, enter the desired values in the parameter screen form press the "Accept" softkey.

The input screen is closed and the technology and positioning blocks are displayed in brackets.

Note

If you enter the positions first, only the drilling cycles for driven tools are offered after creating the positioning block.

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.

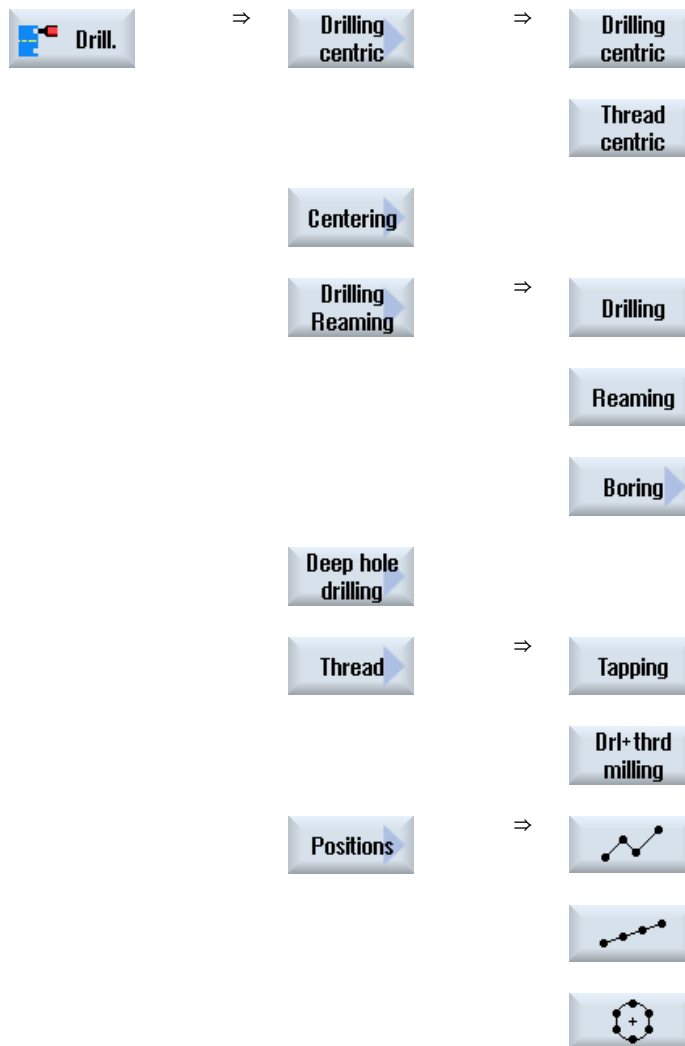
13.6.1 Drilling with Manual Machine

The same range of technological functions (cycles) is available as in automatic mode for drilling on the face or peripheral surface of a workpiece:



Machine manufacturer

Please refer to the machine manufacturer's specifications.

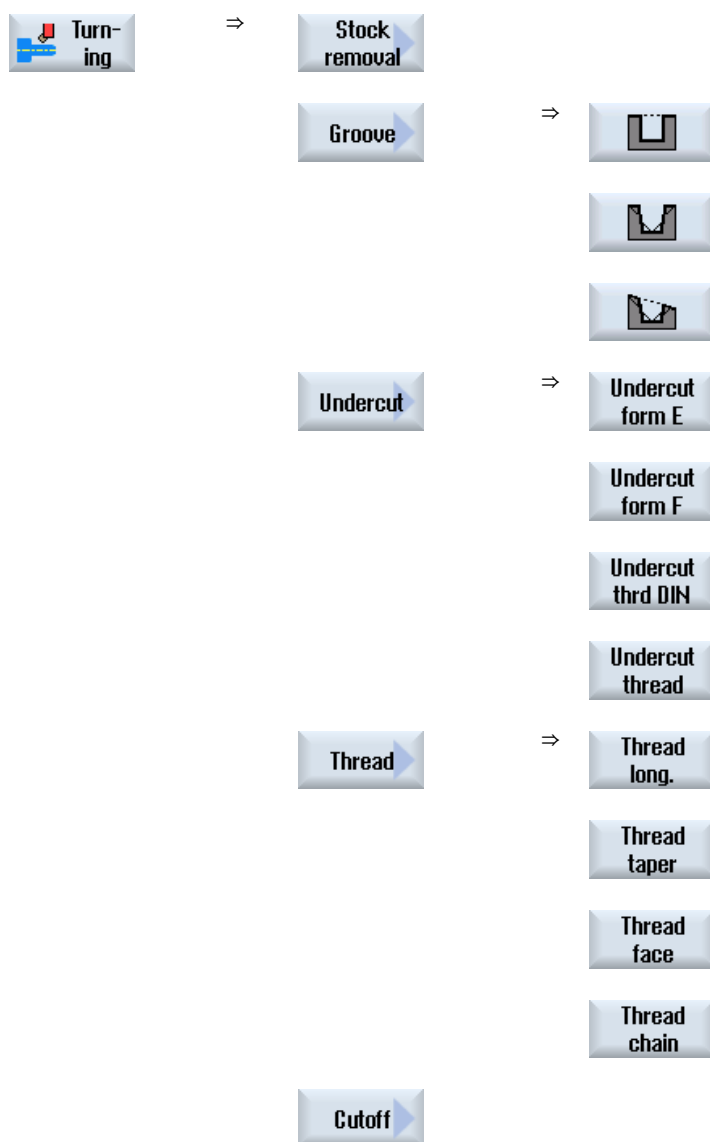


Parameters

The parameters of the input screen forms correspond to the parameters under Automatic (see Section "Drilling (Page 285)").

13.6.2 Turning with manual machine

The same range of technological functions (cycles) is available as in automatic mode for turning:



Parameters

The parameters of the input screen forms correspond to the parameters under Automatic (see Section "Auto-Hotspot").

Thread cutting

In addition to the functions that are made available by "thread-cutting" under Automatic, you can insert idle cuts during the machining process under "Manual Machine."

You can interrupt the infeed of the cutting depth during the machining process by inserting idle cuts, in order to smooth the flanks for example.



You can insert idle cuts using the "Idle cut" softkey.
This softkey is only active during machining.

Thread re-machining

You can rework existing threads, when repairing previously cut threads for example or as the result of changes that arise from re-measuring (see Section "Auto-Hotspot").

If you want to rework some threads, input the initial plunge depth E (inc.). This is the depth that was reached during a previous machining.

Note

By inputting the plunge depth, you avoid unnecessary idle cuts when reworking the threads.

See also

Rotate (Page 323)

Thread synchronizing (Page 119)

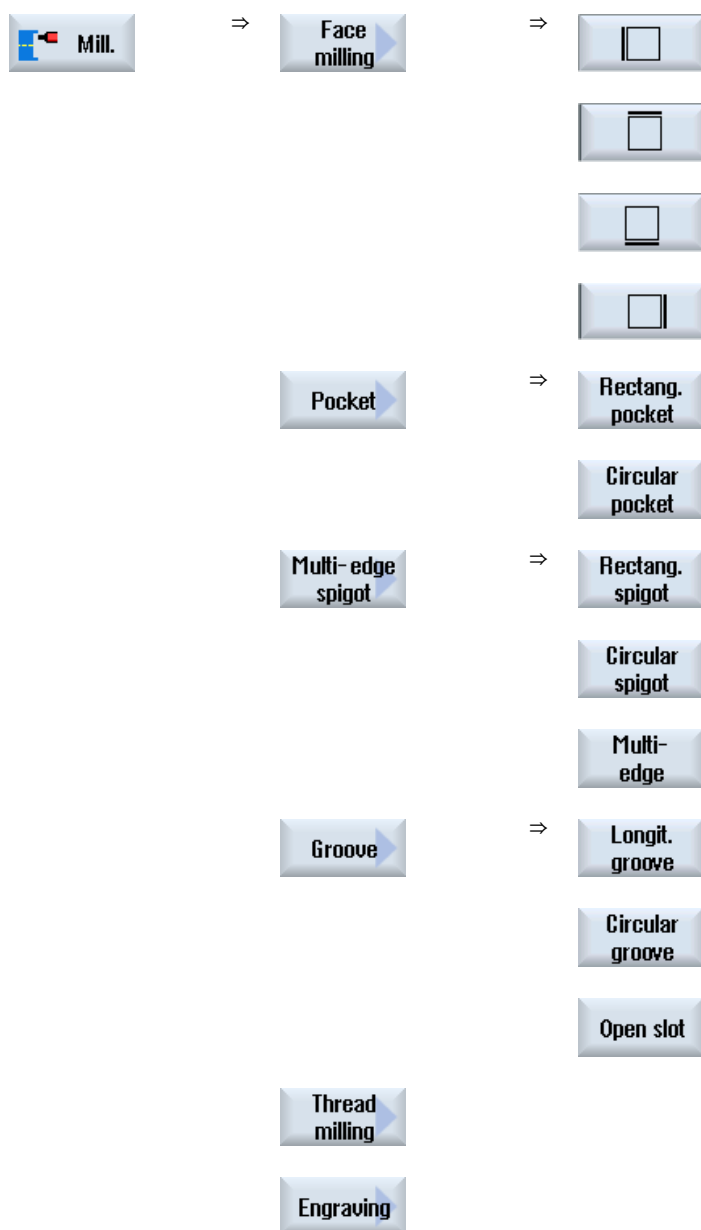
13.6.3 Milling with Manual Machine

The same range of technological functions (cycles) is available as in automatic mode for the milling of simple geometric shapes.



Machine manufacturer

Please refer to the machine manufacturer's specifications.



Parameters

The parameters of the input screen forms correspond to the parameters under Automatic (see Section "Milling (Page 386)").

13.7 Simulation and simultaneous recording

For more complex machining processes, 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 177)"). The execution of the work steps is displayed graphically on the screen.



Software option

You require the option "Simultaneous recording ShopTurn (real-time simulation)" for the simultaneous recording of the work steps.

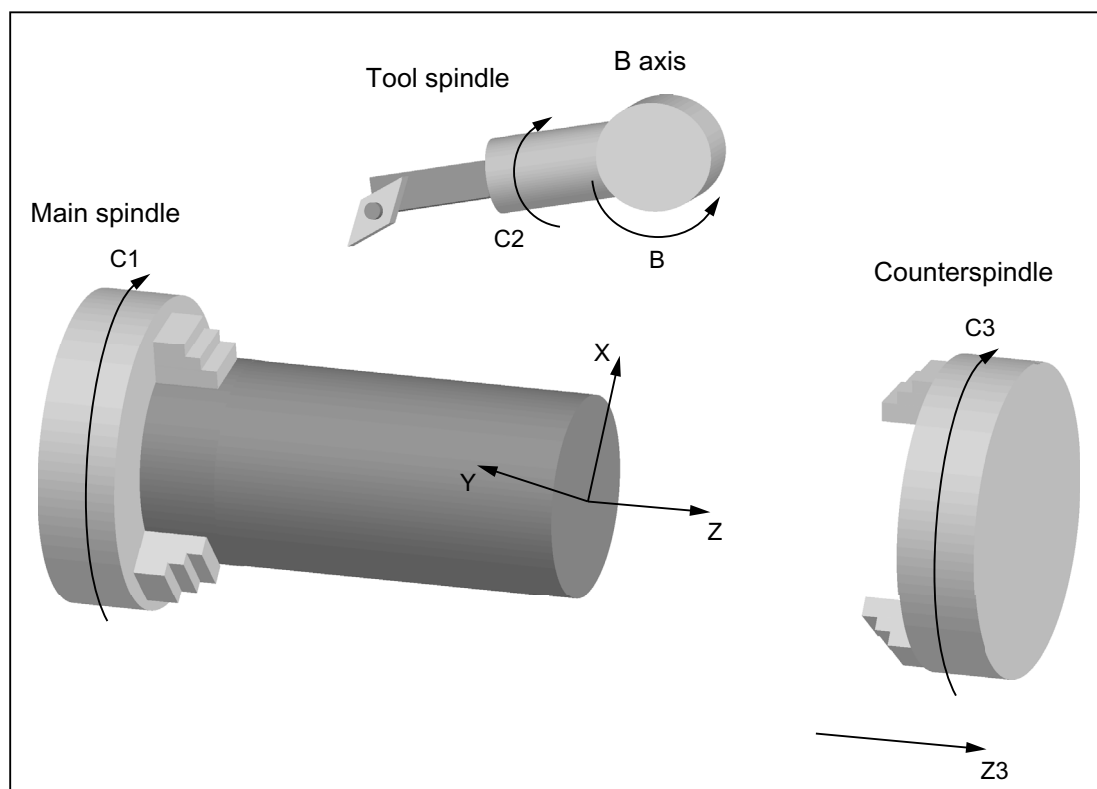
Note

In "Manual Machine", you can simulate a work step with an already opened and filled out parameter screen form.

Working with a B axis

14.1 Lathes with B axis

With an additional B axis, you have the option of aligning milling machines and lathes.



The initial setting in which all tools must be measured is $B=0$.

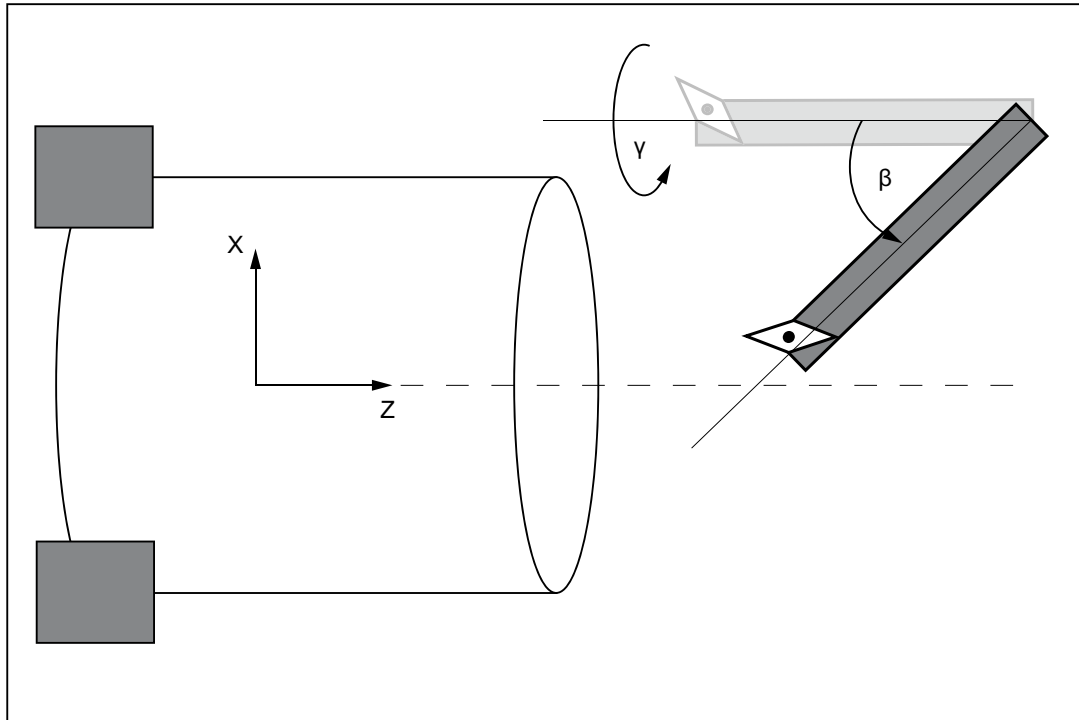
When turning, you can align the tool for special machining operations using the B axis and C axis of the tool spindle.

When milling, you can swivel the workpiece using the B and C axis of the main or counterspindle so that you can mill and drill on inclined surfaces.

The B axis is also used for aligning tools for face and peripheral surface machining.

Alignment angles β and γ

Alignment angles β and γ are required for turning with tool alignment.

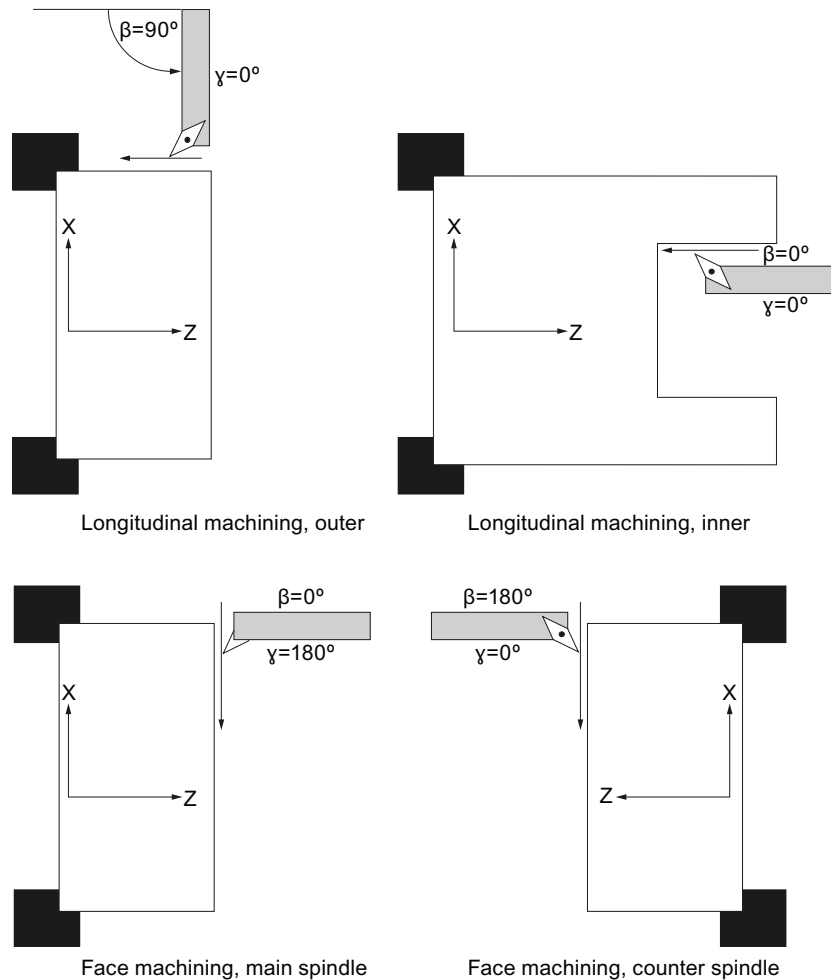


β : Rotation around the Y axis (with the B axis)

γ : Rotation around the Z axis (with the tool spindle)

Turning

Alignment angles allow you to perform a wide range of different turning operations (for example, internal and external longitudinal machining, surface machining with a main spindle and counterspindle, residual material) without changing the tool.



Display of the B axis

The B axis is displayed in the following windows:



- Axis position display in actual value window
- In the "Positioning" window to position axes in manual operation
- You can display the B axis in the zero offset lists using the "Zero offset" softkey and you can also define the offset.

14.2 Tool alignment for turning

The input fields for the β and γ angles for aligning the tool are available in the tool screen and in all turning screens.

β angle

In the entry field " β " you have the possibility of selecting the main alignment of the tool:

-  : $\beta = 0^\circ$
-  : $\beta = 90^\circ$
- Empty : The required angle can be freely entered

Programming when machining with a counterspindle

When machining on the counterspindle, it is programmed in exactly the same way as the main spindle.

The direction display of the arrows depends on the settings.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

γ angle

In the entry field " γ " you have the possibility of selecting the main alignment of the tool:

- 0°
- 180°
- or -
- Free entry field to enter the required angle



Machine manufacturer

Please refer to the machine manufacturer's specifications.

14.3 Milling with a B axis

No special entries are required for face machining and peripheral surface machining.

Face machining

Milling at the face (G17) is realized on the main spindle in the B axis position $B = 0^\circ$. If you are machining at the face (G17) of the counterspindle, then this corresponds to the opposite setting of the B axis position $B = 180^\circ$.

Peripheral surface machining

Milling on the peripheral surface is always realized in the B axis position, $B = 90^\circ$ (main and counterspindle).

Machining on an inclined surface

You can define inclined surfaces in a swivel mask.

You can enter the rotation of the planes around the geometry axes (X, Y, Z) of the tool coordinate system as described in the workpiece drawing. The rotation of the Work in the program is then automatically converted to a rotation for the relevant B and C axis of the machine during machining.

The swivel axes are always rotated to place the tool axis perpendicular to the machining axis for machining. During machining, the rotary axes are stationary.

The coordinate system is adapted to the surface to be machined irrespective of the required rotary axis positions.

14.4 Swiveling

General sequence

- Swivel the coordinate system into the plane to be machined via the swivel screen.
- Machining with the setting "Face B".
- If another machining type follows, swiveling is automatically deselected.

The swiveled coordinates are maintained in the reset state and after Power On. So you can still move out of an inclined hole, for example, with retraction in the +Z direction.



Machine manufacturer



Please refer to the machine manufacturer's specifications.

Swiveling is executed one axis at a time. In the case of axis-by-axis swiveling, the coordinate system is rotated around each axis in turn, with each rotation starting from the previous rotation. The axis sequence can be freely selected.

Procedure



1. The "Spigot Res. Mat." input window opens if you press the "Various" softkey.
2. Press the "Swivel plane" softkey.
3. Press the "Initial position" softkey if you wish to reestablish the basic setting, i.e. you wish to set the values back to 0.
This is done, for example, to swivel the coordinate system back to its original orientation.

Parameter	Description	Unit
T	Tool identifier	
RP	Retraction plane for face B	mm
C0	Positioning angle for machining surface	Degrees
X0	Reference point for rotation	mm
Y0	Reference point for rotation	mm
Z0	Reference point for rotation	mm
Swivel mode 	<ul style="list-style-type: none"> axis-by-axis: Swivel coordinate system axis-by-axis Solid angle: Swivel via solid angle Proj. angle: Swiveling via projection angle Direct: Position rotary axes directly 	
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	Axis angle	The sequence of the axes can be interchanged as required using the Select key.
Y	Axis angle	
Z	Axis angle	
X1	New zero point of rotated surface	mm
Y1	New zero point of rotated surface	mm
Z1	New zero point of rotated surface	mm

Note

Other additive transformations can be added to the offsets before (X0, Y0, Z0) or after (X1, Y1, Z1) swiveling (see Section "Work offsets").

14.5 Approach/retraction

If you want to optimize approach/return for swiveling with the B axis, you can create a special cycle that ignores the automatic approach/retraction strategy.

You can insert the approach/retraction cycle between any machining step program blocks, but not within linked program blocks.

Sequence

The starting point for the approach/retraction cycle is the safety clearance approached after the last machining operation.

If you want to perform a tool change, you can move the tool through a total of 3 positions (P1 to P3) to the tool change point and through a maximum of 3 additional positions (P4 to P6) to the next starting point.

The 1st, 3rd, and 6th positions move the linear axes, while the 2nd and 5th positions move the rotary axes.

If no tool change is needed you can generate no more than 6 motion blocks.

The numbers (1 – 6) represent the processing sequence.

Note

Programming additional positions

If 3 or 6 positions are not sufficient for the approach/retraction, you can call the cycle several times in succession to program further positions.

 CAUTION
--

Note that the tool will move from the last position programmed in the approach/retraction cycle directly to the starting point for the next machining operation.
--

Parameter	Description	Unit
F1	Feedrate to approach the first position Alternatively, rapid traverse	mm/min
X1	1. position (inc or Ø abs)	mm
Z1	1. position (inc or Ø abs)	mm
Y1	Retraction to safety clearance	mm
β 2	Beta angle for 1st swivel movement	Degrees
γ 2	Gamma angle for 1st swivel movement	Degrees
Tracking	The position of the tool tip is maintained during swiveling. Please refer to the machine manufacturer's instructions.	
F3	Feedrate to approach the third position Alternatively, rapid traverse	mm/min
X3	3. position (inc or Ø abs)	mm
Z3	3. position (inc or Ø abs)	mm
Tool change	<ul style="list-style-type: none"> • TlChngPt: Approach the tool change point from the last programmed position and carry out a tool change • Direct: Tool is not changed at the tool change position, but at the last programmed position • No: Tool is not changed 	
T	Tool name (not for tool change "no")	
D	Cutting edge number (not for tool change "no")	
F4	Feedrate for approach to the fourth position Alternatively, rapid traverse	mm/min
X4	4. position (inc or Ø abs)	mm
Z4	4. position (inc or Ø abs)	mm
β 5	Beta angle for 2nd swivel movement	Degrees
γ 5	Gamma angle for 2nd swivel movement	Degrees
Tracking	The position of the tool tip is maintained during swiveling. Please refer to the machine manufacturer's instructions.	
F6	Feedrate to approach the sixth position Alternatively, rapid traverse	mm/min
X6	6. position (inc or Ø abs)	mm
Z6	6. position (inc or Ø abs)	mm
Y6	Retraction to safety clearance	mm

14.6 Position pattern

In drilling and milling operations with face B, position patterns "full circle/pitch circle" provide the following options for machining on inclined surfaces

- with swivel plane
- with C axis

Procedure



1. Press the "Drilling" softkey.



2. Press the "Positions" and "Full circle/pitch circle" softkeys.



Parameter	Description	Unit
	Face B on the swivel plane	
Z0	Z coordinate of the reference point (abs)	mm
X0	X coordinate of the reference point (abs)	mm
Y0	Y coordinate of the reference point (abs)	mm
$\alpha 0$	Starting angle: Angle of 1st hole with reference to X axis. Positive angle: Full circle is rotated counterclockwise. Negative angle: Full circle is rotated in clockwise direction.	Degrees
$\alpha 1$	Indexing angle: After the first hole has been drilled, all additional positions are approached at this angle (only for pitch circle). Positive angle: Additional positions are rotated in counterclockwise direction. Negative angle: Additional positions are rotated in clockwise direction.	Degrees
R	Radius of full circle	mm
N	Number of positions on circle	
Positioning	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.	mm
	Face B - wit C axis	
Z0	Z coordinate of the reference point (abs)	mm
X0	X coordinate of the reference point (abs)	mm
Y0	Y coordinate of the reference point (abs)	mm
$\alpha 0$	Starting angle: Angle of 1st hole with reference to C axis. Positive angle: Full circle is rotated counterclockwise. Negative angle: Full circle is rotated in clockwise direction.	Degrees

Parameter	Description	Unit
$\alpha 1$	Indexing angle: After the first hole has been drilled, all additional positions are approached at this angle (only for pitch circle). Positive angle: Additional positions are rotated in counterclockwise direction. Negative angle: Additional positions are rotated in clockwise direction.	Degrees
N	Number of positions on circle	

14.7 Tool selection for the manual mode

For the preparatory actions in the manual mode, tool selection and spindle control are both performed centrally in the T, S, M window.

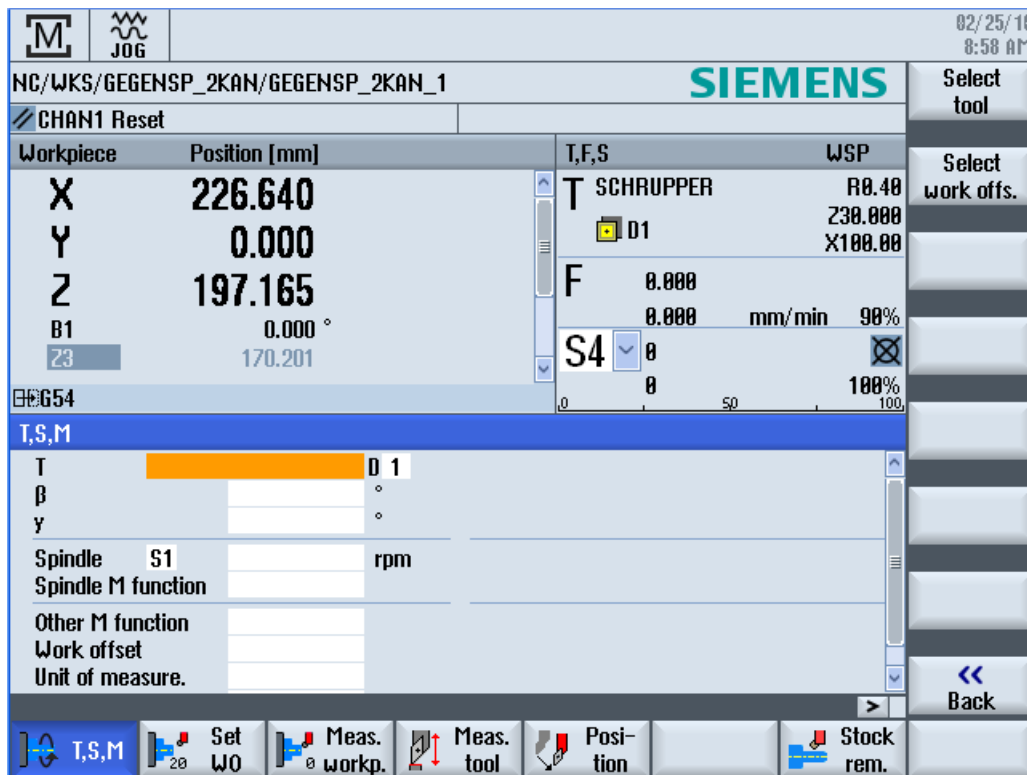


Figure 14-1 TSM window for the B and C axis

Procedure



Aligning the milling and turning tool with β angle:

Press the <SELECT> key and select

- 0° or
- 90° or
- Value entry box to freely enter the angle.



Aligning the turning tool with γ angle:

Press the <SELECT> key and select

- 0°
- 180°
- Value entry box to freely enter the angle.

See also

Selecting a tool and spindle (Page 105)

14.8 Measuring a tool with the B axis

Procedure



1. For the preparatory actions in the manual mode, tool selection and spindle control are both performed centrally in the T, S, M window.



2. Then, as usual, measure the tool using the "Measure tool" function.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

See also

T,S,M window (Page 105)

Measuring a tool manually (Page 75)

Working with two tool carriers

With SINUMERIK Operate, you can work at a lathe with two tool holders, both of which are mounted on an X axis. The tool carriers may be revolvers, multifix, or a combination of both.

The main machining is performed in the negative X axis direction. As both tool holders are mounted on the same axis it is only possible to machine with one tool.

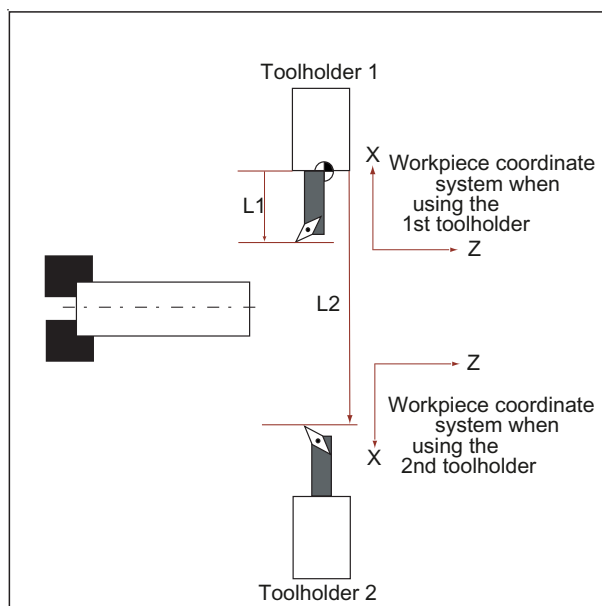
The workpiece is always located between the two tool holders. The tool lengths of all tools, i.e. of both tool holders, have the same reference point, which is usually on tool holder 1. That is why the tool lengths of the tools of the second tool holder are always longer than those of the tools on the first tool holder.

15.1 Programming with two tool carriers

You always program in the basic coordinate system (workpiece coordinate system of the first tool holder). You do not have to take into account in which tool holder the tool is inserted.

Tool on the second tool holder

If a tool on the second tool holder is selected, the X and Y axes are mirrored and the main spindle and counterspindle are offset (rotated) through 180°.



In the simulation, the tool is always displayed on the correct side, just the same as it is used at the machine.

The programmed C offset around 180° only affects C axes, not spindles.

It is not possible to machine a thread with tools that are distributed between both tool holders.

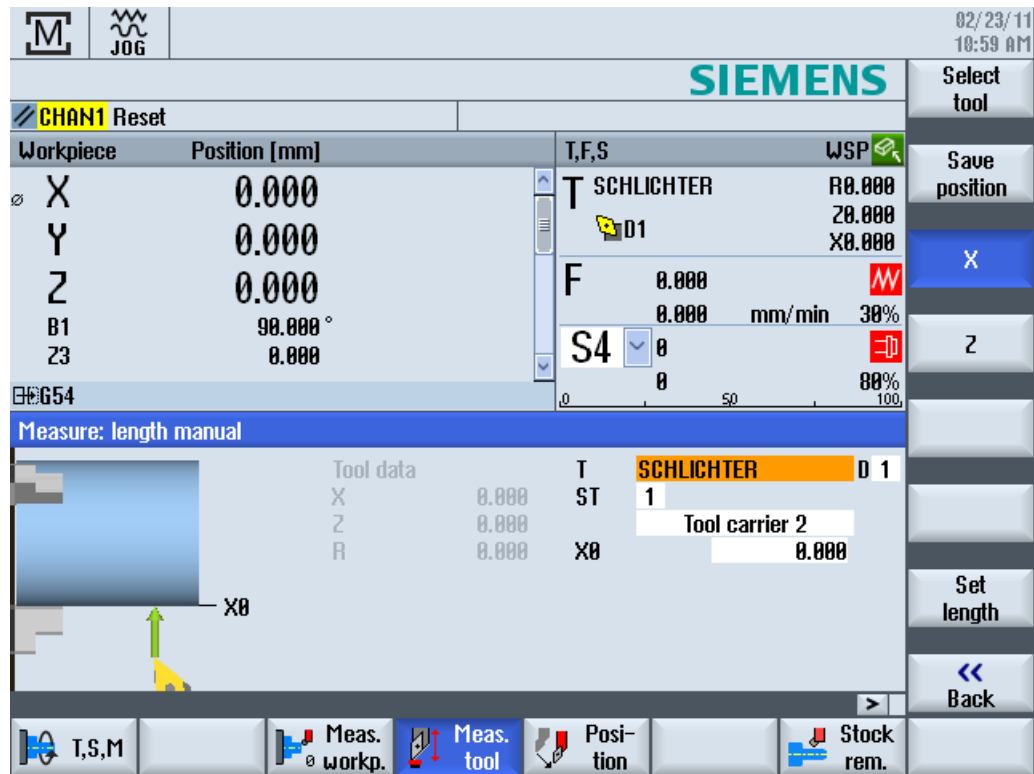
G code programming

The following points must be taken into account for G code programming:

- After a tool change, tool mirroring on the second tool holder is automatically activated.
- When a TRANSMIT command is programmed, tool mirroring on the second tool holder is automatically activated.

15.2 Measure tool

Selection options "Toolholder 1" and "Toolholder 2" are available for scratching when measuring a tool. This is how you set the tool holder in which the tool to be measured is located.



Tool management

16.1 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.

Grinding-specific tool data is provided here if you are working with grinding tools.

Sorting the lists

You can change the sorting within the lists according to:

- The magazine
- The name (tool identifier, alphabetic)
- The tool type
- The T number (tool identifier, numerical)
- The D number

Filtering the lists

You can filter the lists according to the following criteria:

- Only display the first cutting edge
- Only tools that are ready to use
- Only tools that have reached the pre-alarm limit
- Only locked tools
- Only tools with active code

Search functions

You have the option of searching through the lists according to the following objects:

- Tool
- Magazine location
- Empty location

16.2 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.

16.3 Tool types

A number of tool types are available when you create a new tool. The tool type determines which geometry data is required and how it will be computed.

Tool types

New tool - favorites		
Type	Identifier	Tool position
500	- Roughing tool	
510	- Finishing tool	
520	- Plunge cutter	
540	- Threading tool	
550	- Button tool	
560	- Rotary drill	
580	- 3D turning probe	
730	- Stop	
120	- End mill	
140	- Facing tool	
150	- Side mill	
200	- Twist drill	
240	- Tap	
	Multitool	

Figure 16-1 Example of Favorites list

New tool - milling cutter		
Typ	Identifier	Tool position
100	- Milling tool	
110	- Cylindr. ball end	
111	- Conical ball end	
120	- End mill	
121	- End mill corner round.	
130	- Angle head cutter	
131	- Corn.round.ang.hd.cut	
140	- Facing tool	
145	- Thread cutter	
150	- Side mill	
151	- Saw	
155	- Bevelled cutter	
156	- Beveled cutter corner	
157	- Tap, die-sink. cutter	
160	- Drill&thread cut.	

Figure 16-2 Available tools in the "New Tool - Milling Cutter" window

New tool - drill		
Typ	Identifier	Tool position
200	Twist drill	
205	Solid drill	
210	Boring bar	
220	Center drill	
230	Countersink	
231	Counterbore	
240	Tap	
241	Fine tap	
242	Tap, Whitworth	
250	Reamer	

Figure 16-3 Available tools in the "New Tool - Drill" window

New tool - favorites		
Type	Identifier	Tool position
400	Surf. grinding wheel	
410	Facing wheel	
490	Dresser	

Figure 16-4 Tools listed in the "New Tool - Grinding Tools" window

New tool - turning tools		
Type	Identifier	Tool position
500	Roughing tool	
510	Finishing tool	
520	Plunge cutter	
530	Cutting tool	
540	Threading tool	
550	Button tool	
560	Rotary drill	
580	3D turning probe	
585	Calibrating tool	

Figure 16-5 Available tools in the "New Tool - Turning Tools" window








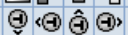
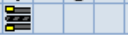

New tool - special tools		
Type	Identifier	Tool position
700	Slotting saw	
710	3D probe	
711	Edge finder	
712	Mono probe	
713	L probe	
714	Star probe	
725	Calibrating tool	
730	Stop	
900	Auxiliary tools	
	Multitool	

Figure 16-6 Available tools in the "New Tool - Special Tools" window

See also

Changing the cutting edge position or tool type (Page 678)

16.4 Tool dimensioning

This section provides an overview of the dimensioning of tools.

Tool types

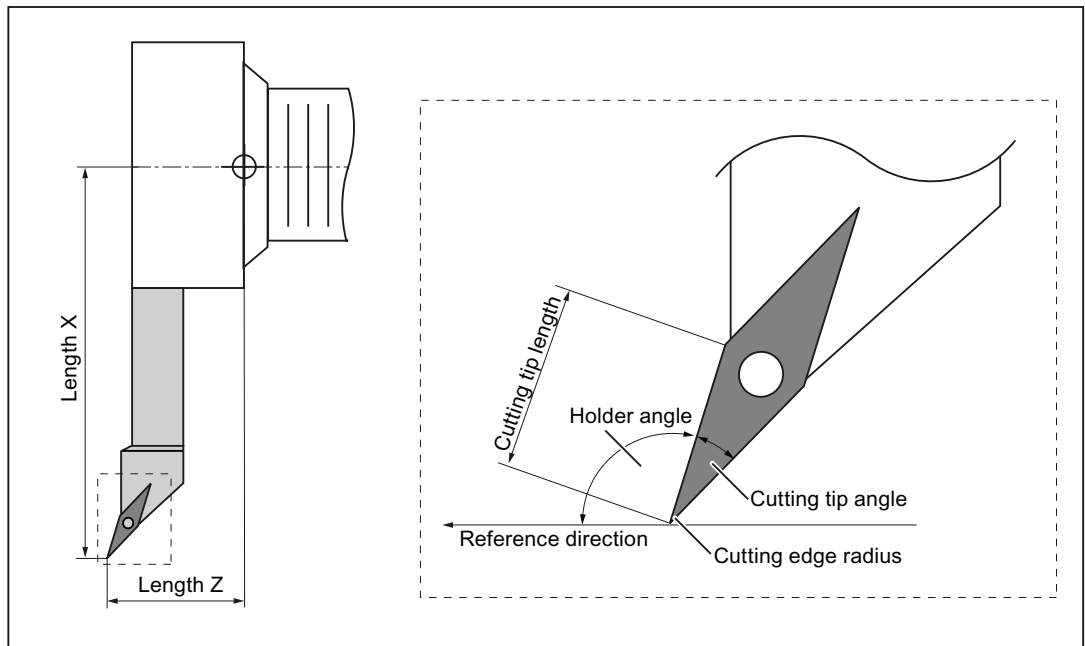


Figure 16-7 Finishing tool (Type 510)

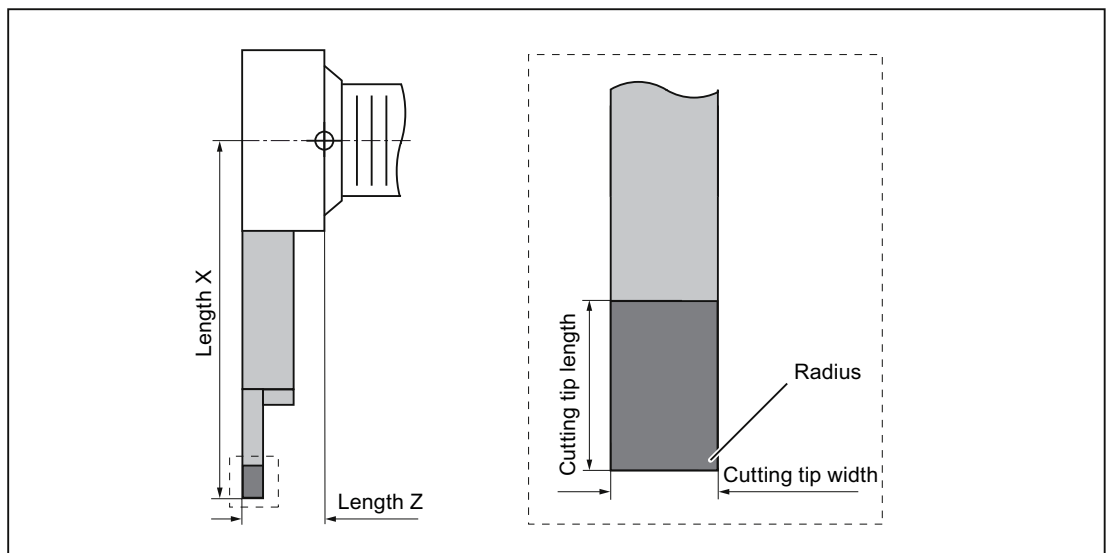


Figure 16-8 Plunge cutter (Type 520)

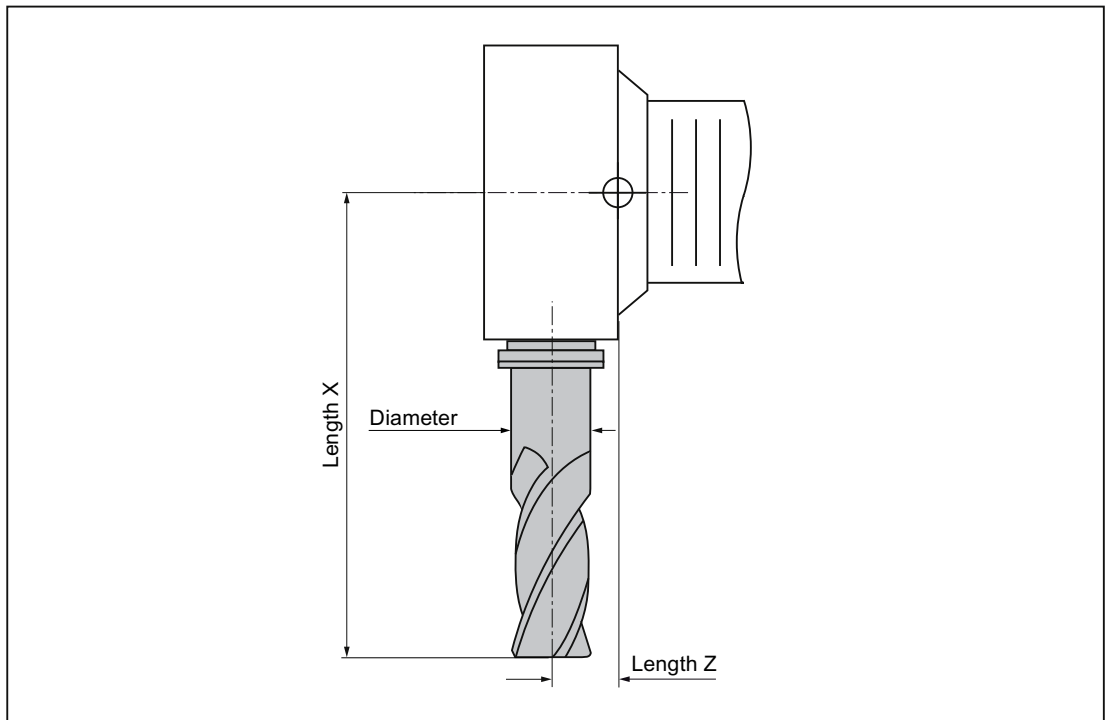


Figure 16-9 Milling cutter (Type 120)

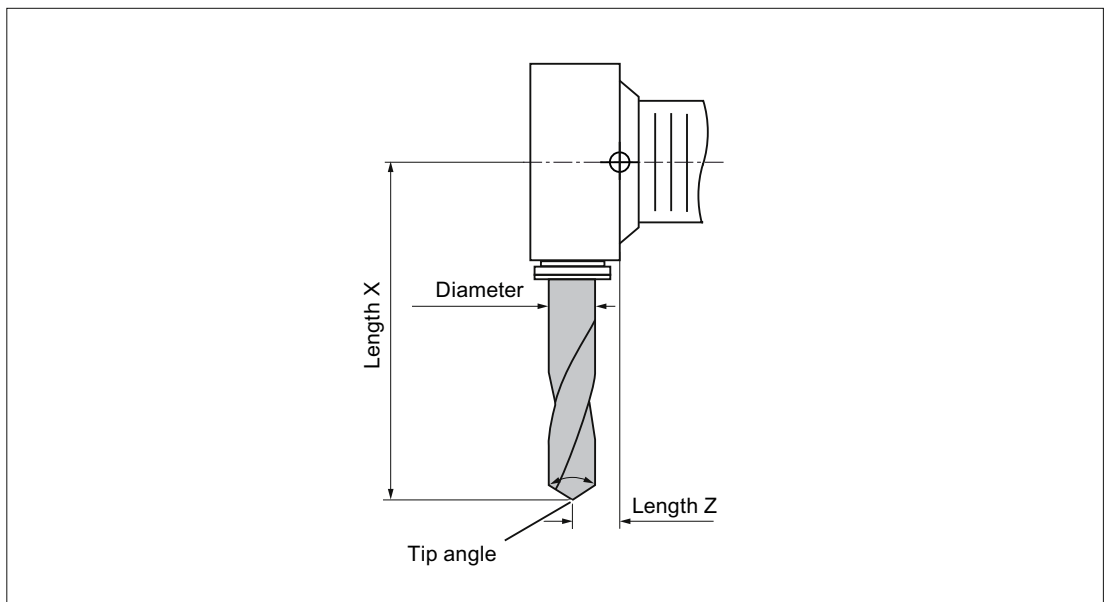


Figure 16-10 Drill (Type 200)

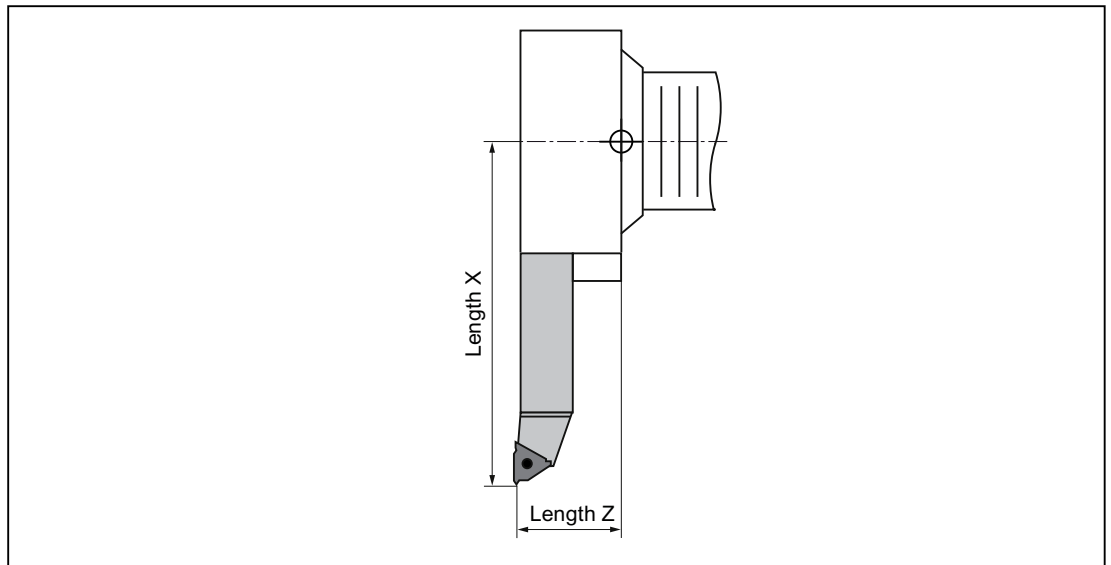


Figure 16-11 Threading tool (Type 540)

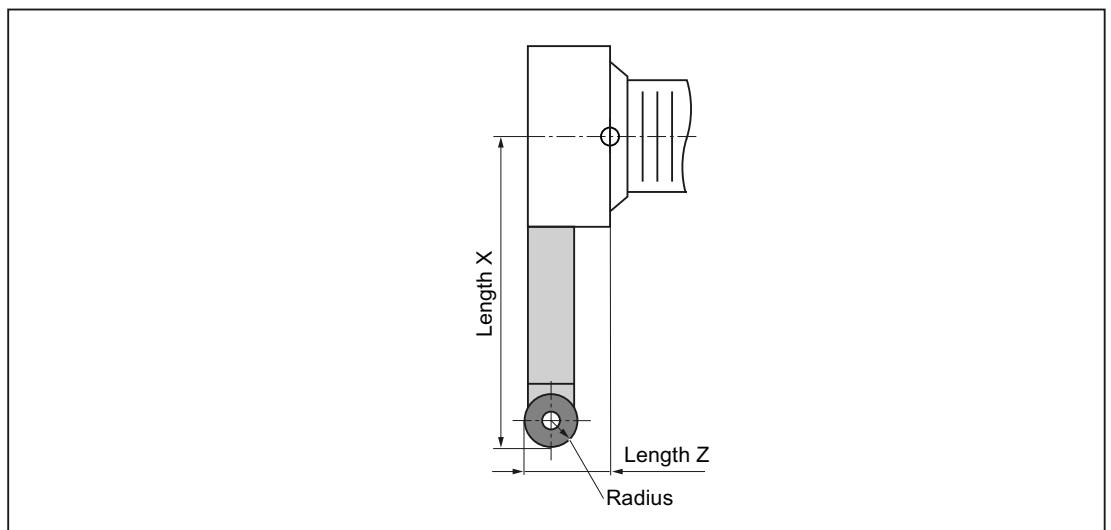


Figure 16-12 Button tool (Type 550)

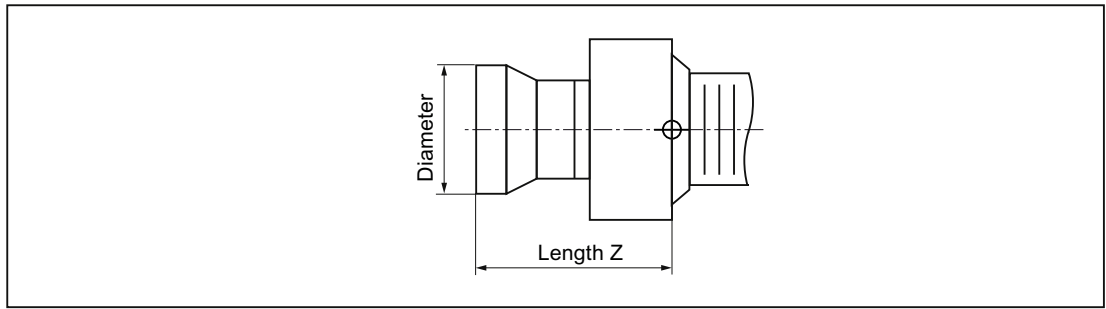


Figure 16-13 Stop (Type 730)

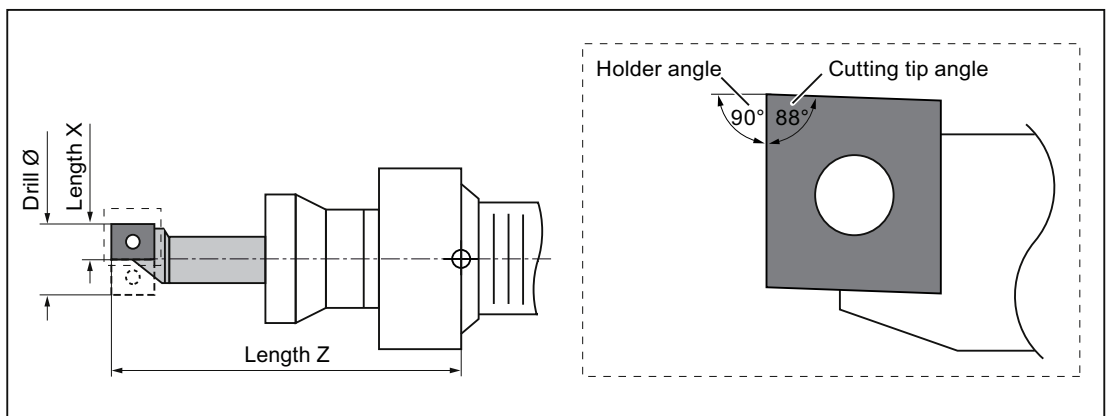


Figure 16-14 Rotary drill (Type 560)

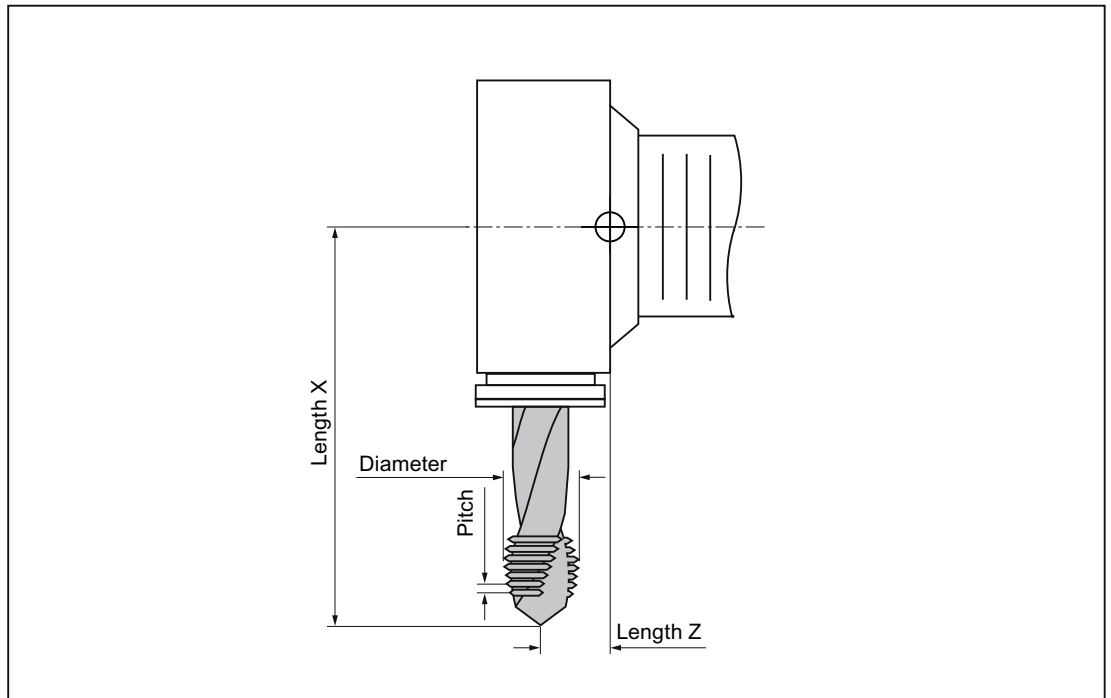


Figure 16-15 Tap (Type 240)

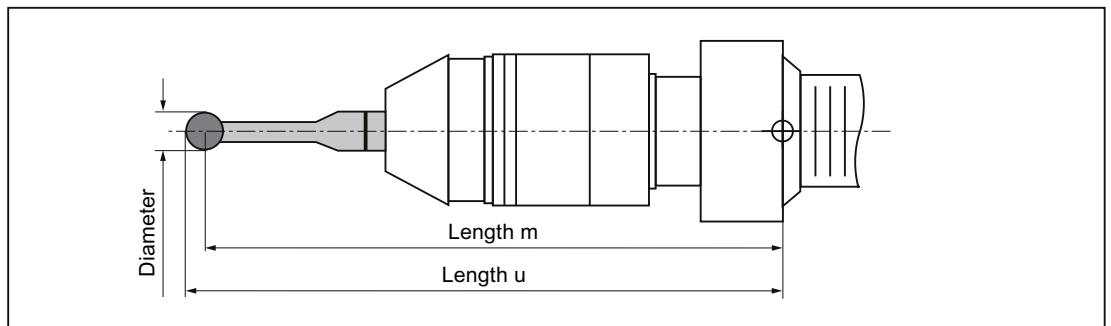


Figure 16-16 3D probe



Machine manufacturer

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

Please refer to the machine manufacturer's specifications.

Note

A 3D probe must be calibrated before use.




16.5 Tool list




All parameters and functions that are required to create and set up the tools are displayed in the tool list.

Each tool is uniquely identified by the tool identifier and the replacement tool number.

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


Tool parameters

Column heading	Meaning
Location BS  *  * * If activated in magazine selection	Magazine/location number <ul style="list-style-type: none"> 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): <ul style="list-style-type: none"> Spindle location as an icon Locations for gripper 1 and gripper 2 (applies only when a spindle with dual gripper is used) as icons.
Type	Tool type Specific tool offset data is displayed depending on the tool type (represented as an icon). The icon identifies the position of the tool; this was selected when the tool was created.
	You have the option of changing the tool position or the tool type using the <SELECT> key.
Tool name	The tool is identified by the name and the 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
Length X, length Z	Tool length Geometry data length X and length Z
Radius	Tool radius

Column heading	Meaning
Width/ Tip width/ Tip angle / Pitch Drilling radius	<p>Cutting edge for Type 150 - side milling cutter and Type 151 - saw</p> <p>Tip width for Type 520 - plunge cutter and Type 530 - cut-off tool</p> <p>Tip angle for Type 200 – twist drill and Type 220 – centering tool and Type 230 – countersink</p> <p>Pitch for Type 240 - tap</p> <p>Drilling radius for Type 560 - rotary drill. Holder angle and cutting tip angle are fixed.</p>
	<p>Cutting edge graphic</p> <p>The cutting edge graphic shows the positioning defined by the holder angle, cut direction and cutting tip angle.</p> <p>Holder angle for Type 500 - rougher and Type 510 - finisher.</p> <p>The reference direction for the holder angle specifies the cut direction.</p> <p>In addition to the holder angle, the cutting tip angle is also specified.</p>
N	<p>Number of teeth for 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.</p>
Tip length	<p>Tip length of a cutting tool or grooving cutter</p> <p>The tip length is required for displaying the tools during the simulation of the program processing.</p>
	<p>Direction of spindle rotation</p> <p>The direction of the spindle's rotation is relative to the tool spindle for powered tools (drilling and milling machines) and to the main or counter-spindle for turning tools.</p> <p>If you are using a drilling or milling machine for "Drilling centric" or "Thread centric", the specified direction of rotation is relative to the cutting direction of the tool. The main spindle then rotates to match the tool.</p> <p><input type="checkbox"/> Spindle is not switched on</p> <p><input type="checkbox"/> CW spindle rotation</p> <p><input type="checkbox"/> CCW spindle rotation</p>
	<p>Coolant 1 and 2 (e.g. internal and external cooling) can be switched on and off.</p> <p>The coolant infeed at the machine does not necessarily have to be set-up.</p>
M1 - M4	<p>Other tool-specific functions such as additional coolant infeed, monitoring functions for speed, tool breakage, etc.</p>

Further parameters

If you have set up unique cutting edge numbers, these are displayed in the first column.

Column heading	Meaning
D no.	Unique cutting edge number
SN	Cutting edge number
SC 	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 "ShopMill/ShopTurn" option.



Machine manufacturer








Please refer to the machine manufacturer's specifications.

References

Information on the configuration and setting up of the tool list can be found in the following references:

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

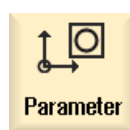
Icons in the tool list

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 tool tip will provide a brief 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		The magazine location is positioned at the loading position.
Red "X"		The magazine location is disabled.

**Machine manufacturer**

Please refer to the machine manufacturer's specifications.

Procedure



Parameter

1. Select the "Parameter" operating area.



Tool list

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

See also

Displaying tool details (Page 676)

Changing the cutting edge position or tool type (Page 678)

16.5.1 Additional data

The following tool types require geometry data that is not included in the tool list display.

Tool types 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

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 refer to the machine manufacturer's specifications.

Procedure



1. The tool list is opened.
2. In the list, select an appropriate tool, e.g. an angle head cutter.
3. Press the "Additional data" softkey.
The "Additional Data - ..." window opens.
The "Additional data" softkey is only active if a tool for which the "Additional Data" window is configured is selected.

16.5.2 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, turning or special tool via the corresponding softkeys.

Note

Grinding tools

Additional grinding tools are available depending on the particular machine configuration.

Procedure



1. The tool list is opened.

2. Place the cursor in the tool list at the position where the new tool should be 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", "Grinders 400-499" "Turntools 500-599" or "Spec.tool 700-900".

...

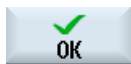


The "New tool - milling cutter", "New tool - drill", "New tool - grinding tools", "New tool - turning tools" or "New tool - special tools" window opens.

4. Select the tool by placing the cursor on the corresponding tool type and on the icon of the desired cutting edge position.



5. If more than four cutting edge positions are available, select the desired cutting edge position with the aid of the <cursor left> and <cursor right> keys.



6. Press the "OK" softkey.
The tool is added to the tool list with a predefined name. If the cursor is located on an empty magazine location in the tool list, then the tool is loaded to this magazine location.

The tool creation sequence can be defined differently.

Multiple load 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

References:

For a description of configuration options, refer to the
Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

16.5.3 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 replacement tool number ST.



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



5. Traverse the tool towards the workpiece in the direction that is to be measured and scratch it.

6. Enter the position of the workpiece edge in X0 or Z0.

If no value is entered for X0 or Z0, the value is taken from the actual value display.



7. Press the "Set length" softkey.

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

16.5.4 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 control 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 "Cutting edges" softkey in the "Tool list".

4. Press the "New cutting edge" softkey.

A new data set is stored in the list.

The cutting edge number is incremented by 1 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 second cutting edge.

6. Repeat this process if you wish to create more tool edge offset data.



7. Position the cursor on the cutting edge that you want to delete and press the "Delete cutting edge" softkey.

The data set is deleted from the list. The first tool cutting edge cannot be deleted.

16.5.5 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.

16.5.6 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 tool list.

When you are loading a tool, the application automatically suggests an empty location. You may also directly specify an empty magazine location.

You can unload tools from the magazine that you are not using at present. HMI then automatically saves the tool data in the tool list in the NC memory outside the magazine.

Should you want to use the tool again later, simply load the tool with the tool data into the corresponding magazine location again. Then the same tool data does not have to be entered more than once.

Procedure

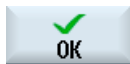


1. The tool list is opened.
2. Place the cursor on the tool that you want to load into the magazine (if the tools are sorted according to magazine location number you will find it at the end of the tool list).
3. Press the "Load" softkey.



The "Load to..." window opens.

The "... location" field is pre-assigned with the number of the first empty magazine location.



4. Press the "OK" softkey to load the tool into the suggested location.

- OR -

Enter the location number you require and press the "OK" softkey.



- OR -

Press the "Spindle" softkey.



- . The tool is loaded into the specified magazine location or spindle.

Several magazines

If you have configured several magazines, the "Load to ..." window appears after pressing the "Load" softkey.

If you do not want to use the suggested empty location, then enter your desired magazine and magazine location. Confirm your selection with "OK".

Multiple load points

If you have configured several loading points for a magazine, then the "Load Point Selection" window appears after pressing the "Load" softkey.

Select the required loading point and confirm with "OK".

Unloading tools



1. Place the cursor on the tool that you would like to unload from the magazine and press the "Unload" softkey.



2. Select the required load point in the "Load Point Selection" window.

3. Confirm your selection with "OK".



- OR -

Undo your selection with "Cancel".

16.5.7 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.

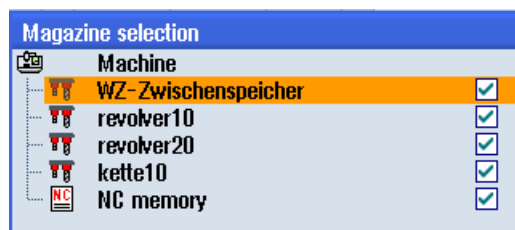
- OR -



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 refer to the machine manufacturer's specifications.

References

For a description of configuration options, refer to the
Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

16.6 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 monitor a tool by a type or any combination of monitoring types can be switched on.

In addition, you can disable tools when you no longer wish to use them.


Tool parameters

Column heading	Meaning
Location BS   * If activated in magazine selection	Magazine/location number <ul style="list-style-type: none"> 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): <ul style="list-style-type: none"> 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. The icon identifies the position of the tool; this was selected when the tool was created.
	You have the option of changing the tool position or the tool type using the <Select> key.








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 X, Δ length Z	Wear for length X, or wear for length Z
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 set up unique cutting edge numbers, these are displayed in the first column.

Column heading	Meaning
D no.	Unique cutting edge number
SN	Cutting edge number
SC	Setup offsets
	Display of the existing setup offsets

Icons in the tool wear 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 wear" softkey.

See also

Displaying tool details (Page 676)

Changing the cutting edge position or tool type (Page 678)

16.6.1 Reactivate tool

You can replace disabled tools or make them ready for use again.

Preconditions

In order to be able to reactivate a tool, the monitoring function must be activated and a setpoint must be stored.

Procedure



1. The tool wear list is opened.



2. Position the cursor on the disabled tool which you would like to reuse.

3. Press the "Reactivate" softkey.

The value entered as the setpoint is entered as the new tool life or workpiece count.

The disabling of the tool is cancelled.

Reactivating and positioning

When the "Reactivate with positioning" function is configured, the selected tool's magazine location will also be positioned at a loading point. You can exchange the tool.

Reactivation of all monitoring types

When the "Reactivation of all monitoring types" function is configured, all the monitoring types set in the NC for a tool are reset during reactivation.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

References

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

Multiple load points

If you have configured several loading points for a magazine, then the "Load Point Selection" window appears after pressing the "Load" softkey.



Select the required load point and confirm with the "OK" softkey.

16.7 Tool data OEM

You have the option of configuring the list according to your requirements.

Depending on the machine configuration, grinding-specific parameters are displayed in the list with OEM tool data.

Grinding tool-specific parameters

Column heading	Meaning
Min. radius	Limit value for the radius of the grinding wheel to monitor the geometry.
Actual radius	Displays the sum of the geometry value, wear value and, if set, the base dimension.
Min. width	Limit value for the width of the grinding wheel to monitor the geometry.
Actual width	The width of the grinding wheel measured, for example, after the dressing.
Max. speed	Maximum speed
Max. peripheral velocity	Maximum peripheral velocity
Wheel angle	Angle of inclined wheel
Monitoring	Monitoring geometry and speed
Tool base dimension	Defines whether the base dimension is used when calculating the SUG and when monitoring the minimum wheel radius.
Spindle number 	Number of programmed spindle (e.g. grinding wheel peripheral velocity) and spindle to be monitored (e.g. wheel radius and width).
Param. rad.calc. 	Selects the parameter to calculate the radius <ul style="list-style-type: none"> • Length X • Length Y • Length Z • Radius
Chaining rule	This parameter defines which tool parameters of tool cutting edge 2 (D2) and tool cutting edge 1 (D1) have to be chained to one another. When the value of a chained parameter is modified, then the other cutting edge is automatically used when chaining the parameter.

References

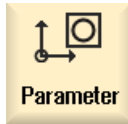
You can find more information on grinding tools in the following manual:

Function Manual, Extended Functions W4: Grinding-specific tool offset and monitoring functions / SINUMERIK 840D sl

Refer to the following document for more information on configuring OEM tool data:

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

Procedure



1. Select the "Parameter" operating area.






2. Press the "OEM tool" softkey.
3. Position the cursor on a grinding tool.

16.8 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 <ul style="list-style-type: none"> 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): <ul style="list-style-type: none"> 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. The icon identifies the position of the tool; this was selected when the tool was created.
	You have the option of changing the tool position or the tool type using the <Select> key.
Tool name	The tool is identified by the name and number of the replacement tool (ST). You can enter the name as text or number. Note: The maximum length of tool names is 31 ASCII characters. The number of characters is reduced for Asian characters or Unicode characters. The following special characters are not permitted: # ".
ST	Number of the replacement tool.
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 a tool as oversized. The tool occupies two half locations left, two half locations right, one half location top and one half location bottom in a magazine.
P	Fixed location coding. The tool is permanently assigned to this magazine location.

Further parameters

If you have set up unique cutting edge numbers, these are displayed in the first column.

Column heading	Meaning
D no.	Unique cutting edge number
SN	Cutting edge number

Magazine list icons

Icon/ Marking		Meaning
Tool type		
Red "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 676)

Changing the cutting edge position or tool type (Page 678)

16.8.1 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 "Load 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.

16.8.2 Relocating a tool

Tools can be directly relocated within magazines to another magazine location, which means that you do not have to unload tools from the magazine in order to load them into a different location.

When you are relocating a tool, the application automatically suggests an empty location. You may also directly specify an empty magazine location.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure



1. The magazine list is opened.

2. Position the cursor on the tool that you wish to relocate to a different magazine location.



3. Press the "Relocate" softkey.

The "... relocate from location ... to location ..." window is displayed. The "Location" field is pre-assigned with the number of the first empty magazine location.

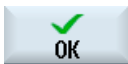


4. Press the "OK" softkey to relocate the tool to the recommended magazine location.

- OR -

Enter the required magazine number in the "...magazine" field and the required magazine location number in "Location" field.

Press the "OK" softkey.



The tool is relocated to the specified magazine location.

Several magazines

If you have set up several magazines, then the "...relocate from magazine... location... to..." window appears after pressing the "Relocate" softkey.

Select the desired magazine and location, and confirm your selection with "OK" to load the tool.

16.8.3 Unload all tools

You have the option of unloading all tools from the magazine list at the same time.

Requirements

The following requirements must be satisfied so that the "Unload 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 opened.



2. Press the "Unload all" softkey.

A prompt is displayed as to whether you really want to unload all tools.



3. Press the "OK" softkey to continue with the unloading of the tools.

The tools are unloaded from the magazine in ascending order according to the magazine location number.



4. Press the "Cancel" softkey if you wish to cancel the unloading operation.

16.9 Sorting tool management lists

When you are working with many tools, with large magazines or several magazines, it is useful to display the tools sorted according to different criteria. Then you will be able to find a specific tool more easily in the lists.

Procedure



1. Select the "Parameter" operating area.



2. Press the "Tool list", "Tool wear" or "Magazine" softkey.

...



3. Press the ">>" and "Sort" softkeys.



The lists are displayed sorted numerically according to magazine location.



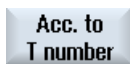
4. Press the "Acc. to type" softkey to display the tools arranged by tool type. Identical types are sorted according to their radius.



Press the "Acc. to name" softkey to display the tool names in alphabetical order.

The replacement tool numbers are used to sort tools with the same names.

- OR -



Press the "Acc. to T number" softkey to display the tools sorted numerically.

The list is sorted according to the specified criteria.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

16.10 Filtering the tool management lists

The filter function allows you to filter-out tools with specific properties in the tool management lists.

For instance, you have the option of displaying tools during machining that have already reached the prewarning limit in order to prepare the corresponding tools for equipping.

Filter criteria

- Only display the first cutting edge
- Only tools that are ready to use
- Only tools that have reached the prewarning limit
- Only locked tools

Note

Multiple selection

You have the option of selecting several criteria. You will receive an appropriate message if conflicting filter options are selected.

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.

16.11 Specific search in the tool management lists

There is a search function in all tool management lists, where you can search for the following objects:

- **Tools**

- You enter a tool name. You can narrow down your search by entering a replacement tool number.

You have the option of only entering a part of the name as search term.

- You enter the D number and activate if necessary, the check box "Active D number".

- **Magazine locations or magazines**

If only one magazine is configured, then the search is made according to the magazine location.

If several magazines are configured, then it is possible to search a specific magazine location in a specific magazine or just to search in a specific magazine.

- **Empty locations**

The empty location search is realized using the tool size. The tool size is defined by the number of half locations required to the right, left, top and bottom. All four directions are of significance for a box magazine. For a chain magazine, a disk-type or a turret, only the half locations to the right and left are of significance. The maximum number of half locations, which a tool can occupy is limited to 7.

If the lists with the location type are used, then the empty location search is made using the location type and location size.

You can enter the location type as numerical value or as text depending on the particular configuration.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

References

For a description of configuration options, refer to the

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

Procedure



1. Select the "Parameter" operating area.



2. Press the "Tool list", "Tool wear" or "Magazine" softkey.

...



3. Press the ">>" and "Search" softkeys.



4. Press the "Tool" softkey if you wish to search for a specific tool.

- OR -



Press the "Magazine location" softkey if you wish to search for a specific magazine location or a specific magazine.

- OR -



Press the "Empty location" softkey if you wish to search for a specific empty location.



5. Press the "OK" softkey.

The search is started.



6. Press the "Search" softkey again if the tool that was found is not the tool that was being searched for.

The search term is kept and with "OK" you start the search for the next tool that corresponds to the entry.



7. Press the "Cancel" softkey to cancel the search.

16.12 Displaying tool details

All of the parameters of the selected tool are listed in the "Tool Details - All Parameters" window.

The parameters are displayed, sorted according to the following criteria

- Tool data
- Grinding data (if grinding tools have been configured)
- Cutting edge data
- Monitoring data

Protection level

You require access level, keyswitch 3 (protection level 4) in order to edit the parameters in the detail window.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure



1. The tool list, the wear list, the OEM tool list or the magazine is opened.

...



2. Position the cursor to the desired tool.
3. If you are in the tool list or in the magazine, press the ">>" softkey and "Details".



- OR -

If you are in the wear list or OEM tool list, press the "Details" softkey.



The "Tool Details" window is displayed.

All of the available tool data is displayed in the list.



4. Press the "Grinding data" softkey if you want to display the grinding data.



5. Press the softkey "Cutting edge data", if you wish to display the cutting edge data.



6. Press the "Monitoring data" softkey if you want to display the monitoring data.



7. Press the "Additional details" softkey.
The "Tool Details - All Parameters" window opens.
All of the tool parameters are displayed in the list.

16.13 Changing the cutting edge position or 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. Press the <Cursor right> or <Cursor left> key to select the new cutting edge position or the tool position for grinding tools.



- OR -

Select the desired tool type in the list of favorites or select the required tool type using the softkeys "Cutters 100-199", "Drill 200-299", "Grinders 400-499" "Turntools 500-599" or "Spec.tool 700-900".

Note: A grinding tool can only be changed into another grinding tool type.



5. Press the "OK" softkey.
The new cutting edge position / tool position or the new tool type is accepted and the corresponding icon is displayed in the "Type" column.

16.14 Settings for tool lists

In the "Settings" window you have the following options to set the view in the tool lists:

- Display only one magazine in "Magazine sort"
 - You can limit the display to one magazine. The magazine is displayed with the assigned buffer magazine locations and the tools not loaded.
 - You may set via a configuration if you want to jump to the next magazine by clicking softkey "Magazine selection" or if the dialog "Magazine selection" is switched over to any magazine.

- Display only spindle in buffer.

In order to display only the spindle location during operation, the remaining locations of the buffer are hidden.

- Switch on adapter-transformed view
 - Geometry lengths and the application offsets are displayed in a transformed manner in the tool list.
 - In the tool wear list, the wear length and the sum offsets are displayed in a transformed manner.



Machine manufacturer

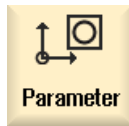
Please refer to the machine manufacturer's specifications.

References

For further information about configuring the settings, please refer to the following references:

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

Procedure



1. Select the "Parameter" operating area.



2. Press the "Tool list", "Tool wear" or "Magazine" softkey.

...



3. Press the "Continue" and "Settings" softkeys.



4. Activate the corresponding check box for the desired setting.

16.15 Working with 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.

Typical applications

One application for using multitools on lathes with revolver and counterspindle is to equip the multitool with two turning tools. One turning tool for machining on the main spindle and one turning tool for machining on the counterspindle.

An additional application is to use multitools on lathes equipped with a tool spindle. A multitool, on which several tools have been mounted, is inserted in the tool spindle at this machine. A tool change within the multitool is realized by positioning the multitool, i.e. by rotating the tool spindle.

Geometrical arrangement of the tools on the multitool

The multitool knows the parameters "No. of locations" and "Type of clearance determination of the locations".

The type of clearance 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.

16.15.1 Tool list for multitool

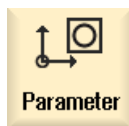
If you work with a multitool, the tool list is supplemented by the column for the multitool location number. As soon as the cursor is at a multitool in the tool list, certain column headings change.

Column heading	Meaning
Location	Magazine/location number
MT loc.	Multitool location number
TYPE	Symbol for multitool
Multitool name	Name of the multitool.
Loc. qty.	Number of locations of the multitool.
Type distance	<ul style="list-style-type: none"> Angle Clearance of the multitool locations in degrees Location number Multitool location number

TOA 1 Tool list										REVOLVER
Loc.	MT LO.	Type	Multitool name			No. loc.	Type Dist.			
1			FRAESER_MANTEL_R1	2	1	80.000	0.000	3.000	3	
2			MULTITOOL			2	Loc. no.			
	1		GEWINDESTAHL	2	1	0.000	0.000	0.000		
	2		GEWINDESTAHL	3	1	0.000	0.000	0.000		

Figure 16-17 Tool list with multitool in the revolver

Procedure



1. Select the "Parameter" operating area.



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

16.15.2 Create multitool

The multitool can be selected in the list of favorites as well as in the list of special tool types.

New tool - favorites		
Typ	Bezeichner	Werkzeuglage
500	- Roughing tool	
510	- Finishing tool	
520	- Plunge cutter	
540	- Threading tool	
550	- Button tool	
560	- Rotary drill	
580	- 3D turning probe	
730	- Stop	
120	- End mill	
140	- Facing tool	
150	- Side mill	
200	- Twist drill	
240	- Tap	
	Multitool	

Figure 16-18 List of favorites with multitool

New tool - special tools		
Typ	Bezeichner	Werkzeuglage
700	- Slotting saw	
710	- 3D probe	
711	- Edge finder	
730	- Stop	
900	- Auxiliary tools	
	Multitool	

Figure 16-19 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 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 -

Press the "Special tool 700-900" softkey.



4. Select the multitool and press the "OK" softkey.
The "New Tool" window appears.



5. Enter the name of the multitool, define the number of multitool locations and select how the clearance type is determined (e.g. multitool location number).

If you selected an angle to determine the clearance, then enter the clearance to the reference location as angular value for each multitool location.

New tool				
Multitoolname	Platz Anz.	Art Abstand	Abstand	
			Winkel	
MULTITOO3	3	Winkel	1	0.000
			2	120
			3	230

The multitool is created in the tool list.

The tool creation sequence can be defined differently.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

16.15.3 Equipping multitool with tools

Requirement

A multitool has been created in the tool list.

Procedure



1. The tool list is opened.



2. Select the required multitool, position the cursor at an empty multitool location and press the "New tool" softkey. Using the appropriate selection list, e.g. favorites, select the required tool



- OR -



Position the cursor on the tool that you wish to load into the multitool and press the "Load" and "Multitool" softkeys.



The "Load to..." window opens.

TOA 1 Tool list										REVOLVER	
Loc.	MT L.O.	Type	Tool name	ST	D	Length X	Length Z	∅	N		
1			FRAESER_MANTEL_R1	2	1	80.000	0.000	3.000	3		
2			MULTITOOL				2	Loc. no.			
	1		GEWINDESTAHL	2	1	0.000	0.000	0.000			
3											0.0
4											5.0 35
5											3.0 55
6											100
											100
7			SCHLICHTER_STIRN	1	1	85.000	165.000		1		135.0 35
8			ZENTRIERER	1	1	0.000	307.148	10.000	2		90.0

Load on ...

Es wurde kein passender Leerplatz gefunden

... Multitool Platz

2



Select the required multitool and the multitool location to which you wish to load the tool.

16.15.4 Removing a tool from multitool

If the multitool was mechanically re-assigned (i.e. new tool were mounted), then 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 is located, which is to be removed. 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 would like to remove and delete from the multitool and press the "delete tool" softkey.

16.15.5 Delete 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.

16.15.6 Loading and unloading multitool

Procedure



1. The tool list is opened.

2. Position the cursor at the multitool that you wish to load into the magazine.



3. The "Load to" window opens.

The "... location" field is pre-assigned with the number of the first empty magazine location.



4. Press the "OK" softkey to load the 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.

Unloading a multitool



1. Position the cursor on the multitool that you would like to unload from the magazine and 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.

16.15.7 Reactivating the multitool

Multitool and tools located on the multitool can be disabled independently of one another.

If a multitool is disabled, then the tools of the multitool can no longer be changed in using a tool change.

If only one tool on a multitool has a set monitoring function and the lifetime or the unit quantity has expired, then the tool and the multitool on which the tool is located are disabled. The other tools on the multitool are not disabled.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

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										Chain_1_128
Loc.	MT LO.	Type	Tool name	ST	D	gth	ΔRadius	T C	Quantity	Set val	Prewar limit	D
1/4			SCHRUPPER	1	1 000		0.000					<input type="checkbox"/>
1/6			EINSTECHER	2	1 000		0.000					<input type="checkbox"/>
1/8			MULTITOOL2									<input checked="" type="checkbox"/>
	1		SCHRUPPER_HS	1	1 000		0.000	T	80.0	100.0	30.0	<input type="checkbox"/>
	2		SCHRUPPER_GS	1	1 000		0.000	C	0	20	10	<input checked="" type="checkbox"/>
1/10												

Reactivating

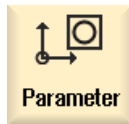
If a tool with expired lifetime or unit quantity that is mounted on a multitool is reactivated, then for this tool, the lifetime/unit quantity is set to the setpoint and the tool and the multitool are re-enabled (disable status is removed).

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 would like 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 to 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 refer to the machine manufacturer's specifications.

References

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

16.15.8 Relocating a multitool

Multitools can be directly relocated within magazines to another magazine location, which means that you do not have to unload multitools with the associated tools from the magazine in order to relocate them to a different location.

When you are relocating a multitool, the system automatically recommends an empty location. You may also directly specify an empty magazine location.

Procedure



1. Select the "Parameter" operating area.
2. Press the "Magazine" softkey.
3. Position the cursor at the multitool that you wish to relocate to a different magazine location.
4. Press the "Relocate" softkey.
The "... relocate from location ... to location ..." window is displayed. The "Location" field is pre-assigned with the number of the first empty magazine location.
5. Press the "OK" softkey to relocate the multitool to the recommended magazine location.

OR -

Enter the required magazine number in the "...magazine" field and the required magazine location number in "Location" field.

Press the "OK" softkey.

The multitool with the tools is relocated to the specified magazine location.

16.15.9 Positioning multitool

You can position a magazine. In this case, a magazine location is positioned to the loading point.

Multitools, which 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.

2. Position the cursor at the multitool location, which you wish to bring into the machining position (the multitool is located in the spindle).



3. Press the "Position multitool" softkey.

If the cursor is at a multitool location of a multitool, which is located in the spindle, the multitool location is positioned to the machining position.

Managing programs

17.1 Overview

You can access programs at any time via the Program Manager for execution, editing, copying, or renaming.

Programs that you no longer require can be deleted to release their storage space.

NOTICE

Execution from USB-FlashDrive

Direct execution from a USB-FlashDrive is not recommended.

There is no protection against contact problems, falling out, breakage through knocking or unintentional removal of the USB-FlashDrive during operation.

Disconnecting it during operation will result in the stopping of the machining and thus to the workpiece being damaged.

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 "ShopMill/ShopTurn" option for this.

In order to calculate different programs for multiple clamping, you require the option "Multiple clamping of different workpieces".

Storage for programs

Possible storage locations are:

- NC
- Local drive
- Network drives
- USB drives
- RS-232-C



Software options

To display the "Local drive" softkey, you require the "Additional 256 MB HMI user memory on CF card of the NCU" option (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-FlashDrive)
- Network drives

Choosing storage locations

In the horizontal softkey bar, you can select the storage location that contains the directories and programs that you want to display. In addition to the "NC" softkey, via which the passive file system data can be displayed, additional softkeys can be displayed.

The "USB" softkey is only operational when an external storage medium is connected (e.g. USB-FlashDrive on the USB port of the operator panel).

Displaying PDF and HTML documents

You can display HTML documents as well as PDFs on all drives of the program manager (e.g. in the local drive or USB) and via the data tree of the system data.

However, a preview of the documents is only possible for PDFs.

Structure of the directories

In the overview, the symbols 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.

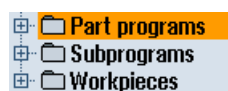


Figure 17-1 Program directory in the program manager

The plus sign in front of empty directories is removed after they have been read for the first time.

The directories and programs are always listed complete with the following information:

- Name
The name can be a maximum of 24 characters long
Permissible characters include all upper-case letters (without accents), numbers, and underscores.
- Type
Directory: WPD
Program: MPF
Subprogram: SPF
Initialization programs: INI
Job lists: JOB
Tool data: TOA
Magazine assignment: TMA
Zero points: UFR
R parameters: RPA
Global user data/definitions: GUD
Setting data: SEA
Protection zones: PRO
Sag: CEC
- Size (in bytes)
- Date/time (of creation or last change)

Active programs

Selected, i.e. active programs are identified using a green symbol.

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 17-2 Active program shown in green

17.1.1 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.

Proceed as follows



1. Select the "Program manager" operating area.



2. Press the "NC" softkey.

17.1.2 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 the option "Additional 256 MB 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.

Procedure



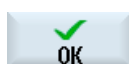
1. The local drive is selected.



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.

17.1.3 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
Execution from USB-FlashDrive
Direct execution from the USB-FlashDrive is not recommended.

Partitioned USB-FlashDrive (only 840D sl and TCU)

If the USB-FlashDrive has several partitions, these are displayed in a tree structure as a subtree (01,02,...).

For EXTCALL calls, enter the partition (e.g. USB:/02/... or //ACTTCU/FRONT/02/... or //ACTTCU/FRONT,2/... or //TCU/TCU1/FRONT/02/...)

You can also configure any partition (e.g. //ACTTCU/FRONT,3).

Procedure



1. Select the "Program manager" operating area.



2. Press the "USB" softkey.

Note

The "USB" softkey can only be operated when a USB-FlashDrive is inserted in the front interface of the operator panel.

17.2 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.

17.2 Opening and closing the program



5. Press the "NC Select" softkey to switch to the "Machine" operating area and begin execution.

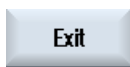


When the program is running, the softkey is deactivated.

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.

17.3 Executing a program

When you select a program for execution, the controller automatically switches 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



Program
manager

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.



Select

The controller 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.

Execution of the workpiece is started.

Note

Only workpieces/programs that are located in the NCK memory, local drive or USB drive can be selected for execution.

17.4 Creating a directory / program / job list / program list

17.4.1 Creating a new directory

Directory structures help you to manage your program and data transparently. You can create subdirectories in a directory on the local drive and on USB/network drives.

In a subdirectory, in turn, you can create programs and then create program blocks for them.

Note

Directory names must end in .DIR or .WPD. The maximum name length is 49 characters including the extension.

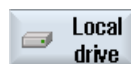
All letters (except accented characters), numbers, and underscores are permitted for name assignment. These names are automatically converted to upper-case letters.

This limitation does not apply for work on USB/network drives.

Procedure



1. Select the "Program manager" operating area.



2. Select your chosen 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.

17.4.2 Creating a new workpiece

You can set up various types of files such as main programs, initialization files, tool offsets, etc. in a workpiece.

Note

Workpiece directories

828D provides you with the option of nesting tool directories. You must note that the length of the call line is restricted. You will be informed if the maximum number of characters is reached when entering the workpiece name.

Under a workpiece directory (WPD), 840D sl does not allow any additional workpiece directories to be created in the NC memory. You can create a free directory structure on the local drive, USB and network drives; this means that you have the option to create other workpieces or any number of directories in a workpiece. If you copy this data to the NC memory then the length of the call line is checked.

Procedure



1. Select the "Program manager" operating area.



2. Select the desired storage location and position the cursor on the folder in which you would like to create a workpiece.



3. Press the "New" softkey.
The "New Workpiece" window appears.



4. If necessary, select a template if any are available.
5. Enter the desired workpiece name and press the "OK" softkey.

The name can be a maximum of 24 characters long.

You can use any letters (except accented), digits or the underscore symbol (_).

The directory type (WPD) is set by default.

A new folder with the workpiece name will be created.

The "New G Code Program" window opens.



6. Press the "OK" softkey again if you want to create the program.

The program will open in the editor.

17.4.3 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.



Program names can be a maximum of 24 characters long.

You can use all letters (with the exception of special characters, language-specific special characters, Asian or Cyrillic characters), numbers and underscores (_).

17.4.4 New ShopTurn program

In the part program and workpiece directories, you can create ShopTurn 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 "ShopTurn" softkey.
The "New Step Sequence Program" window opens.
The "ShopTurn" type is specified.



5. Enter the desired program name and press the "OK" softkey.

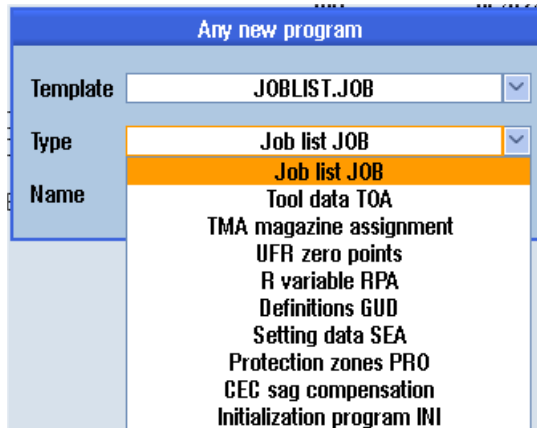
The program name can contain up to 28 characters (name + dot + 3-character extension).

You can use all letters (with the exception of special characters, language-specific special characters, Asian or Cyrillic characters), numbers and underscores (_).

17.4.5 Storing any new file

In each directory or subdirectory you can create a file in any format that you specify.

This does not apply to the NC memory. Here 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 appears



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).

The name can be a maximum of 24 characters long.

You can use any letters (except accented), digits or the underscore symbol (_).



5. Press the "OK" softkey.

17.4.6 Creating a job list

For every workpiece, you can create a job list for extended workpiece selection.

In the job list, you specify instructions for program selection in different channels.

Syntax

The job list contains the SELECT instructions.

SELECT <program> CH=<channel number> [DISK]

The SELECT instruction selects a program for execution in a specific NC channel. The selected program must be loaded into the working memory of the NC. The DISK parameter enables the selection of external execution (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.

17.4.7 Creating a program list

You can also enter programs in a program list that are then selected and executed from the PLC.

The program list may contain up to 100 entries.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure



1. Select the "Program manager" operating area.



2. Press the menu forward key and the "Program list" softkey. The "Prog.-list" window opens.



3. Place the cursor in the desired line (program number).



4. Press the "Select program" softkey.

The "Programs" window opens. The data tree of the NC memory with workpiece, part program and subprogram directory is displayed.



5. Place the cursor on the desired program and press the "OK" softkey.

The selected program is inserted in the first line of the list together with its path.

- OR -

Enter the program name directly in the list.

If you are making entries manually, check that the path is correct (e.g. //NC/WKS.DIR/MEINPROGRAMM.WPD/MEINPROGRAMM.MPF).

//NC and the extension (.MPF) may be added automatically.

With multi-channel machines, you can specify in which channel the program is to be selected.



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.

17.5 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.

17.6 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

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 appears.

3. Enter the desired search term in the "Text" field.

Note: When searching for a file, enter the complete name with extension (e.g. DRILLING.MPF).

4. When required, activate the checkbox "Observe upper and lower case".



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.



17.7 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.

17.8 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
 SELECT	Renders or expands a selection. You can only select individual elements.
 SHIFT  	Renders a consecutive selection.
 INSERT	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 +  SHIFT Pressed	Expand selection consecutively up to the next click.
Left mouse +  CTRL Pressed	Expand selection to individual elements by clicking. An existing selection will expand to include the element you clicked.

17.9 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 in a different location.

Note

You can only paste directories on local drives and on USB or network drives.

Note

Write protection

If the current directory is write-protected for the user, then the function is not offered.

Note

When you copy directories, any missing endings are added automatically.

All letters (except accented characters), numbers, and underscores are permitted for name assignment. The names are automatically converted to upper-case letters, and extra dots are converted to underscores.

Example

If the name is not changed during the copy procedure, 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.

If a directory/program of the same name already exists in this directory, you are informed. 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.

17.10 Deleting a directory/program

Delete programs or directories from time to time that you are no longer using to maintain a clearer overview of your data management. Back up the data beforehand, if necessary, on an external data medium (e.g. USB FlashDrive) or on a network drive.

Please note that when you delete a directory, all programs, tool data and zero point data and subdirectories that this directory contains are deleted.

Temp directory for ShopTurn

If you want to free up space in the NCK memory, delete the contents of the "TEMP" directory. This is where ShopTurn stores the programs that are created internally for calculating solid machining operations.

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.

17.11 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 executed 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.

References

A detailed description of the configuration can be found in the following documentation:

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

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.

17.12 Viewing PDF documents

You can display HTML documents as well as PDFs on all drives of the program manager (e.g. in the local drive or USB) and 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, i.e. the local or USB drive.

- 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 that you want to display and press the "Open" softkey.

The PDF is shown 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 or reduce the size of the display.

4. Press the "Search" softkey if you want to search for specific texts in the PDF.

5. Press the "View" softkey to change the representation of the PDF. A new vertical softkey bar appears.

6. Press the "Zoom page width" softkey to display the document in full width on the screen.

- OR -

Press the "Zoom page height" 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.

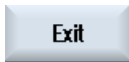
- OR -



Press the "Rotate right" softkey to rotate the document through 90 degrees to the right.



7. Press the "Back" softkey to return to the previous window.



8. Press the "Close" softkey to exit the PDF display.

17.13 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 (.wpc):

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
"/card/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 the option "Additional 256 MB HMI user memory on CF card of the NCU" (not for SINUMERIK Operate on PCU50/PC).

NOTICE
<p>Execution from USB-FlashDrive</p> <p>Direct execution from a USB-FlashDrive is not recommended.</p> <p>There is no protection against contact problems, falling out, breakage through knocking or unintentional removal of the USB-FlashDrive during operation.</p> <p>Disconnecting it during operation will result in immediate stopping of the machining and, thus, to the workpiece being damaged.</p>



Machine manufacturer

Processing EXTCALL calls can be enabled and disabled.
Please refer to the machine manufacturer's specifications.

17.14 Backing up data

17.14.1 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 and, using the cursor keys or the mouse, select the required directories or files.



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 "" (replaces any character string) and "?" (replaces 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 (e.g. archive ARC (binary format)), enter the desired name and press the "OK" softkey.
A message informs you if archiving was successful.

17.14.2 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 contents of the selected files (XML, ini, hsp, syf files, programs) in a preview.

You can display information about the file, such as path, name, date of creation and change, in a Properties window.

Requirement

The access rights depend on the relevant areas and range from protection level 7 (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 option "Additional HMI user memory on CF card of NCU".

NOTICE

USB-FlashDrive

USB-FlashDrives are not suitable as persistent memory media.

Procedure



1. Select the "Start-up" 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 and, using the cursor keys or the mouse, select the required directories or files.



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 "*" (replaces any character string) and "?" (replaces 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 (e.g. archive ARC (binary format)), enter the desired 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 in .arc format is created in the selected directory.

17.14.3 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

In order to read-in user archives in the "Program Manager" operating area, you require the option "Additional HMI memory on CF Card of NCU" (not for SINUMERIK Operate on PCU50 / PC).

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 (*.arc) 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.

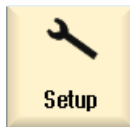
See also

Searching directories and files (Page 712)

17.14.4 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 "Start-up" 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.



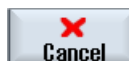


- OR -

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.

17.15 Setup data

17.15.1 Backing up setup data

Apart from programs, you can also save tool data and zero point settings.

You can use this option, for example, to back up tools and zero point data for a specific machining step program. If you want to execute this program at a later point in time, you will then have quick access to the relevant settings.

Even tool data that you have measured on an external tool setting station can be copied easily into the tool management system using this option.

Backing-up job lists

If you wish to backup a job list, which contains ShopTurn 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	
Tool data	<ul style="list-style-type: none"> • No • All used in the program (only for ShopTurn program and job list only with ShopTurn programs) • Complete tool list
Tool data for ShopTurn programs -- only available for job list with ShopTurn and G code programs	<ul style="list-style-type: none"> • No • All used in the program • Complete tool list
Tool data for G code programs -- only available for job list with ShopTurn and G code programs	<ul style="list-style-type: none"> • No • Complete tool list
Magazine assignment	<ul style="list-style-type: none"> • Yes • No

Data	
Zero points	<ul style="list-style-type: none"> • No The selection box "Basis zero point" is hidden • All used in the program (only for ShopTurn program and job list only with ShopTurn programs) • All
Zero points for ShopTurn programs -- only available for job list with ShopTurn and G code programs	<ul style="list-style-type: none"> • 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 ShopTurn and G code programs	<ul style="list-style-type: none"> • No The selection box "Basis zero point" is hidden • All
Basic zero points	<ul style="list-style-type: none"> • No • Yes
Directory	The directory is displayed, in which the selected program is located.
File name	Here you have the option of changing the suggested file names.

Note**Magazine assignment**

You can only read out the magazine assignments if your system provides support for loading and unloading tool data to and from the magazine.

Procedure



1. Select the "Program Manager" operating area.



2. Position the cursor on the program whose tool and zero point data you wish to back up.

...



3. Press the ">>" and "Archive" softkeys.



4. Press the "Setup data" softkey.
The "Backup setup data" window opens.



5. Select the data you want to back up.
6. When required, change the specified name of the originally selected program here in the "File name" field.



7. Press the "OK" softkey.
The setup data will be set up in the same directory in which the selected program is stored.
The file is automatically saved as INI file.

Note

Program selection

If a main program as well as an INI file with the same name are in a directory, when selecting the main program, initially, the INI file is automatically started. In this way, unwanted tool data can be changed.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

17.15.2 Reading-in set-up data

When reading-in, you can select which of the backed-up data you wish to read-in:

- 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.

17.16 RS-232-C

17.16.1 Reading-in and reading-out archives

Availability of the RS-232-C serial interface

You have the option of reading-out and reading-in archives in the "Program Manager" operating area as well as in the "Start-up" operating area via the RS-232-C serial interface.

- SINUMERIK Operate in the NCU

The softkeys for the RS-232-C are available as soon as an option module is connected and the slot is occupied.

- SINUMERIK Operate on PCU 50.3 and PCU 50.5

The softkeys for the RS-232-C are always available.

Reading-out archives

The files to be sent (directories or 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

Only archives can be read-in via the RS-232-C. These are transferred and then subsequently unzipped.

Note

Start-up archive

If you read in a start-up archive via the RS-232-C, 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. Using the SinuCom commissioning and service tool SinuCom ARC, you can process the archive in the binary format and in the commissioning archive.

Procedure



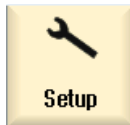
1. Select the "Program manager" operating area, and press the "NC" or "Local drive" softkey.



...



- OR -



- Select the "Start-up" operating area and press the "System data" softkey.



Reading-out archives

2. Select the directories or the files that you wish to send to RS-232-C.
3. Press the ">>" and "Archive" softkeys.



4. Press the "RS-232-C send" softkey.

- OR -

Reading in an archive



- Press the "RS-232-C receive" softkey if you wish to read-in files via RS-232-C.

17.16.2 Setting V24 in the program manager

V24 setting	Meaning
Protocol	The following protocols are supported for transfer via the V24 interface: <ul style="list-style-type: none"> • RTS/CTS (default setting) • Xon/Xoff
Transfer	It is also possible to use a secure protocol for data transfer (ZMODEM protocol). <ul style="list-style-type: none"> • Normal (default setting) • secure <p>For the selected interface, secure data transfer is set in conjunction with handshake RTS/CTS.</p>
Baud rate	Transfer rate: Transfer rates of up to 115 kbaud can be selected. The baud rate that can be used depends on the connected device, the cable length and the general electrical conditions. <ul style="list-style-type: none"> • 110 • • 19200 (default) • ... • 115200
Archive format	<ul style="list-style-type: none"> • Punched tape format (default setting) • Binary format (PC format)
V24 settings (details)	
Interface	<ul style="list-style-type: none"> • 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). <ul style="list-style-type: none"> • None (default setting) • Odd • Even
Stop bits	Number of stop bits for asynchronous data transfer. <ul style="list-style-type: none"> • 1 (default) • 2
Data bits	Number of data bits for asynchronous data transfer. <ul style="list-style-type: none"> • 5 bits • ... • -8 bits (default setting)
XON (hex)	Only for punched tape format
XOFF (hex)	Only for punched tape format

V24 setting	Meaning
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



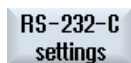
1. Select the "Program Manager" operating area.



2. Press the "NC" or "Local drive" softkey.



3. Press the ">>" and "Archive" softkeys.



4. Select the "V24 settings" softkey.
The "Interface: V24" window 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.

Setting up drives

18.1 Overview

Set up connections

Up to 8 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
- CompactFlash Card of the NCU, only for SINUMERIK Operate in the NCU
- Network drives
- Local hard disk of the PCU 50.3, only for SINUMERIK Operate on the PCU



Software option

In order to use the CompactFlash Card as data carrier, you require the option "Additional 256 MB HMI user memory on CF card of NCU" (not for SINUMERIK Operate of PCU50 / PC).

Note

The USB interfaces of the NCU are not available for SINUMERIK Operate and therefore cannot be configured.

18.2 Setting up drives



The "Set-Up Drives" window is available in the "Start-up" operating area for configuration.

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
Type	No drive	No drive defined.
	USB local	Access to the USB memory medium is only realized via the TCU to which it is connected. USB drives are automatically identified if the memory medium is inserted when SINUMERIK Operate powers-up.
	USB global	All of the TCUs in the plant network can access the USB memory medium. - USB global is not possible under Windows!
	NW Windows	Network drive
	Local drive	Local drive Hard disk or user memory on the CompactFlash card
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.
	X204	For SIMATIC Thin Client the USB interface is X204.
Device		Names of the TCU to which the USB storage medium is connected, e.g. tcu1. The NCU must already know the TCU name.
Partition		Partition number on the USB memory medium, e.g. 1. If a USB hub is used, specify the USB port of the hub.
Path		<ul style="list-style-type: none"> Start direct directory of the data carrier that is connected via the local drive. Network path to a directory that has been released in the network. This path must always start with "//", e.g. //Server01/share3.
Access level		Assign access rights to the connections: From protection level 7 (key switch position 0) to protection level 1 (password: Manufacturer). The particular assigned protection 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 it is automatically separated into several lines. If a blank is present, then this is used as line separator.

Entry		Meaning
Softkey icon	No icon	No icon is displayed on the softkey.
	sk_usb_front.png 	Icon file name. Is displayed on the softkey.
	sk_local_drive.png 	Icon file name. Is displayed on the softkey.
Text file	slpmdialog	File for softkey dependent on the language. If nothing is specified in the input fields, the text appears on the softkey as was specified in the input field "Softkey text". If your own text files are saved, then the text ID, which is used to search for the text file, is specified in the "Softkey text" input field.
Text context	SIPmDialog	
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.

Error messages

Error message	Meaning
Error occurred when closing a connection	An existing drive was not able to be deactivated.
Error occurred when establishing a connection.	Drive connection was not able to be established.
Error occurred while establishing a connection: Incorrect entry or no authorization.	Drive connection was not able to be established.
Incorrect data	The entered data is either incorrect or inconsistent.
Function not available	The function is not supported with the current software release.
Unknown error - error code:%1	Error was not able to be assigned.

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 data for the corresponding drive or enter the necessary data.
4. Press the "Activate drive" softkey.
The drive is activated.

The operating system now checks the entered data and whether the connection is established. An OK message is output in the dialog line if an error is not identified.

The drive can be accessed.

- OR -

If the operating system identifies an error, then you receive an error message.

Press the "OK" softkey.



You then return to the "Set Up Drives" window. Check and correct your entries and re-activate the drive.



If you press the "OK" softkey, the modified data is accepted without any feedback message. You do not receive any message about a successful or unsuccessful connection.



If you press the "Cancel" softkey, then all of the data that has not been activated is rejected.

19.1 HT 8 overview

The mobile SINUMERIK HT 8 handheld terminal combines the functions of an operator panel and a machine control panel. It is therefore suitable for visualization, operation, teach in, and programming at 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.

It also has membrane keys for traversing the axes, for numeric input, for cursor control, and for machine control panel functions like start and stop.

It is equipped with an emergency stop button and two 3-position enabling buttons. You can also connect an external keyboard.

References

For more information about connection and startup of the HT 8, see the following references:

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

Customer keys

The four customer keys are freely assignable and can be set up customer-specifically by the machine manufacturer.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Integrated machine control panel

The HT 8 has an integrated machine control panel consisting of keys (e.g. start, stop, traversing keys, etc.), and keys reproduced as softkeys (see machine control panel menu).

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.

Enabling button

The HT 8 has two enabling buttons. Thus, you can initiate enabling functions for operations that require enabling (e.g. displaying and operating of traversing keys) with either your right hand or your left hand.

Enabling buttons are available for the following key positions:

- Released (no activation)
- Enabling (center position) - enabling for channel 1 and 2 is on the same switch
- Panic (completely pushed through)

Traversing keys

To traverse the axes of your machine using the traversing keys of the HT 8, you must select "JOG" mode or either the "Teach In" or "Ref.Point" submode. Depending on the setting, the enabling button must be activated.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

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 can display the operating area menu by touching the display symbol for the active operating area.

Handwheel

The HT 8 is available with a hand wheel.

References

For information about connecting the hand wheel, refer to:

Operator Components and Networking Manual; SINUMERIK 840D sl/840Di sl

19.2 Traversing keys

The traversing keys are not labeled. However, you can display a label for the keys in place of the vertical softkey bar.

Labeling of the traversing keys is displayed for up to six axes on the touch panel by default.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Showing and hiding

You can link the showing and hiding of the label to activation of the enabling button, for example. In this case, the traversing keys are displayed when you press the enabling button.

If you release the enabling button, the traversing keys are hidden again.



Machine manufacturer

Please refer to the machine manufacturer's specifications.



All existing vertical and horizontal softkeys are covered or hidden, i.e. other softkeys cannot be used.

19.3 Machine control panel menu

Here you select keys from the machine control panel which are reproduced by the software by touch operation of the relevant softkeys.

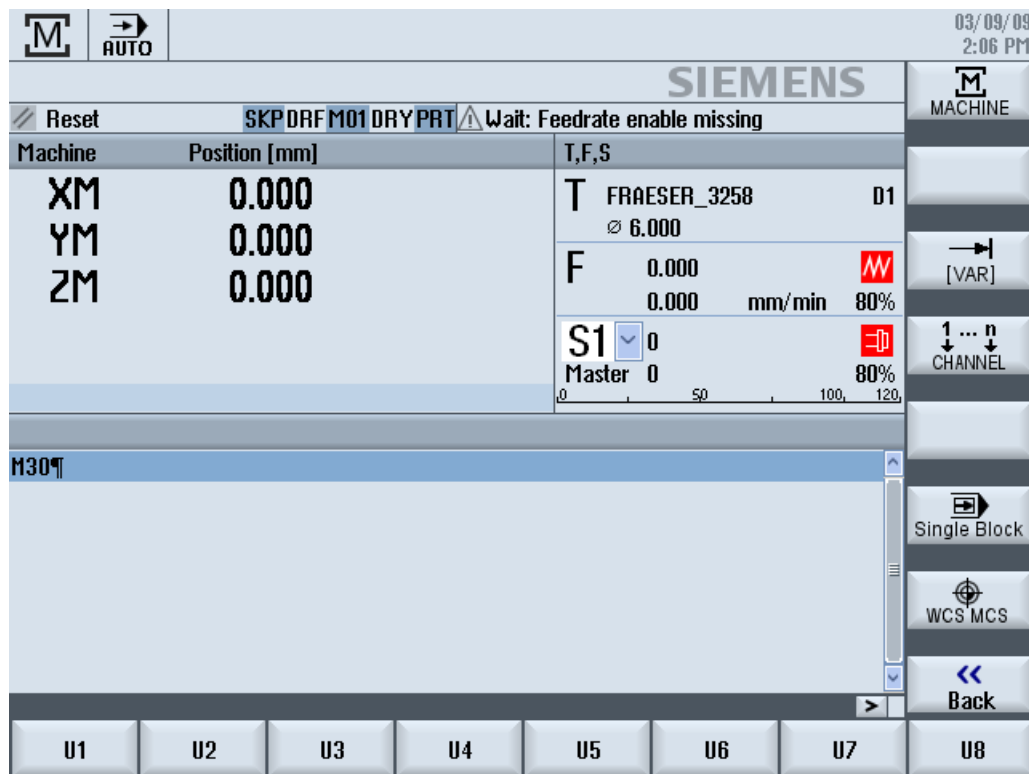
See chapter "Controls on the machine control panel" for a description of the individual keys.

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).



You can expand the user softkey bar to display eight additional softkeys via the menu forward key.



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	Change the channel
"Single Block" softkey	Switch single block execution on/off
"WCS MCS" softkey	Switch between WCS and MCS
"Back" softkey	Close the window.

Note

The window will automatically disappear when changing regions areas with the "Menu Select" key.

19.4 Virtual keyboard

The virtual keyboard is used as the input device for touch operator panels.

It opens when you double-click an operator element with input capability (editor, edit field). The virtual keyboard can be positioned anywhere on the operator interface. In addition, you can toggle between a full keyboard and a reduced keyboard that only includes the number block. Moreover, with the full keyboard, you can toggle between English key assignments and the keyboard assignment for the current language setting.

Procedure

1. Click in the required input field in order to place the cursor there.
2. Click the input field.
The virtual keyboard is displayed.
3. Enter your values via the virtual keyboard.
4. Press the <INPUT> key.



- OR -

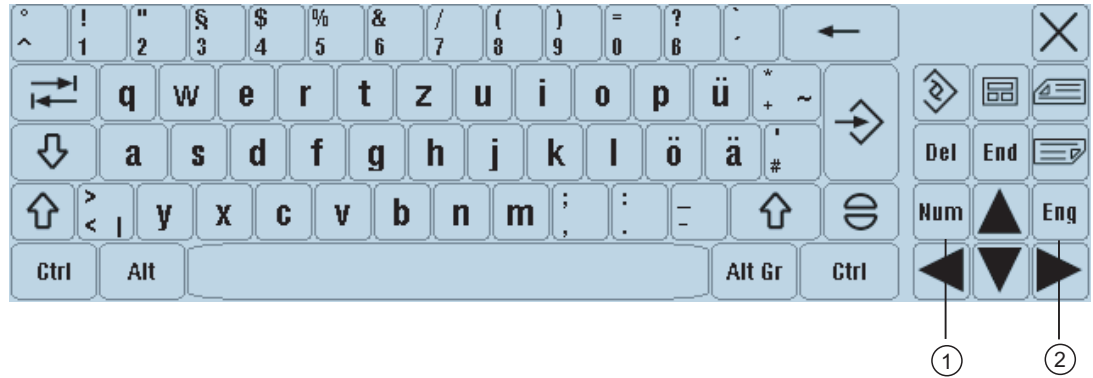
Position the cursor on an another operator element.

The value is accepted and the virtual keyboard is closed.

Positioning of the virtual keyboard

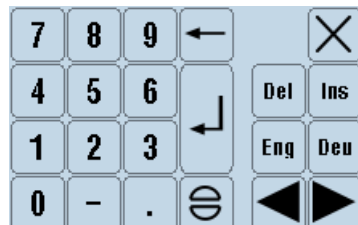
You can position the virtual keyboard anywhere in the window by pressing the empty bar next to the "Close window" icon with your finger or a stylus and moving it back and forth.

Special keys on the virtual keyboard



- 1 Num:
Reduces the virtual keyboard to the number block.
- 2 Eng:
Toggles the keyboard assignment between the English keyboard assignment and the keyboard assignment for the current language setting.

Number block of the virtual keyboard



Use the "Deu" or "Eng" keys to return to the full keyboard with the English keyboard assignment or the keyboard assignment of the current language setting.

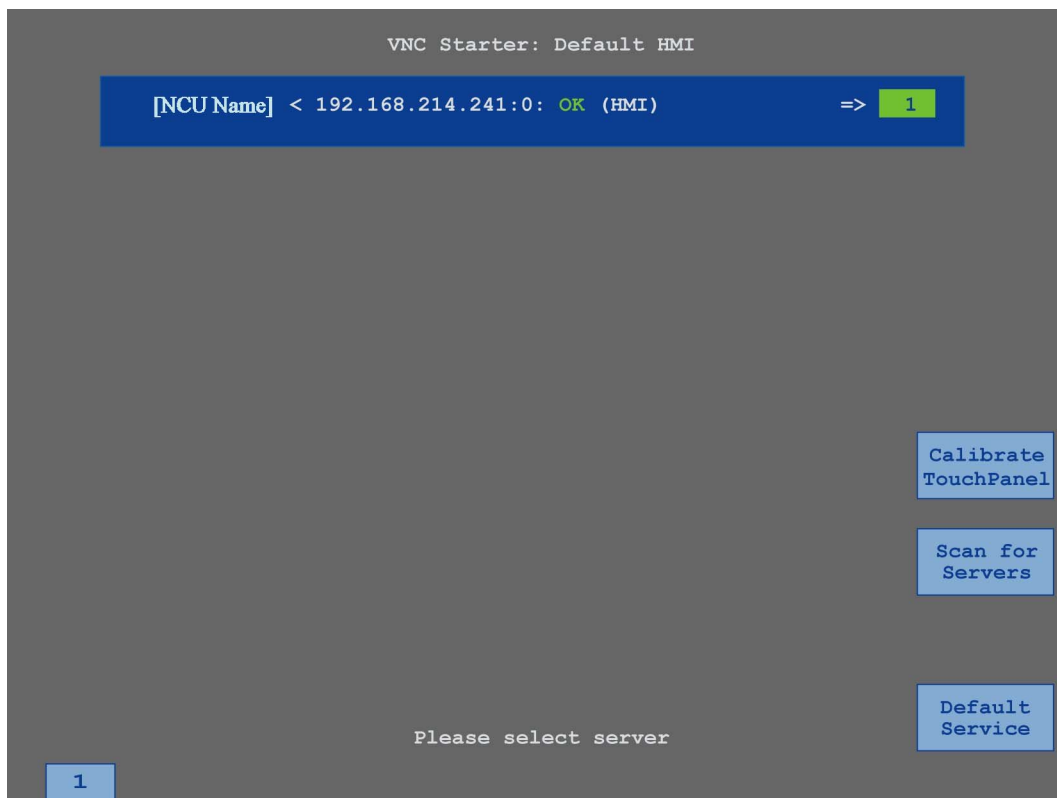
19.5 Calibrating the touch panel

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 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

20.1 Overview

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 controller 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).

NOTICE
Ctrl-E Deactivating profiles
Disable Ctrl-E profiles before a series startup in order to prevent the NCU unintentionally shutting down.

**Machine manufacturer**

Please refer to the machine manufacturer's specifications.

20.2 Displaying energy consumption

You have the option of displaying the actual energy consumption values of the individual axes and the total energy consumption.

Display	Meaning
Manual	<ul style="list-style-type: none"> No input Entering and displaying a fixed value Displaying a value from a data source
Sentron PAC	Displaying an actual value measured by a SENTRON PAC device.
Total, drives	Display of the total of all measured drive values.
Total, machine	Addition of the values of all axes as well as a fixed value and Sentron PAC.

The display in the table depends on the configuration.

References

Information on the configuration is provided in the following reference:

System Manual "Ctrl-Energy", SINUMERIK 840D sl / 828D

Procedure



1. Select the "Parameter" operating area and press the "Ctrl-Energy" softkey.



- OR -



+

Press the <Ctrl> + <E> keys.



2. Press the "Energy analysis" softkey.
The "SINUMERIK Ctrl-Energy Analysis" window opens.
You obtain a display of the actual active power for all of the axes.

20.3 Measuring and saving the energy consumption

For the currently selected axes, 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. The measurement should take single drives into account.

They specify in which channel the start and stop of the part program should be initiated and which number of repetitions are to be measured.

Save measurements

Save the measured consumption values so that you can subsequently compare the data.

Note

Up to three data records are displayed. The oldest data record is automatically overwritten if there are more than three measurements.

Measurement duration







The measurement duration is limited. When the maximum measurement duration is reached, the measurement is terminated and a corresponding message is output in the dialog line.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure

- | | |
|---|---|
|  | 1. The "Ctrl-Energy Analysis" window opens. |
|  | 2. Press the "Start measurement" softkey.
The "Setting Measurement: Select Device" window opens. |
|  | 3. Select the desired device in the list, possibly activate the "Measure part program" checkbox, enter the number of repetitions, possibly select the desired channel, and press the "OK" softkey.
The trace is started. |
|  | 4. Press the "Stop measurement" softkey.
The measurement is ended. |
|  | 5. Press the "Graphic" softkey, to track the measurement characteristic. |
|  | 6. If you want to track the consumption values, press the "Details" softkey. |
| | 7. 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.

References

Information on the configuration is provided in the following reference:
System Manual "Ctrl-Energy", SINUMERIK 840D sl / 828D

20.4 Long-term measurement of the energy consumption

The long-term measurement of energy consumption is performed in the PLC and saved. The values from times in which the HMI is not active are also recorded.

Measured values

The infeed and regenerative power values as well as the sum of the power are displayed for the following periods:

- Current and previous day
- Current and previous month
- Current and previous year

Requirement

SENTRON PAC is connected.

Procedure



1. The "Ctrl-Energy Analysis" window opens.
2. Press the "Long-term measurement" softkey.
The "SINUMERIK Ctrl-Energy Analysis Long-term Measurement" window opens.
The results of the long-term measurement are displayed.
4. Press the "Back" softkey to terminate the long-term measurement.

20.5 Displaying measured curves

Display	Meaning
Start of the measurement	Shows the time at which the measurement was started by the pressing the "Start measurement" softkey.
Duration of the measurement [s]	Shows the measuring duration in seconds until the "Stop measurement" softkey is pressed.
Device	Displays the selected measured component. <ul style="list-style-type: none"> • Manual • Total, drives • Total, machine
Supplied energy [kWh]	Shows the supplied energy of 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.

Procedure





1. The "SINUMERIK Ctrl-Energy Analysis" window opens and you have already performed and saved measurements.
2. Press the "Graphic" and "Saved measurements" softkeys.
The measurement curves are displayed in the "SINUMERIK Ctrl-Energy Analysis" window.
3. Press the "Saved measurements" softkey again if you only want to see the actual measurement.
4. Press the "Details" softkey for the exact measurement data and consumption values of the last three saved measurements and possibly a current measurement.

20.6 Using the energy-saving profile

In the "SINUMERIK Ctrl-Energy Energy-Saving Profile" window, you can display all of the defined energy-saving profiles. Here, directly activate the required energy-saving profile - or inhibit or release profiles.

SINUMERIK Ctrl-Energy energy-saving profiles

Display	Meaning
Energy-saving profile	All energy-saving profiles are listed.
active in [min]	The remaining time up 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.

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

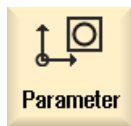
The selection and function of the displayed energy-saving profiles can differ. Please refer to the machine manufacturer's specifications.

References

Information on the configuration of the energy-saving profiles is provided in the following reference:

System Manual "Ctrl-Energy", SINUMERIK 840D sl / 828D

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 Energy-Saving Profiles" window opens.



3. Position the cursor on the required energy-saving profile and press the "Activate immediately" softkey if you wish to directly activate this state.



4. 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 disabled. The energy-saving profile cannot be active, i.e. it is grayed out and displayed without time details.
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 can be non-active.



6. Press the "Enable all" softkey to withdraw the disable for all profiles.

Easy Message (828D only)

21.1 Overview

Easy Message enables you to be informed about certain machine states by means of SMS messages via a connected modem:

- For example, you would like to be informed about emergency stop states
- You would like to know when a batch has been completed

Control commands

- HMI commands are used to activate or deactivate a user.

Syntax: [User ID] deactivate, [User ID] activate

- A special area is reserved in the PLC to which you can send commands in the form of PLC bytes using SMS commands.

Syntax: [User ID] PLC DataByte

The user ID is optional and required only if a corresponding ID has been specified in the user profile. The string PLC indicates that a PLC byte is to be written. It is followed by the data byte to be written in the following format: Base#Value. Base can take the values 2, 10 and 16 and defines the number base. Then follows the separator # and the value of the byte. Only positive values are allowed to be sent.

Examples:

2#11101101

10#34

16#AF



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Active users

In order to receive an SMS message for certain events, you must be activated as user.

User logon

As registered user, you can log on via SMS to inquire about messages.

Action log

You can obtain precise information about incoming and outgoing messages via SMS logs.

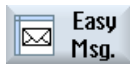
References

Information on the GSM modem can be found in the
PPU SINUMERIK 828D Manual

Calling the SMS Messenger



1. Select the "Diagnostics" operating area.



2. Press the "Easy Msg." softkey.

21.2 Activating Easy Message

To commission the connection to the modem for the SMS Messenger, activate the SIM card at the initial start-up.

Requirement

The modem is connected and activated.



Machine manufacturer

The modem is activated via the machine data 51233
\$MSN_ENABLE_GSM_MODEM.

Please refer to the machine manufacturer's specifications.

Procedure

Activating the SIM card



1. Press the "Easy Msg." softkey.
The "SMS Messenger" window appears.

"Status" shows that the SIM card has not been activated with a PIN.



2. Enter the PIN, repeat the PIN and press the "OK" softkey.



3. If you made an incorrect entry several times, enter the PUK code in the "PUK Input" window and press the "OK" softkey to activate the PUK code.

The "PIN input" window is opened and you can enter the PIN number as usual.

Activating a new SIM card



1. Press the "Easy Msg." softkey.
The "SMS Messenger" window appears.

"Status" shows that the connection to the modem has been activated.



2. Press the "Settings" softkey.



3. Press the "Delete PIN" softkey to delete the stored PIN.
Enter the new PIN in the "PIN Input" window at the next power up.

21.3 Creating/editing a user profile

User identification

Display	Meaning
User name	Name of the user to be created or logged on.
Telephone number	Telephone number of the user to which the messages are to be sent. The telephone number must include the country code in order that control commands can identify the sender (e.g. +491729999999)
User ID	<p>The user ID has 5 digits (e.g. 12345)</p> <ul style="list-style-type: none"> • It is used to activate and deactivate the user via SMS. (e.g. "12345 activate") • The ID is used to additionally verify the incoming and outgoing messages and to activate the control commands.

Events that can be selected





You must set-up the events for which you receive notification.

Requirement

The modem is connected.

Procedure

Creating a new user

- 
 1. Press the "User profiles" softkey.
The "User Profiles" window appears.
- 
 2. Press the "New" softkey.
3. Enter the name and telephone number of the user.
4. If required, enter the ID number of the user.
5. In the area "send SMS for the following events" area, activate the appropriate checkbox and when required, enter the desired value (e.g. the unit quantity, which when it is reached, a notification should be sent).
- OR -
- 
 Press the "Default" softkey.
The appropriate window is opened and displays the default values.
- 
 6. Press the "Send test message" softkey.
An SMS message with predefined text is sent to the specified telephone number.

Editing user data and events



1. Select the user whose data you want to edit and press the "Edit" softkey.

The input fields can be edited.

2. Enter new data and activate the desired settings.

- OR -



Press the "Default" softkey to accept the default values.

21.4 Setting-up events

In the "Send SMS for the following events" area, select the events using the check box, which when they occur, an SMS is sent to the user.

- Programmed messages from the part program (MSG)

In the part program, program an MSG command via which you receive an SMS.

Example: MSG ("SMS: An SMS from a part program")

- Select the following events using the <SELECT> key
 - The workpiece counter reaches the following value
An SMS is sent if the workpiece counter reaches the set value.
 - The following program progress is reached (percent)
An SMS is sent if, when executing a part program, the set progress is reached.
 - Actual NC program reaches runtime (minutes)
An SMS is sent after the set runtime has been reached when executing an NC program.
 - Tool usage time reaches the following value (minutes)
An SMS is sent if the usage time of the tool reaches the set time when executing a part program (derived from \$AC_CUTTING_TIME).
- Messages/alarms from the Tool Manager
An SMS is sent if messages or alarms are output to the Tool Manager.
- Measuring cycle messages for tools
An SMS is sent if measuring cycle messages are output that involve tools.
- Measuring cycle messages for workpieces
An SMS is sent if measuring cycle messages are output that involve workpieces.
- Sinumerik messages/alarms (error when executing)
An SMS is sent if NCK alarms or messages are output that cause the machine to come to a standstill.
- Machine faults
An SMS is sent if PLC alarms or messages are output that cause the machine to come to a standstill (i.e. PLC alarms with Emergency Off response).

- Maintenance intervals
An SMS is sent if the service planner registers pending maintenance work.
- Additional alarm numbers:
Here, specify additional alarms where you should be notified if they occur.
You can enter individual alarms, several alarms or alarm number ranges.
Examples:
1234,400
1000-2000
100,200-300

Requirement

- The user profile window is opened.
- You selected the event "Measuring cycle messages for tools", "Measuring cycle messages for workpieces", "Sinumerik messages/alarms (errors when executing)", "Machine faults" or "Maintenance intervals".

Editing events



1. Activate the required check box and press the "Details" softkey.
The appropriate window opens (e.g. "Measuring cycle messages for workpieces") and shows a list of the defined alarm numbers.



2. Select the corresponding entry and press the "Delete" softkey to remove the alarm number from the list.

- OR -



Then press the "New" softkey if you wish to create a new entry.
The "Create new entry" window opens.



Enter the data and press the "OK" softkey to add the entry to the list.



Press the "Save" softkey to save the settings for the result.



3. Press the "Standard" softkey to return to the standard settings for the events.

21.5 Logging an active user on and off

Only active users receive an SMS message for the specified events.

You can activate users, already created for Easy Message, with certain control commands via the user interface or via SMS.



The connection has been established to the modem.

Procedure



1. Press the "User profiles" softkey.



2. Select the desired user in the User name field and press the "User active" softkey



Note

Repeat step 2 to activate further users.

- OR -

Send an SMS with the User ID and the "activate" text to the control (e.g. "12345 activate")



If the telephone number and the user ID match the stored data, the user profile is activated.

You receive a message of success or failure per SMS.



3. Press the "User active" softkey to log off an activated user.





- OR -

Send an SMS with the "deactivate" text (e.g. "12345 deactivate") to log off from the Messenger.

An SMS message is not sent to the deactivated user for the events specified in the user profile.

21.6 Displaying SMS logs

The SMS data traffic is recorded in the "SMS Log" window. In this way, it is possible to see the chronological sequence of activates when a fault occurs.

Symbols	Description
	Incoming SMS message for the Messenger.
	Message that has reached the Messenger, but which has not been processed (e.g. incorrect user ID or unknown account).
	SMS message sent to a user.
	Message that has not reached the user because of an error.

Requirement



The connection has been established to the modem.

Procedure



1. Press the "SMS log" softkey.



The "SMS Log" window appears.

All the messages that have been sent or received by the Messenger are listed.



Note

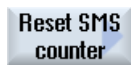
Press the "Incoming" or "Outgoing" softkey to restrict the list.

21.7 Making settings for Easy Message

You can change the following Messenger configuration in the "Settings" window:

- Name of the controller that is part of an SMS message
- Number of sent messages
 - The SMS counter provides information on all sent messages.
 - Limit the number of sent messages in order to receive an overview of the costs through SMS messages, for example.

Setting the SMS counter to zero



When a set limit is reached, no further SMS messages are sent. Press the "Reset SMS counter" softkey to reset the counter to zero.

Requirement



The connection has been established to the modem.

Procedure



1. Press the "Settings" softkey.
2. Enter an arbitrary name for the controller in the "Machine name" field.
3. If you want to limit the number of sent SMS messages then select the "Specify limit for SMS counter" entry and enter the desired number.



When the maximum number of messages is reached, you obtain a corresponding error message.

Note

Check the SMS log to see the exact time when the limit was reached.

- OR -



3. Press the "Default" softkey. If you have freely selected a machine name, this is replaced by a default name (e.g. 828D).

Easy Extend (828D only)

22.1 Overview

Easy Extend enables machines to be retrofitted with additional units, which are controlled by the PLC or that require additional NC axes (such as bar loaders, swiveling tables or milling heads), at a later point in time. These additional devices are easily commissioned, activated, deactivated or tested with Easy Extend.

The communication between the operator component and the PLC is performed via a PLC user program. The sequences to be executed for the installation, activation, deactivation and testing of a device are stored in a statement script.

Available devices and device states are displayed in a list. The view of the available devices can be controlled for users according to their access rights.

The subsequent chapters are selected for example only and are not available in every statement list.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Up to 64 devices can be managed.

References

SINUMERIK 828D Turning and Milling Commissioning Manual

22.2 Enabling a device

The available device options are protected by a password.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure



1. Select the "Parameter" operating area.



2. Press the menu forward key and then the "Easy Extend" softkey.
A list of the connected devices is displayed.







3. Press the "Enable function" softkey.
The "Enabling of the Devices Option" window is opened.



4. Enter the option code and press the "OK" softkey.
A tick appears in the appropriate checkbox in the "Function" column and the function is enabled.

22.3 Activating and deactivating a device

Status	Meaning
	Device activated
	System waiting for PLC checkback signal
	Device faulty
	Interface error in the communication module

Procedure



1. Easy Extend is opened.



2. You can select the desired device in the list with the <Cursor up> and <Cursor down> keys.



3. Position the cursor on the device option for which the function has been unlocked and press the "Activate" softkey.
The device is marked as activated and can now be used.



4. Select the desired activated device and press the "Deactivate" softkey to switch the device off again.

22.4 Commissioning Easy Extend

As a rule, the "Easy Extend" function has already been commissioned by the machine manufacturer. If a first commissioning has not been performed or if, for example, function tests are to be performed again (e.g. when retrofitting additional devices), this is possible at any time.

The "Start-up" softkey has been declared as Manufacturer data class (M).

Procedure



1. Select the "Parameter" operating area.



2. Press the menu forward key and then the "Easy Extend" softkey.



3. Press the "Start-up" softkey.
A new vertical softkey bar appears.



4. Press the "Comm. start-up" softkey to start the commissioning.
Before starting, a complete data backup is generated which you can then use in an emergency.



5. Press the "Cancel" softkey if you want to abort the commissioning prematurely.



6. Press the "Restore" softkey to load the original data.



7. Press the "Device function test" softkey to test the machine manufacturer's intended function.

Service Planner (828D only)

23.1 Performing and monitoring maintenance tasks

With the "Service Planner", maintenance tasks have been set up that have to be performed at certain intervals (e.g. top up oil, change coolant).

A list is displayed of all the maintenance tasks that have been set up together with the time remaining until the end of the specified maintenance interval.

The current status can be seen in the status display.

Messages and alarms indicate when a task has to be performed.

Acknowledging a maintenance task




Acknowledge the message when a maintenance task has been completed.

Note

Protection level

You require protection level 2 (service) to acknowledge completed maintenance tasks.

Service Planner

Display	Meaning	
Pos	Position of the maintenance task in the PLC interface.	
Maintenance task	Name of the maintenance task.	
Interval [h]	Maximum time until next servicing in hours.	
Remaining time [h]	Time until the interval expires in hours.	
Status	  	Display of the current status of a maintenance task. The maintenance task has been started. The maintenance task is completed. The maintenance task is deactivated.

Procedure



1. Select the "Diagnostics" operating area.



2. Press the menu forward key and then the "Service planner" softkey. The window with the list of all the maintenance tasks that have been set up appears.



3. Perform the maintenance task when the maintenance interval has nearly expired or when prompted to do so by alarms or a warning.



4. After you have performed a pending maintenance task and the task is signaled as "Completed", position the cursor at the appropriate task and press the "Servicing performed" softkey. A message is displayed confirming the acknowledgment, and the maintenance interval is restarted.

Note

You can perform the maintenance tasks before the interval expires. The maintenance interval is restarted.

23.2 Set maintenance tasks

You can make the following changes in the list of maintenance tasks in the configuration mode:

- Set up a maximum of 32 maintenance tasks with interval, initial warning and number of warnings to be acknowledged
- Change the interval, time of the initial warning and the number of warnings to be output
- Delete a maintenance task
- Reset the times of the maintenance tasks

Acknowledging a maintenance task




You can acknowledge the maintenance tasks with the "Servicing performed" softkey.

Note

Protection level

You require protection level 1 (manufacturer) to set up and edit maintenance tasks.

Service Planner

Display	Meaning	
Pos	Position of the maintenance task in the PLC interface.	
Maintenance task	Name of the maintenance task.	
Interval [h]	Maximum time until next servicing in hours.	
1. warning [h]	Time in hours at which an initial warning is displayed.	
Number of warnings	Number of warnings that can be acknowledged by the operator before an alarm message is output for the last time.	
Remaining time [h]	Time until the interval expires in hours. The remaining time cannot be edited.	
Status	  	Display of the current status of a maintenance task. The maintenance task has been started. The maintenance task is completed. The maintenance task is deactivated, i.e. the time has been stopped.
	The status cannot be edited.	

Procedure



1. Select the "Diagnostics" operating area.



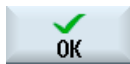
2. Press the menu forward key and then the "Service planner" softkey.
The window opens and displays a list of all the tasks that have been set up.



The values cannot be edited.



3. Press the "New maintenance task" softkey to set up a new maintenance task.



A message informs you that a new maintenance task will be set up at the next free position. Enter the required data in the columns and press the "OK" softkey.

- OR -



Position the cursor on the desired maintenance task and press the "Change task" softkey to change the associated times.

All columns except Remaining time and Status can be edited.

- OR -



Press the "Reset all" softkey to reset all the times.

- OR -



Position the cursor on the desired maintenance task and press the "Delete task" softkey to remove the maintenance task from the list.

Ladder Viewer and Ladder add-on (828D only)

24.1 PLC diagnostics

A PLC user program consists to a large degree of logical operations to implement safety functions and to support process sequences. These logical operations include the linking of various contacts and relays. These logic operations are displayed in a ladder diagram.

Ladder add-on tool

As a rule, the failure of a single contact or relay results in a failure of the whole system.

Using the Ladder add-on tool, you can perform a PLC diagnosis in order to find fault causes or program errors.

Editing of interrupt routines

You can edit the following interrupt programs:

- INT_100 - interrupt program, (is executed before the main program)
- INT_101 - interrupt program, (is executed after the main program)

Marshalling data

Using the Ladder add-on tool, you can "re-wire" inputs (via INT_100) or outputs (via INT_101) for service purposes.

Generating INT_100 / INT_101 block

If one or several INT_100 or INT_101 blocks are missing, they can be added via the vertical softkey bar. If these INT blocks exist in a project, they can be deleted via the vertical softkey bar. You also have the opportunity to change the networks of a program on the control as well as to save and load these changes.

Note

Saving the PLC project when changing the operating area

If you have created INT_100/INT_101 blocks or inserted, removed or edited networks in an INT block, you must save the project before you change from the PLC area into another operating area. Transfer the project into the PLC using the "Download to CPU" softkey. If this is not done, all of the changes will be lost and must be re-entered.

24.2 Structure of the user interface

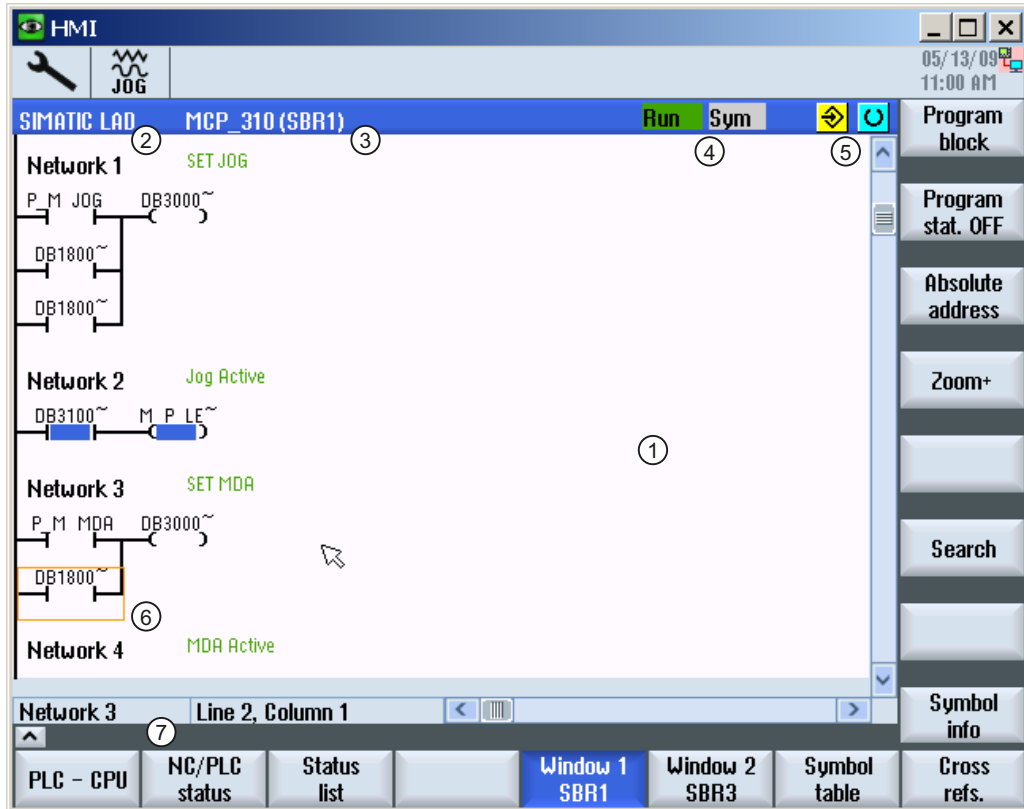




Figure 24-1 Screen structure

Table 24- 1 Key to screen layout













Screen element	Display	Meaning
1	Application area	
2	Supported PLC program language	
3	Name of the active program block Representation: Symbolic name (absolute name)	
4	Program status	
		
	Run	Program is running
	Stop	Program is stopped
	Status of the application area	
	Sym	Symbolic representation
Abs	Absolute representation	
5	Display of the active keys (<INPUT>, <SELECT>)	
		
6	Focus Performs the tasks of the cursor	
7	Information line Displays information, e.g. for searching	









24.3 Control options

In addition to the softkeys and the navigation keys, there are further shortcuts in this area.

Shortcuts

The cursor keys move the focus over the PLC user program. When the window borders are reached, scrolling is performed automatically.

Shortcuts	Action
	To the first column of the row
CTRL 	
END	To the last column of the row
CTRL 	
	Up a screen
	Down a screen
 	One field to the left, right, up or down
 	
CTRL 	To the first field of the first network
-or-	
CTRL 	
CTRL 	To the last field of the last network
-or-	

Shortcuts		Action
		
		Open the next program block in the same window
		Open the previous program block in the same window
		<p>The function of the Select key depends on the position of the input focus.</p> <ul style="list-style-type: none"> • Table line: Displays the complete text line • Network title: Displays the network comment • Command: Displays all operands
		If the input focus is positioned on a command, all operands including the comments are displayed.

24.4 Displaying PLC properties

The following PLC properties can be displayed in the "SIMATIC LAD" window:

- Operating state
- Name of the PLC project
- PLC system version
- Cycle time
- Machining time of the PLC user program

Procedure



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



2. Press the "PLC" softkey.
The ladder diagram display opens and displays the PLC information.



3. Press the "Reset mach.time" softkey.
The data of the machining time is reset.

24.5 Displaying information on the program blocks

You can display all the logic and graphic information of a program block.

Display program block

In the "Program block" list, select the program block that you want to display.

Logic information

The following logic information is displayed in a ladder diagram (LAD):

- Networks with program parts and current paths
- Electrical current flow through a number of logical operations

Further information

- Properties

Name of the block, author, number of the subprogram, data class, date it was generated, date of the last change and comment.

- Local variable

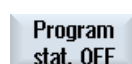
Name of the variable, variable type, data type and comment.

Access protection

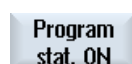


If a program block is password-protected. The ladder diagram display can be enabled using the "Protection" softkey.

Displaying the program status



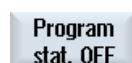
1. Press the "Program stat. OFF" softkey to hide the program status in the status display.



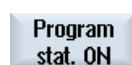
2. Press the "Program stat. ON" softkey to unhide the program status in the status display.

Display progress status

If your PLC has the function "Progress status" available, the status values are displayed at the time of execution of the operations. The status of the local data memory and the accumulators is also displayed.



1. Press the "Program stat. OFF" softkey to hide the program status in the status display.



2. Press the "Program stat. ON" softkey to unhide the program status in the status display.

Change colors for displaying of progress or program status

In progress status, different colors are used to display information.

Display	Color
Signal flow of power rail, when status active	Blue
Signal flow in the networks	Blue
All operations that are active and that are executed without error (corresponds to signal flow)	Blue
Status of the Boolean operations (corresponds to signal flow)	Blue
Times and counters active	Green
Error during execution	Red
No signal flow	Gray
No network executed	Gray
STOP mode	Gray

Note

Colored display in program status

In the display of the program status only the color of the signal flow is relevant.

Enlarging/reducing the display of the ladder diagram



1. Press the "Zoom +" softkey to enlarge the section of the ladder diagram.

After enlarging, the "Zoom -" softkey is available.



2. Press the "Zoom -" softkey to reduce the section of the ladder diagram again.

Procedure



1. Ladder add-on tool is opened.



2. Press the "Window 1" or "Window 2" softkey.

...



3. Press the "Program block" softkey.
The "Program block" list is displayed.



4. Press the "Properties" softkey if you wish to display additional information.

- OR -



Press the "Local variables" softkey if you wish to display data of a variable.

24.6 Displaying and editing NC/PLC variables

The "NC/PLC Variables" window enables the monitoring and modification of NC system variables and PLC variables.

You receive the following list in which you enter the desired NC and PLC variables in order to display the actual values.

- Variable
Address for NC/PLC variable.
Faulty variables have a red background and # is displayed in the Value column.
- Comment
Arbitrary comment on the variables.
The column can be shown or hidden.
- Format
Specification of the format in which the variable is to be displayed.
The format can be permanently defined (e.g. floating-point).
- Value
Display of the current value of the NC/PLC variable.

Procedure



1. Ladder add-on tool is opened.



2. Press the "NC/PLC variables" softkey.
The "NC/PLC Variables" window appears.

24.7 Downloading a PLC user program

Download the project data into the PLC if some changes have been made to the project data and a new PLC user program is available.

When the project data is loaded, the data classes are saved and loaded to the PLC.

Requirement

Check whether the PLC is in Stop mode.

Note





PLC in RUN mode

If the PLC is in RUN mode, a corresponding message is displayed and the "Load in Stop" and "Load in Run" softkeys appear.

With "Load in Stop", the PLC is set to Stop mode and the project is stored and loaded to the CPU.

With "Load in Run", the loading operation is continued and the PLC project loaded to the PLC. Only those data classes that have really been changed are loaded, i.e. generally INDIVIDUAL data classes.

Procedure

- | | |
|---|---|
|  | 1. Ladder add-on tool is opened.
You have changed project data. |
|  | 2. Press the "PLC Stop" softkey if the PLC is in the run mode. |
|  | 3. Press the "Load to CPU" softkey to start the loading operation.
All data classes are loaded. |
|  | 4. When the PLC project has been loaded, press the "PLC Start" softkey to switch the PLC to Run mode. |

24.8 Editing the local variable table

You have the option of editing the local variable table of an INT block.

Insert local variable

If you have inserted new networks or operands, it may be necessary to insert new variables in the local variable table of an INT block.

Name	Freely assign.
Variable type	Selection: <ul style="list-style-type: none">• IN• IN_OUT• OUT• TEMP
Data type	Selection: <ul style="list-style-type: none">• BOOL• BYTE• WORD• INT• DWORD• DINT• REAL
Comment	Freely assign.

Procedure



...



1. The ladder diagram display (LAD) is opened.
2. Press the "Program block" softkey.
3. Press the "Local variables" softkey.
The "Local Variables" window appears and lists the created variables.
4. Press the "Edit" softkey.
The fields can be edited.
5. Enter a name, select the variable and data type and, if required, enter a comment.
6. Press the "Attach line" softkey if you want to add a further variable and enter the data.

- OR -
Select the relevant variable and press the "Delete line" softkey to remove the variable from the list.

24.9 Creating a new block

Create INT blocks if you wish to make changes with the PLC user program.

Name	INT _100, INT_101
	The number from the selection field "Number, interrupt program" is taken for the name of the INT block.
Author	A maximum of 48 characters is permitted.
Number of the interrupt program	100 101
Data class	Individual
Comment	A maximum of 100 lines and 4096 characters are permitted.

Note

Access protection

You have the option of protecting blocks that have been newly created against being accessed.

Procedure



...



1. The ladder diagram display (LAD) is opened.
2. Press the "Program block" softkey to open the list of program blocks.
3. Press the "Add" softkey.
The "Properties" window appears.
4. Enter the name of the author, number of the INT block and, if required, a comment.
The data class of the block is specified.
5. Press the "OK" softkey to transfer the block to the list.

24.10 Editing block properties

You can edit the title, author and comments of an INT block.

Note

You cannot edit the block name, interrupt number and data class assignment.

Procedure



1. The ladder diagram display is opened.



2. Select the relevant block and press the "Program block" softkey.



3. Press the "Properties" softkey.
The "Properties" window appears.

24.11 Inserting and editing networks

You can create a new network and then insert operations (bit operation, assignment, etc.) at the selected cursor position.

Only empty networks can be edited. Networks, that already include statements, can only be deleted.

A simple, single line can be edited for each network. You can create a maximum of 3 columns per network.

Column	Operation	
Column 1	<ul style="list-style-type: none"> • NO contact • NC contact 	- - - / -
Column 2 (optional)	NOT Rising edge Falling edge Assign Set Reset	- NOT - - P - - N - -() -(S) -(R)
Column 3 (only possible if no assign, set or reset operations were specified in the second column)	Assign Set Reset	-() -(S) -(R)

Note

Logical AND (serial contact) and logical OR (parallel contact) are not possible.

The bit combinations comprise one or several logical operations and the assignment to an output / bit memory.

If the cursor is moved further to the left with the arrow key, the type of assignment or a logic operation can be selected. A further logic operation cannot be placed to the right of an assignment. A network must always be terminated with an assignment.

References

For further information about PLC programming, please refer to:

Function Manual Basic Functions, PLC for SINUMERIK 828D (P4)

Procedure



1. An interrupt routine has been selected.

2. Press the "Edit" softkey.

3. Position the cursor on a network.

4. Press the "Insert network" softkey.

- OR -

Press the <INSERT> key.

If the cursor is positioned on "Network x", a new, empty network is inserted behind this network.

5. Position the cursor on the desired element below the network title and press the "Insert operation" softkey.

The "Insert Operation" window appears.

6. Select the desired bit operation (NC contact or NO contact) or assignment and press the "OK" softkey.

7. Press the "Insert operand" softkey.

8. Enter the logic operation or the command and press the <INPUT> key to complete the entry.

9. Position the cursor on the operation that you want to delete and press the "Delete operation" softkey.

- OR -

Position the cursor on the title of the network that you want to delete and press the "Delete network" softkey.

- OR -

Press the key.

The network, including all the logic operations and operands, or the selected operation is deleted.

24.12 Editing network properties

You can edit the network properties of an INT block.

Network title and network comment

The title can have a maximum of three lines and 128 characters. The comment can have a maximum of 100 lines and 4096 characters.

Procedure



1. The ladder diagram display (LAD) is opened.



2. Use the cursor keys to select the network that you want to edit.



3. Press the <SELECT> key.
The "Network title / comment" window opens and shows the title and a possibly assigned comment for the selected network.



5. Press the "Change" softkey.
The fields can be edited.








6. Enter the changes and press the "OK" softkey to transfer the data to the user program.

24.13 Displaying/canceling the access protection

You can password protect your program organizational units (POUs) in the PLC 828 programming tool. This prevents other users from accessing this part of the program. This means that it is invisible to other users and is encrypted when it is downloaded.

A lock symbol is used to show a password-protected POU in the block overview and in the ladder diagram.

Procedure

-  1. The ladder diagram display (LAD) is opened.
-  2. Select the relevant program organizational unit (POU) in the overview and press the "Program block" softkey.
-  3. Press the "Protection" softkey.
 The "Protection" window appears.
-  4. Enter your password and press the "Accept" softkey.

24.14 Displaying and editing symbol tables

You can display the symbol tables that are used to obtain an overview of the global operands available in the project - which you can then edit.

The name, address and possibly also a comment is displayed for each entry.

Procedure



1. Ladder add-on tool is opened.



2. Press the "Symbol table" softkey.
The list with the symbol table entries is displayed.



3. Press the "Edit" softkey if you want to change entries.
The display fields can be edited.



4. Use the cursor keys to select the desired entry and the field to be changed.



5. Enter the value to be changed.

- OR -



Press the "Attach line" softkey to insert an empty line after the selected entry.

- OR -



Press the "Delete line" softkey to remove the selected entry from the list.

- OR -

Enter a new value in the selected field.



7. Press the "OK" softkey to confirm your action.

24.15 Searching for operands

You can use the search function to quickly reach points in very large programs where you would like, for example, to make changes.

Restricting the search

- "Window 1" / "Window 2", "Symbol table"

With "Go to", you can jump directly to the desired network.








- "Cross references"

With "Go to", you can jump directly to the desired line.

Requirement

Window 1 / window 2, the symbol tables or the list of cross references is opened.

Procedure

- | | |
|---|---|
|  | 1. Press the "Find" softkey.
A new vertical softkey bar appears. The "Find / Go To" window opens at the same time. |
|  | 2. Select the "Find operand" entry in the first input field if you are searching for a specific operand and enter the search term in the "Find" input field. |
|  | 3. Select the search range (e.g. Find all). |
|  | 4. Select the "In this program unit" or "In all program units" entry if you are in "Window 1" or "Window 2" or in the symbol table in order to restrict the search. |
|  | 5. Press the "OK" softkey to start the search.
If the operand you are searching for is found, the corresponding line is highlighted. |
|  | Press the "Continue search" softkey if the operand found during the search does not correspond to the element you are looking for. |
| - OR - | |
|  | Press the "Cancel" softkey if you want to cancel the search. |

Further search options



1. Press the "Go to start" softkey to jump to the start of the ladder diagram in window 1 or window 2, or the list (cross references, symbol table).



2. Press the "Go to end" softkey to jump to the end of the ladder diagram in window 1 or window 2, or the list (cross references, symbol table).

24.16 Inserting/deleting a symbol table

New user symbol tables can be generated and changed. Tables that are no longer used can be deleted.

Note

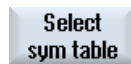
Delete symbol table

The "Delete" softkey is only available if a user symbol table has been selected.

Procedure



1. The symbol table is opened.



2. Press the "Select sym. table" softkey.
The "Symbol Table - Selection" window appears.



3. Position the cursor at the desired position and press the "Insert sym. table" softkey.

The "Create Symbol Table" window appears.



4. Enter a symbolic name and press the "OK" softkey.
The new user symbol table is inserted in the line after the cursor position.

- OR -



Select a symbol table and press the "Change symbol table" softkey if you wish to change the properties of the symbol table.



5. Position the cursor on the symbol table that you want to delete and press the "Delete" softkey.

24.17 Displaying the network symbol information table

All of the symbolic identifiers used in the selected network are displayed in the "Network symbol information table" window.

The following information is listed:

- Names
- Absolute addresses
- Comments

The symbol information table remains empty for networks that do not contain any global symbols.

Procedure



1. The ladder diagram display (LAD) is opened.



2. Select the desired network and press the "Symbol info" softkey. The "Network Symbol Information Table" window appears.
3. Use the cursor keys to move within the table.



24.18 Displaying and editing PLC signals

PLC signals are displayed and can be changed here in the "PLC status list" window.

The following lists are shown

Inputs (IB)

Bit memories (MB)

Outputs (QB)

Variables (VB)

Data (DB)







Setting the address

You can go directly to the desired PLC address to monitor the signals.

Editing

You can edit the data.

Procedure

- | | |
|---|---|
|  | 1. Ladder add-on tool is opened. |
|  | 2. Press the "Status list" softkey.
The "Status List" window appears. |
|  | 3. Press the "Set address" softkey.
The "Set Address" window appears. |
|  | 4. Activate the desired address type (e.g. DB), enter the value and press the "OK" softkey.
The cursor jumps to the specified address. |
|  | 5. Press the "Change" softkey.
The "RW" input field can be edited. |
|  | 6. Enter the desired value and press the "OK" softkey. |

24.19 Displaying cross references

You can display all the operands used in the PLC user project and their use in the list of cross references.

This list indicates in which networks an input, output, bit memory etc. is used.

The list of cross references contains the following information:

- Block
- Address in the network
- Context (command ID)

Symbolic and absolute address

You can choose between specification in absolute or symbolic address.

Elements for which there are no symbolic identifiers are automatically displayed with absolute identifiers.

Opening program blocks in the ladder diagram











From the cross references, you have the option of going directly to the location in the program where the operand is used. The corresponding block is opened in window 1 or 2 and the cursor is set to the corresponding element.

Searching

Using a specific search, you can go directly to the location that you wish to view in more detail:

- Search for operand
- Jump to sought line

Procedure


- | | | |
|---|----|---|
|  | 1. | Ladder add-on tool is opened. |
|  | 2. | Press the "Cross refs." softkey.
The list of cross references appears and the operands are displayed sorted according to absolute address. |
|  | 3. | Press the "Symbol. address" softkey.
The list of operands is displayed sorted according to symbolic address. |
|  | 4. | Press the "Absolute address" softkey to return to the display showing the absolute addresses. |
|  | 5. | Select the desired cross reference and press the "Open in window 1" or "Open in window 2" softkey.
The ladder diagram is opened, and the selected operand is marked. |
|  | | |
|  | 6. | Press the "Find" softkey.
The "Find / Go To" window appears. |
|  | 7. | Select "Find operand" or "Go to" and enter the sought element or the desired line and select the search order (e.g. search up). |
|  | 8. | Press the "OK" softkey to start the search. |
|  | 9. | If an element is found that corresponds to the sought element, but is not at the appropriate position, press the "Continue search" softkey to find where the search term occurs next. |

Alarm, error and system messages

25.1 Displaying alarms

If faulty conditions are recognized in the operation of the machine, then an alarm will be generated and, if necessary, the machining will be interrupted.

The error text that is displayed together with the alarm number gives you more detailed information on the error cause.

 WARNING
<p>Please check the situation in the plant on the basis of the description of the active alarm(s). Eliminate the cause/s of the alarm/s and acknowledge it/them as instructed.</p> <p>Failure to observe this warning will place your machine, workpiece, stored settings and possibly even your own safety at risk.</p>

Alarm overview

You can display all upcoming alarms and acknowledge them.

The alarm overview contains the following information:

- Date and time
- Cancel criterion
 - specifies the key or softkey 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.

25.1 Displaying alarms



4. Position the cursor on an alarm.

...



5. Press the key that is specified as acknowledgement symbol to delete the alarm.

- OR -



Press the "Delete HMI alarm" softkey to cancel an HMI alarm.






- OR -



Press the "Acknowledge alarm" softkey to delete a PLC alarm of the SQ type (alarm number as of 800000).

The softkeys are activated when the cursor is on the corresponding alarm.

Acknowledgement symbols

Symbol	Meaning
	Turn the unit off and back on (main switch), or press NCK POWER ON.
	Press the <RESET> key.
 ...	Press the <ALARM CANCEL> key. - OR - Press the "Acknowl. HMI alarm" softkey.
	
	Press the key provided by the manufacturer.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

25.2 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 card/user/sinumerik/hmi/log/alarm_log.

25.3 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.

25.4 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 softkey "Alarm list", "Messages" or "Alarm log" to display the requested messages and alarms.

...



3. Press the "Sort" softkey.



The list of entries is sorted in ascending order according to date, i.e. the most recent information is at the end of the list.



4. Press the softkey "Descending" to sort the list in a descending order. The most recent event is shown at the beginning 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 "Ascending" softkey if you wish to display the list in an ascending order.

25.5 PLC and NC variables

25.5.1 Displaying and editing PLC and NC variables

The "NC/PLC Variables" window allows NC system variables and PLC variables to be monitored and changed.

You receive the following list in which you can enter the desired NC/PLC variables in order to display the actual values.

- 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)
Outputs	Output bit (Ax), output byte (ABx), output word (AWx), output double word (ADx)
Bit memory	Memory bit (Mx), memory byte (MBx), memory word (MWx), memory double word (MDx)
Times	Time (Tx)
Counters	Counter (Cx)
Data	Data block (DBx): Data bit (DBXx), data byte (DBBx), data word (DBWx), data double word (DBDx)

Formats	
B	Binary
H	Hexadecimal
D	Decimal without sign
+/-D	Decimal with sign
F	Floating point (for double words)
A	ASCII character

Notation for variables

- PLC variables
 - EB2
 - A1.2
 - DB2.DBW2
- NC variables
 - NC system variables - notation
 - \$AA_IM[1]
 - User variables/GUDs - notation
 - GUD/MyVariable[1,3]
 - OPI - notation
 - /CHANNEL/PARAMETER/R[u1,2]


Note

NC system variables and PLC variables

- System variables can be dependent on the channel. When the channel is switched over, the values from the corresponding channel are displayed.
 - For user variables (GUDs) it is not necessary to make a specification according to global or channel-specific GUDs. The indices of GUD arrays are, just like NC variables in the system variable syntax, 0-based; this means that the first element starts with the index 0.
 - Using the tooltip, for NC system variables, you can display the OPI notation (with the exception of GUDs).
-

Changing PLC variables

Changes can only be made to the PLC variables with the appropriate password.

 DANGER
Changes in the states of NC/PLC variables have a major impact on the machine. Incorrect configuration of the parameters can endanger human life and cause damage to the machine.

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" are displayed in the full length.



6. Position the cursor in the "Format" field and choose 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 again.



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 of the existing variables and insert this.

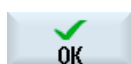
The "Select Variable" window opens.



10. Press the "Filter/search" softkey to restrict the display of variables (e.g. to mode groups-variables) using the "Filter" selection box and/or select the desired variable using the "Search" input box.



Press the "Delete all" softkey if you would like to delete the entries for the operands.



11. Press the "OK" softkey to confirm the changes or the deletion.

- OR -



Press the "Cancel" softkey to cancel the changes.

Note

"Filter/Search" when inserting variables

The start value for "Filter/Search" of variables differs.

For example, to insert the variable \$R[0], set "Filter/Search":

- 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.
-

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 softkeys "Operand +" and "Operand -" do not act as index, e.g. for \$AA_IM[X1].



Examples

DB97.DBX2.5
Result: DB97.DBX2.6

\$AA_IM[1]
Result: \$AA_IM[2]



MB201
Result: MB200
/Channel/Parameter/R[u1,3]
Result: /Channel/Parameter/R[u1,2]

25.5.2 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, then 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.

25.5.3 Load symbols

PLC data can also be edited via symbols.

To do this, the symbol tables and texts for the symbols in the PLC project must have been suitably prepared (STEP7) and made available in SINUMERIK Operate.

Preparing PLC data

Save the generated files in the `/oem/sinumerik/plc/symbols` directory.

Procedure



1. The variable view is opened.



2. Press the ">>" and "Load icons" softkeys.
The "Import PLC symbols: *.snh" window opens.



3. In the folder `/oem/sinumerik/plc/symbols`, select the `PlcSym.snh` file to import the symbols and then click on "OK".



4. In the folder `/oem/sinumerik/plc/symbols`, select the `PlcSym.snt` file to import the symbols and then press the "OK" softkey.



You will obtain an appropriate note if the tables were successfully imported.



5. Press the "OK" softkey.
You return to the "NC/PLC variables" window.
6. Restart the SINUMERIK Operate in order to activate the files.

25.6 Version

25.6.1 Displaying version data

The following components with the associated version data are specified in the "Version data" window:

- System software
- Basic PLC program
- PLC user program
- System extensions
- 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 displays 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 "Diagnosis" operating area.



2. Press the "Version" softkey.
The "Version Data" window appears.
Data from the available components are 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.

25.6.2 Save information

All the machine-specific information of the controller is combined in a configuration via the user interface. You can save 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. The following options are available:

- In the "Name:" text 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.
- In text field "Comment", you can add a comment, which is stored with the configuration data.

Select the following via a checkbox:

- Version data (.TXT): Output of pure version data in text format.
- Configuration data (.XML): Output of configuration data in XML format. The configuration file contains the data you entered under Machine identity, the license requirements, the version information and the logbook entries.



7. Press the "OK" softkey to start the data transfer.

25.7 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")

Output of the logbook

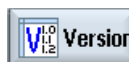
You have the possibility of exporting the logbook by generating a file using the "Save version" function in which the logbook is contained as section.

25.7.1 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



You have the option of changing the address data of the end customer using the "Change" softkey.

25.7.2 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

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

Once you have stored an entry this can no longer be changed or deleted.

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 "Search..." softkey and 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.

3. 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.

25.8 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`

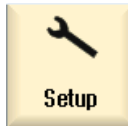
Procedure

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 run in ascending order from "SCR_SAVE_0001.png" to "SCR_SAVE_9999". You can create up to 9,999 screenshots.

Copy file



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



2. Press the "System data" softkey and open the specified folder.

As you cannot open screenshots in SINUMERIK Operate, you must copy the files to a Windows PC either via "WinSCP" or via a USB-FlashDrive.

You can open the files using a graphics program, e.g. "Office Picture Manager".

25.9 Remote diagnostics

25.9.1 Setting remote access

You can influence the remote access to your control in the "Remote diagnostics (RCS)" window.

Here, rights for all types of remote control are set. 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, then this is still dependent on a manual or automatic confirmation.

Rights for remote access

The "Specified from PLC" field shows the access rights for remote access or remote monitoring specified from the PLC.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

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

The combination of the settings in the HMI and in the PLC show the valid status as to whether access is permitted or not. This is displayed in the "Resulting from" line.

Settings for the confirmation dialog box

If the settings made for "Specified from the PLC" and "Selected in the HMI" permit access from outside, then this is however, still dependent on either a 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.

For the case that there is no local operation, then the control behavior can be set for this particular scenario. You define how long this window is displayed and whether, after the confirmation has expired, the remote access is automatically rejected or accepted.

Display of the state



Remote monitoring active



Remote control active

If remote access is active, using these icons you will be informed in the status line as to whether a remote access is presently 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 is opened.



3. Press the "Change" softkey.
The "Selected in the HMI" is activated.



4. If you desire remote control, select the entry "Permit remote control".

In order that remote control is possible, the entry "Permit remote control" must be specified in the fields "Specified from the PLC" and "Selected in the HMI".

5. Enter new values in the group "Behavior for confirming remote access" if you wish to change the behavior for confirming remote access.



6. Press the "OK" softkey.
The settings are accepted and saved.

References

For a description of configuration options, refer to the
Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

25.9.2 Permit modem

You can permit remote access to your control via a teleservice adapter IE connected at X127.



Machine manufacturer

Please refer to the machine manufacturer's specifications.



Software option

You need the "MC Information System RCS Host" option to display the "Permit modem" softkey.

Procedure



1. The "Remote diagnostics (RCS)" window is opened.



2. Press the "Permit modem" softkey.

Access to the control via modem is enabled so that a connection can be established.



3. To block access again, press the "Permit modem" softkey again.

25.9.3 Request remote diagnostics

Using the "Request remote diagnostics" softkey, from your control you have the option of actively requesting remote diagnostics with your machinery construction OEM.

The access via modem must be enabled if the access is to be made via modem.



Machine manufacturer

Please refer to the machine manufacturer's specifications.

When requesting remote diagnostics, you obtain a window with the corresponding 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



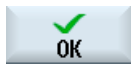
1. The "Remote diagnostics (RCS)" window is opened.



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.

References

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

25.9.4 Exit remote diagnostics

Procedure



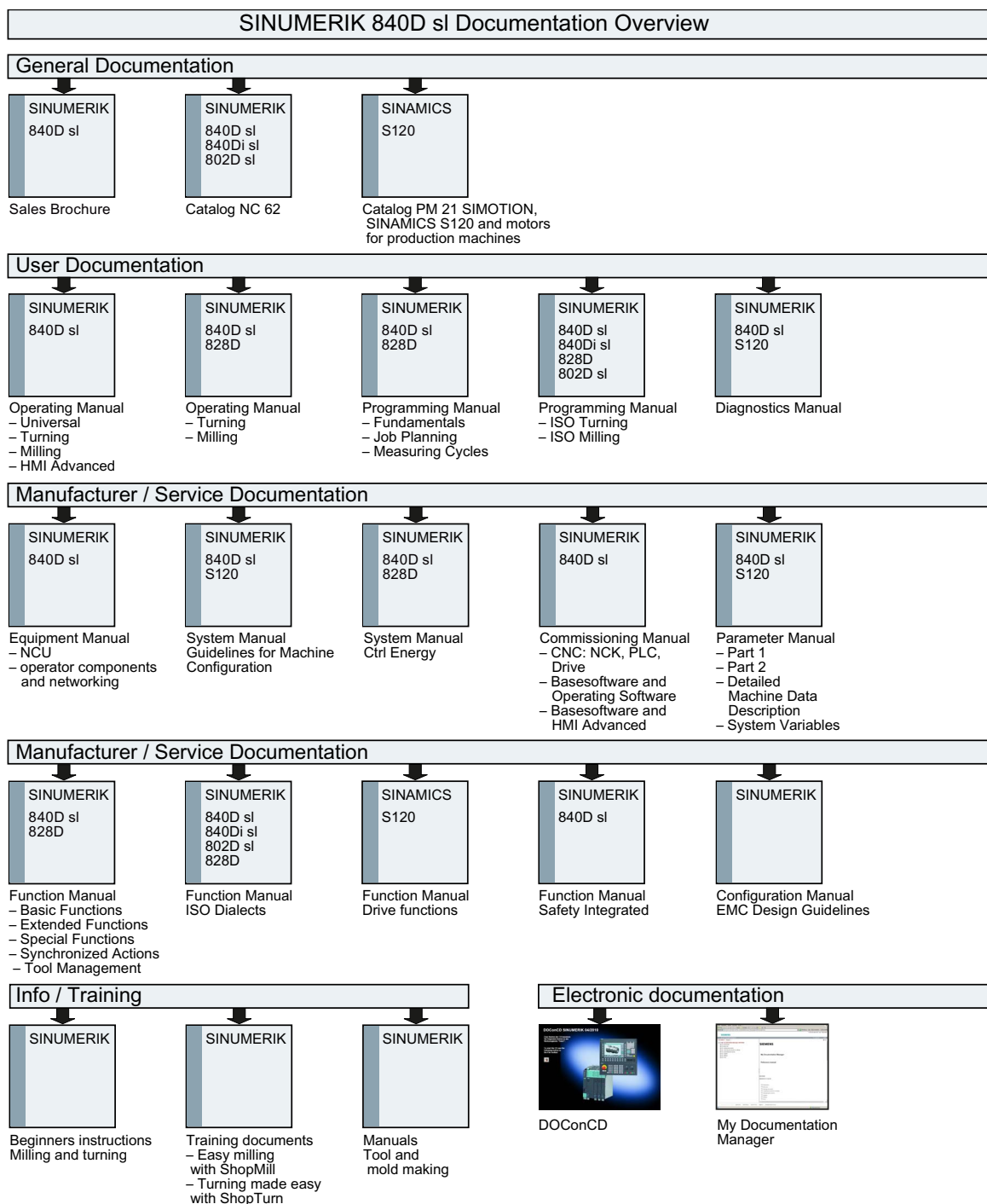
1. The "Remote diagnostics (RCS)" is opened and it is possible that remote monitoring or remote access is active.

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".

Appendix

A.1 Overview



Index

A

- Absolute dimensions, 231
- Actual-value display, 44
- Adapter-transformed view, 679
- Alarm log
 - display, 815
 - Sorting, 817
- Alarms
 - Displaying, 813
 - Sorting, 817
- Align milling tool - CYCLE800
 - Function, 485
 - Parameters, 487
- Alignment angle
 - B axis, 622
- Any file
 - Creating, 707
- Approach/retract cycle
 - Parameter, 259
- Approach/retraction cycle
 - ShopTurn program, 258
- Arbitrary positions - CYCLE802
 - Function, 311
- Archive
 - Generate in the program manager, 727
 - generating in the system data, 728
 - read in archive from system data, 732
 - reading-in in the Program Manager, 731
- Asian characters, 54
- Auxiliary functions
 - H functions, 169
 - M functions, 169
- Axes
 - Defined increment, 111
 - Direct positioning, 113
 - Referencing, 64
 - Repositioning, 133
 - Traversing, 110
 - Variable increment, 112

B

- B axis
 - Alignment angle, 622

- Approach/retraction, 626
 - Function, 619
 - Measure tool, 632
 - Milling, 623
 - Position pattern, 628
 - Swiveling, 624
- Backing up
 - Data, 727, 728
 - Setup data, 734
- Base offset, 84
- Basic blocks, 128
- Binary format, 727
- Blank input
 - Function, 208
 - Parameter, 209, 545
- Block
 - Search, 100, 138
 - Searching, 135, 139
- Block search
 - Mode, 140
 - Multi-channel view, 565
 - Program interruption, 100, 138
 - Search pointer, 100, 135, 138, 139
 - Search target parameters, 140
 - Search target specification, 137
 - Using, 135
- Boring - CYCLE86
 - Function, 293
 - Parameter, 296

C

- Calculator, 51
 - Fits, 363
- Centering - CYCLE81
 - Function, 286
 - Parameter, 288
- Channel switchover, 69
- Chuck dimensions, 96
- Circle, 506
 - Manual Machine, 610
- Circle position pattern - HOLES2
 - Function, 317
- Circular pocket - POCKET4
 - Function, 394
 - Parameter, 400
- Circular position pattern - HOLES2
 - Parameter, 319

- Circular spigot - CYCLE77
 - Function, 404
 - Parameter, 408
- Circumferential groove - SLOT2
 - Function, 416
- Circumferential slot - SLOT2
 - Parameter, 420
- Clamping the spindle, 234
- Context-sensitive online help, 59
- Continuous-path mode, 592
- Contour call - CYCLE62
 - Function, 365, 452
 - Parameter, 366, 453
- Contour elements
 - Representation, 352, 442
- Contour milling - CYCLE63
 - Function, contour pocket, 465
 - Function, contour spigot, 471
 - Function, residual material contour pocket, 468
 - Function, residual material contour spigot, 474
 - Parameter, contour spigot, 473
 - Parameters, contour pocket, 468
 - Parameters, residual material contour pocket, 470
 - Parameters, residual material contour spigot, 476
- Contour milling - CYCLE64
 - Function, centering, 460
 - Function, predrilling, 460
 - Parameters, centering, 463
 - Parameters, predrilling, 464
- Contour turning
 - Overview, 351
- Contour turning - CYCLE952
 - Function, 366
 - Function, grooving, 374
 - Function, grooving residual material, 378
 - Function, plunge turning, 380
 - Function, plunge turning residual material, 384
 - Function, stock removal residual, 372
 - Parameters, grooving, 377
 - Parameters, grooving residual material, 380
 - Parameters, plunge turning, 383
 - Parameters, plunge turning residual material, 385
 - Parameters, stock removal, 371
 - Parameters, stock removal residual material, 374
- Contours
 - Programming, 351
- Coordinate system
 - Switching over, 71
- Coordinate transformation, 500
- Copying
 - Directory, 717
 - Program, 717
- Counterspindle, 96
- Create
 - Program block, 155
- Creating
 - Any file, 707
 - Directory, 703
 - G code program, 705
 - Job list, 708
 - Multi-channel program, 538
 - Multitool, 683
 - NC directory on the local drive, 697
 - Program list, 710
 - Workpiece, 704
- Ctrl-Energy
 - Energy analysis, 760
 - Energy-saving profiles, 765
 - Functions, 759
 - Measuring the energy consumption, 761
 - Saved measuring curves, 764
- Cut-off - CYCLE92
 - Function, 349
 - Parameter, 350
- Cutting edge numbers, 650
- Cutting edge width, 649
- Cutting tip length, 649
- Cutting tip width, 649
- CYCLE61- face milling
 - Function, 386
 - Parameter, 389
- CYCLE62 - new contour
 - Function, 354, 444
 - Parameter, 446
- CYCLE62- contour call
 - Function, 365, 452
 - Parameter, 366, 453
- CYCLE63 - contour milling
 - Function, contour pocket, 465
 - Function, contour spigot, 471
 - Function, residual material contour pocket, 468
 - Function, residual material contour spigot, 474
 - Parameter, contour spigot, 473
 - Parameters, contour pocket, 468
 - Parameters, residual material contour pocket, 470
 - Parameters, residual material contour spigot, 476
- CYCLE64 - contour milling
 - Function, centering, 460
 - Function, predrilling, 460
 - Parameters, centering, 463
 - Parameters, predrilling, 464
- CYCLE70 - engraving
 - Function, 434
 - Parameter, 441

- CYCLE70 - thread milling
 Function, 430
 Parameters, 432
- CYCLE72 - Path milling
 Function, 453
 Parameter, 458
- CYCLE76 - rectangular spigot
 Function, 400
 Parameter, 404
- CYCLE77 - circular spigot
 Function, 404
 Parameter, 408
- CYCLE78 - Drilling and thread milling
 Function, 306
 Parameter, 309
- CYCLE79 - multi-edge
 Function, 408
 Parameter, 411
- CYCLE801 - grid/frame position pattern
 Function, 314
 Parameter, 316
- CYCLE802 - arbitrary positions
 Function, 311
- CYCLE802 - freely programmable positions
 Parameter, 313
- CYCLE81 - centering
 Function, 286
 Parameter, 288
- CYCLE82 - drilling
 Function, 289
 Parameter, 291
- CYCLE83 - deep-hole drilling
 Function, 296
 Parameter, 300
- CYCLE832 - High Speed Settings
 Function, 490
 Parameters, 492
- CYCLE84 - tapping without compensating chuck
 Function, 300
 Parameter, 305
- CYCLE840 - tapping with compensating chuck
 Function, 300
 Parameter, 305
- CYCLE85 - reaming
 Function, 291
 Parameter, 293
- CYCLE86 - boring
 Function, 293
 Parameter, 296
- CYCLE899 - open slot
 Function, 420
 Parameter, 427
- CYCLE92 - cut-off
 Function, 349
 Parameter, 350
- CYCLE930 - groove
 Function, 327
 Parameter, 329
- CYCLE940 - undercut
 Function, DIN thread, 333
 Function, form E, 330
 Function, form F, 330
 Function, thread, 333
 Parameters, DIN thread, 334
 Parameters, form E, 331
 Parameters, form F, 332
 Parameters, thread, 335
- CYCLE951 - stock removal
 Function, 323
 Parameter, 326
- CYCLE952 - contour turning
 Function, 366
 Function, grooving, 374
 Function, grooving residual material, 378
 Function, plunge turning, 380
 Function, plunge turning residual material, 384
 Function, stock removal residual, 372
 Parameters, grooving, 377
 Parameters, grooving residual material, 380
 Parameters, plunge turning, 383
 Parameters, plunge turning residual material, 385
 Parameters, stock removal, 371
 Parameters, stock removal residual material, 374
- CYCLE98 - thread chain
 Function, 346
 Parameter, 348
- CYCLE99 - thread turning
 Function, face thread, 336
 Function, longitudinal thread, 336
 Function, tapered thread, 336
 Parameters, face thread, 345
 Parameters, tapered thread, 343
- Cycles
 Current levels, 204
 Hiding cycle parameters, 216
 Screen forms, 204
- D**
- Data block (SB2), 126
- Deep-hole drilling - CYCLE83
 Function, 296
 Parameter, 300
- Delete

- Multitool, 686
- Deleting
 - Directory, 719
 - Program, 719
- Device
 - Activate/deactivate, 779
 - Enabling, 778
- Directory
 - Copying, 717
 - Creating, 703
 - Deleting, 719
 - Highlight, 715
 - Pasting, 717
 - Properties, 720
 - Selecting, 715
- Displaying
 - Energy consumption, 760
 - HTML documents, 722
 - PDF documents, 722
 - Program level, 129
- DRF (handwheel offset), 143
- Drilling
 - Manual Machine, 613
- Drilling - CYCLE82
 - Function, 289
 - Parameter, 291
- Drilling and thread milling - CYCLE78
 - Function, 306
 - Parameter, 309
- Drilling centered
 - Function, 495
 - Parameters, 497
- Drilling radius, 649
- Drive
 - Error messages, 745
 - Logical drive, 743
 - Setting up, 744
- DRY (dry run feed), 143
- Dual editor, 156
- Duplo number, 648

E

- Easy Extend, 777
 - Activate/deactivate device, 779
 - Enabling a device, 778
 - First commissioning, 780
- Easy Message, 767
 - Commissioning, 769
 - Settings, 776
 - User log on/off, 774
- Editor

- Calling, 149
 - Settings, 157
- Elongated hole - LONGHOLE
 - Function, 428
 - Parameter, 429
- Enabling button, 748
- Energy consumption
 - Displaying, 760
 - Measuring, 761
- Energy-saving profiles, 765
- Engraving - CYCLE60
 - Function, 434
 - Parameter, 441
- Equipping tools
 - Multitool, 685
- Executing
 - Program, 701
- EXTCALL, 724

F

- Face milling - CYCLE61
 - Function, 386
 - Parameter, 389
- Feed data
 - Actual value window, 46
- Fit calculator
 - Master dimension programming, 363
- Freely programmable positions - CYCLE802
 - Parameter, 313
- Function
 - Align milling tool - CYCLE800, 485
 - Setting milling tool - CYCLE800, 488

G

- G code program
 - Blank input, 208
 - Creating, 705
- G functions
 - Display all G groups, 168
- G Functions
 - Displaying selected G groups, 166
- Global user variables, 579
- Grid/frame position pattern - CYCLE801
 - Function, 314
 - Parameter, 316
- Groove - CYCLE930
 - Function, 327
 - Parameter, 329

H

- Handheld terminal 8, 747
- Handwheel
 - Assigning, 100
- High Speed Settings - CYCLE832
 - Function, 490
 - Parameters, 492
- Highlight
 - Directory, 715
 - Program, 715
- HOLES1 - line position pattern
 - Function, 314
 - Parameter, 316
- HOLES2 - circle position pattern
 - Function, 317
- HOLES2 - circular position pattern
 - Parameter, 319
- HT 8
 - Enabling button, 748
 - Overview, 747
 - Touch Panel, 756
 - Traversing keys, 750
 - User menu, 752
 - Virtual keyboard, 754
- HTML documents
 - Displaying, 722

I

- Incremental dimensions, 231
- Interruption point
 - Approaching, 100, 138

J

- Job list
 - Creating, 708
 - Multi-channel support, 542

L

- Ladder viewer, 785
- Limit stops, 606
- Line position pattern - HOLES1
 - Function, 314
 - Parameter, 316
- Loading
 - Multitool, 687
- Logbook, 827
 - display, 827

- Edit the address data, 827
 - Entry search, 828
 - Making an entry, 828
 - Output, 825
- LONGHOLE - elongated hole
 - Function, 428
 - Parameter, 429
- Longitudinal slot - SLOT1
 - Function, 411
 - Parameter, 416

M

- M functions, 169
- Machine control panel
 - Operator controls, 37
- Machine-specific information
 - Save, 825
- Machining
 - Canceling, 124
 - Starting, 123
 - Stopping, 123
- Machining schedule
 - ShopTurn program, 222
- Machining step program, 221
- Magazine
 - Positioning, 669
- Magazine list, 667
- Magazine management, 639
- Maintenance tasks
 - Monitoring/performing, 781
 - Setting up, 783
- Manual
 - Retraction, 114
- Manual Machine, 603
 - Circle, 610
 - Drilling, 613
 - Manual mode, 607
 - Milling, 615
 - Set limit stop, 606
 - Setting the zero offset, 605
 - Simulating, 617
 - Single-cycle machining, 612
 - Straight line, 609
 - Traversing axes, 607
 - Turning, 614
- Manual mode, 105
 - Circle, 610
 - Settings, 121
 - Straight line, 609
 - Traversing axes, 110
- Master dimension

- Fit calculator, 363
- MDA
 - Deleting a program, 104
 - Executing a program, 104
- MDI
 - Loading a program, 102
 - Saving a program, 103
- Measurement
 - Tool, 655
- Measuring
 - Tool, 75
- Messages
 - Displaying, 816
 - Sorting, 817
- Milling
 - B axis, 623
 - Manual Machine, 615
- Mirroring
 - Parameters, 504
- Mode groups, 69
- Mold making view
 - Changing a graphic, 164
 - Changing the section, 165
 - Editing a program block, 162
 - Programs, 159
 - Searching for program blocks, 163
 - Starting, 161
- Multi-channel data
 - Multi-channel support, 539
- Multi-channel editor, 538
 - Synchronized view, 556
 - Time synchronous view, 559
 - Wait points, 559
- Multi-channel program
 - Creating, 538
 - Editing a G code program, 543
 - Editing the ShopTurn program, 545
- Multi-channel support
 - Changing the job list, 542
 - Defining multi-channel data, 539
 - Multi-channel view, 537
 - Program blocks, 553
 - Running-in a program, 564
 - Simulation, 561
- Multi-channel view, 529
 - "Machine" operating area,
 - Block search, 565
 - Multi-channel support, 537
 - Program control, 565
 - Settings, 535, 570
- Multi-edge - CYCLE79
 - Function, 408

- Parameter, 411
- Multitool, 681
 - Creating, 683
 - Delete, 686
 - Equipping tools, 685
 - Loading, 687
 - Parameters in the tool list, 682
 - Positioning, 691
 - Reactivating, 688
 - Relocating, 690
 - Removing tool, 686
 - Unloading, 687

N

- NC directory
 - creating on local drive, 697
- NC/PLC variables
 - Changing, 820
 - Displaying, 818
 - Load symbols, 823
- New contour - CYCLE62
 - Function, 354, 444
 - Parameter, 446
- Number of teeth, 649

O

- Offset, 501
- Online help
 - Context-sensitive, 59
- Open
 - Program, 699
 - Second program, 156
- Open slot - CYCLE899
 - Function, 420
 - Parameter, 427
- Operating area
 - Changing, 47
- Operating mode
 - AUTO, 68
 - Changing, 47
 - JOG, 67, 105
 - MDA, 68
- Operator panel fronts, 25

P

- Parameter
 - Calculating, 50
 - Changing, 50

- Entering, 49
 - Setting milling tool - CYCLE800, 489
 - Parameters
 - Align milling tool - CYCLE800, 487
 - Pasting
 - Directories, 717
 - Program, 717
 - Path milling - CYCLE72
 - Function, 453
 - Parameter, 458
 - Path motion, 506
 - PDF documents
 - Displaying, 722
 - Pinyin
 - Input editor, 54
 - PLC diagnostics
 - Ladder add-on tool, 785
 - PLC symbols
 - Loading, 823
 - POCKET3 - rectangular pocket
 - Function, 389
 - Parameter, 394
 - POCKET4 - circular pocket
 - Function, 394
 - Parameter, 400
 - Polar coordinates, 233
 - Positioning
 - Magazine, 669
 - Multitool, 691
 - Positions
 - display, 320
 - Hiding, 320
 - Preview
 - Program, 714
 - Program
 - Closing, 699
 - Copying, 717
 - Creating with cycle support, 206
 - Deleting, 719
 - Executing, 701
 - Highlight, 715
 - Open, 699
 - Opening a second program, 156
 - Pasting, 717
 - Preview, 714
 - Properties, 720
 - Selecting, 715
 - Program block, 226
 - Changing, 247
 - Copying and inserting, 153
 - Current, 47, 128
 - Delete, 153
 - linked, 226
 - Numbering, 154
 - Repeat, 244
 - Replacing, 239
 - Searching, 150
 - Selecting, 153
 - Program blocks, 155
 - Multi-channel support, 553
 - Program control
 - Modes of operation, 143
 - Multi-channel view, 565
 - Program editing, 131
 - Program header, 237
 - Program level
 - Displaying, 129
 - Program list
 - Creating, 710
 - Program Manager, 693
 - Searching directories and files, 712
 - Program runtime, 172
 - Program settings
 - changing, 248
 - Parameters, 249
 - Program views
 - G code program, 200
 - ShopTurn program, 222
 - Programmed stop 1, 143
 - Programmed stop 2, 143
 - Programming graphics
 - ShopTurn program, 222
 - Programming variables, 217, 256
 - Programs
 - Correcting, 47, 128, 131
 - Editing, 149
 - Managing, 693
 - Renumbering blocks, 154
 - Replacing text, 151
 - Running-in, 126
 - Searching for a program position, 150
 - Selecting, 125
 - Teach-in, 589
 - Properties
 - Directory, 720
 - Program, 720
 - Protection levels
 - Softkeys, 57
 - PRT (no axis motion), 143
 - Punched tape format, 727
- Q**
- Quantity, 246, 662

- R**
- R parameters, 578
 - Reactivating
 - Multitool, 688
 - Tool, 664
 - Reading in
 - Setup data, 737
 - Reaming - CYCLE85
 - Function, 291
 - Parameter, 293
 - Rectangular pocket - POCKET3
 - Function, 389
 - Parameter, 394
 - Rectangular spigot - CYCLE76
 - Function, 400
 - Parameter, 404
 - Reference point approach, 64
 - Relocating
 - Multitool, 690
 - Tool, 670
 - Remote access
 - permit, 833
 - Setting, 831
 - Remote diagnostics, 831
 - Exit, 834
 - requesting, 833
 - Removing tool
 - Multitool, 686
 - Repeat positions
 - Function, 322
 - Parameter, 322
 - Replacement tool number, 648
 - Repositioning, 133
 - Retract, 114
 - Retraction
 - Manually, 114
 - RG0 (reduced rapid traverse), 143
 - Rotation, 502
 - Rough and fine offsets, 85
 - Running-in
 - Multi-channel program, 564
 - Copy, 830
 - Creating, 830
 - Open, 830
 - Search
 - in the Program Manager, 712
 - Logbook entry, 828
 - Search mode, 140
 - Search pointer, 100, 135, 138, 139
 - Selecting
 - Directory, 715
 - Program, 715
 - Service Planner, 781
 - Setting actual values, 73
 - Setting milling tool - CYCLE800
 - Function, 488
 - Parameter, 489
 - Settings
 - Editor, 157
 - For automatic operation, 174
 - For manual operation, 121
 - Multi-channel view, 535, 570
 - Teach-in, 601
 - Tool lists, 679
 - Setup data
 - Backing up, 734
 - reading in, 737
 - ShopTurn program
 - Approach/retraction cycle, 258
 - Circle with known center point, 511
 - Circle with known radius, 513
 - Coordinate transformation, 500
 - Creating, 235
 - Cutting edge (D), 240
 - Cutting rate (V), 241
 - Entering the quantity of workpieces, 246
 - Feedrate (F), 241
 - Machining cycle, approach/retraction, 229
 - Machining planes, 227
 - Machining type, 242
 - Machining with counterspindle, 521
 - Mirroring, 504
 - Polar coordinates, 515
 - Program blocks, 239
 - Program header, 237
 - Program settings, 248
 - Radius compensation, 240
 - Rotation C axis, 505
 - Scaling, 503
 - Spindle speed (S), 241
 - Straight and circle, 506
 - Straight line, 508
 - Straight line polar, 517
- S**
- Save
 - Setup data, 734
 - SB (single blocks), 143
 - SB1, 126
 - SB2, 126
 - SB3, 126
 - Screenshots

- Structure, 226
- Tool (T), 240
- Simulation
 - Alarm display, 198
 - Changing a graphic, 194
 - Manual Machine, 617
 - Multi-channel support, 561
 - Program control, 192
 - Showing and hiding the path display, 191
 - Views for multi-channel support, 562
- Simultaneous recording, 179
- Single block
 - Coarse (SB1), 126
 - Fine (SB3), 126
- Skip blocks, 145
- SKP (skip blocks), 143
- SLOT1 - longitudinal slot
 - Function, 411
 - Parameter, 416
- SLOT2 - circumferential groove
 - Function, 416
- SLOT2 - circumferential slot
 - Parameter, 420
- SMS messages, 767
 - Log, 775
- Spindle chuck data, 96
 - Parameter, 98
 - Store chuck dimensions, 96
- Spindle data
 - Actual value window, 46
- Spindle speed limitation, 95
- Status display, 41
- Stock removal
 - Contour turning - CYCLE952, 366
 - CYCLE952 - contour turning, 366
 - in JOG, 116
 - Parameters, 117
- Stock removal - CYCLE951
 - Function, 323
 - Parameter, 326
- Straight line, 506
 - Manual Machine, 609
- Submode
 - REF POINT, 67
 - REPOS, 67
 - TEACH IN, 68
- Switching off, 63
- Switching on, 63
- Switching over
 - Channel, 69
 - Coordinate system, 71
 - Unit of measurement, 72

- Swivel plane
 - Parameters, swivel plane, 485
- Swivel tool
 - Parameters, align turning tool, 486
- Symbol tables, 804
- Synchronized actions
 - Displaying status, 170
- Synchronized view
 - Multi-channel editor, 556
- Synchronizing
 - Thread, 119
- Synchronizing a counterspindle
 - Function, 572
 - Multi-channel support, 572

T

- Tailstock, 97
- Tapping with compensating chuck - CYCLE840
 - Function, 300
 - Parameter, 305
- Tapping without compensating chuck - CYCLE84
 - Function, 300
 - Parameter, 305
- Teach-in, 589
 - Changing blocks, 598
 - Circle intermediate position CIP, 596
 - Deleting blocks, 600
 - General sequence, 590
 - Inserting a position, 591
 - Inserting blocks, 594
 - Motion type, 592
 - Parameter, 591
 - Rapid traverse G0, 595
 - Selecting a block, 599
 - Settings, 601
 - Traversing block G1, 595
- Templates
 - Creating, 711
 - Storage locations, 711
- Thread
 - Synchronizing, 119
- Thread centered
 - Function, 497
 - Parameters, 499
- Thread chain - CYCLE98
 - Function, 346
 - Parameter, 348
- Thread milling - CYCLE70
 - Function, 430
 - Parameters, 432
- Thread turning - CYCLE99

- Function, face thread, 336
- Function, longitudinal thread, 336
- Function, tapered thread, 336
- Parameters, face thread, 345
- Parameters, longitudinal thread, 340
- Parameters, tapered thread, 343
- Tip angle, 649
- Tool
 - Change type, 678
 - Changing the cutting edge position, 678
 - Delete, 657
 - Details, 676
 - Dimensioning, 643
 - Loading, 657
 - Measure manually, 75
 - Measurement, 655
 - Measuring, 75
 - measuring using a magnifying glass, 81
 - Reactivating, 664
 - Relocating, 670
 - Unloading, 657
- Tool data
 - Actual value window, 45
 - Backing up, 734
 - reading in, 737
- Tool life, 662
- Tool list, 648
 - Open, 648
- Tool lists
 - Settings, 679
- Tool management, 637
 - List filtering, 673
 - Sorting lists, 672
- Tool parameters, 643
- Tool types, 640
- Touch Panel
 - Calibrating, 756
- Transformed view, 679
- Turning
 - Manual Machine, 614

U

- Undercut - CYCLE940
 - Function, DIN thread, 333
 - Function, form E, 330
 - Function, form F, 330
 - Function, thread, 333
 - Parameters, DIN thread, 334
 - Parameters, form E, 331
 - Parameters, form F, 332
 - Parameters, thread, 335

- Unique cutting edge numbers
 - Unique, 650
- Unit of measurement
 - Switching over, 72
- Unloading
 - Multitool, 687
- User agreement, 65
- User data, 577
- User variables
 - Activating, 586
 - Channel GUD, 581
 - Defining, 586
 - Global GUD, 579, 586
 - Local LUD, 583
 - Program PUD, 584
 - R parameters, 578
 - Searching, 585

V

- Variable screen forms, 822
- Virtual keyboard, 754

W

- Wear list, 661
- Work offsets
 - Calling, 243
- Working area limitation, 94
- Workpiece
 - Creating, 704
- Workpiece counter, 172

Z

- Zero offsets
 - Active ZO, 86
 - Delete, 92
 - Displaying details, 90
 - Overview, 84, 87
 - Settable ZO, 89
 - Setting, 73
- Zero point settings
 - Backing up, 734
 - reading in, 737