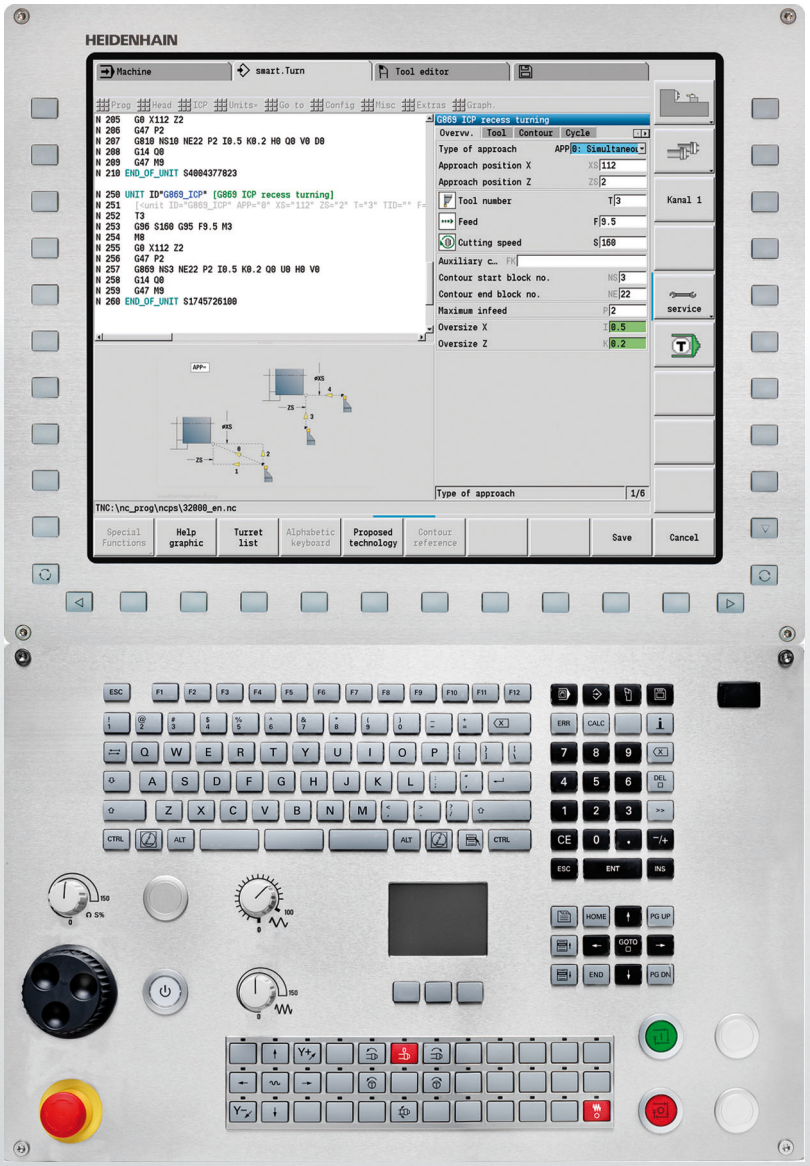
HEIDENHAIN



User's Manual

CNC PILOT 640

NC Software
688946-02
688947-02

English (en)
1/2014

Controls and displays of the CNC PILOT

Keys on visual display unit

Key	Function
	Switches the help graphics between outside and inside machining (only in the cycle programming)
	No function
	Soft keys for selecting functions on screen
	Switches to the soft-key menu at left / right
	Switches to the next menu in the PLC menu

Operating mode keys

Key	Function
	Machine operating modes: ■ Manual Operation ■ Program Run
	Programming modes ■ smart.Turn ■ DINplus ■ DIN/ISO
	Tables for tool data and technology data
	Organization: ■ Parameters ■ File organization ■ Transfer ■ Diagnosis

smart.Turn keys

Key	Function
	Go to the next form
	Next/previous group

Navigation keys

Key	Function
	Up/Down arrow keys
	Left/Right arrow keys
PG UP PG DN	Screen page or dialog page up/down
HOME END	Go to beginning of program/list or to end of program/list

Numeric keypad

Key	Function block
0 9	Number keys 0-9: ■ Numeric input keys ■ Menu operation
.	Decimal point
−/+	Switchover between positive and negative values
ESC	Escape key: Cancelation of dialogs and next higher menu level
INS	Insert key: OK in dialogs and new NC blocks in the editor
DEL	Delete block: Deletes the selected area
	Backspace: Deletes the character to the left of the cursor
CE	CE key: Deletes the error messages in the machine operating mode
ENT	Enter: Confirms the input

Special keys

Key	Function
ERR	Error key: Opens the error window
CALC	Starts the integrated calculator
	Info key: Shows additional information in the parameter editor
GOTO	Activates special functions, such as input options or input of characters as on an alphabetic keyboard

Machine operating panel

Key	Function
	Cycle start
	Cycle stop
	Feed rate stop
	Spindle stop
	Spindle on – M3/M4 direction
	Spindle jog – M3/M4 direction. The spindle rotates as long as you press the key.
	Manual direction keys +X/–X

Operating panel of the CNC PILOT



CNC PILOT 640, software and features

This manual describes functions that are available in the CNC PILOT with NC software number 688946-02 and 688947-02.

The programming of smart.Turn and DIN PLUS are not included in this manual. These functions are described in the User's Manual for smart.Turn and DIN PLUS Programming (ID 685556-xx). Please contact HEIDENHAIN if you require a copy of this manual.

The machine manufacturer adapts the features offered by the control to the capabilities of the specific machine tool by setting machine parameters. Therefore, some of the functions described in this manual may not be among the features provided by the CNC PILOT on your machine tool.

Some of the CNC PILOT functions that are not available on every machine are:

- Positioning of spindle (M19) and driven tool
- Operations with the C or Y axis

Please contact your machine manufacturer for detailed information on the features that are supported by your machine tool.

Many machine manufacturers and HEIDENHAIN offer programming courses. We recommend these courses as an effective way of improving your programming skill and sharing information and ideas with other CNC PILOT users.

HEIDENHAIN also offers the DataPilot MP 620 or DataPilot CP 640 software for personal computers, which is designed to simulate the functions of the MANUALplus 620 and CNC PILOT 640. The DataPilot is suitable for both shop-floor programming as well as off-location program creation and testing. It is also ideal for training purposes. The DataPilot can be run on PCs with WINDOWS operating systems.

Intended place of operation

The CNC PILOT complies with the limits for a Class A devices in accordance with the specifications in EN 55022, and is intended for use primarily in industrially-zoned areas.

Legal information

This product uses open source software. Further information is available on the control under

- Programming and Editing operating mode
- MOD function
- LICENSE INFO soft key

New functions of software 688945-02

- In the program simulation, the current contour description (of workpiece blank and finished part) can be mirrored and saved. In smart.Turn, these contours can be reinserted (see page 481)
- On machines with opposing spindle, the workpiece spindle can now be selected in the TSF menu (see page 94)
- On machines with opposing spindle, its datum can be shifted (see Page 94)
- The user documentation is now also available in the context-sensitive help system TURNguide (see Page 64)
- You can make your own project folder in the project management, so that you can centrally manage associated files (see Page 118)
- With a manual tool change system it is possible to insert tools that are not in the turret during a program run (see Page 492)
- Engraving cycles are now available in the Teach-In mode of operation (see Page 331)
- During tool data backup, you can now select in a dialog window the data to be saved or restored (see Page 561)
- The G30 function is now available for converting G functions, M functions or spindle numbers, as well as for mirroring traverse paths and tool dimensions (see the smart.Turn and DIN Programming User's Manual)
- The "traverse to a fixed stop" function (G916) is now available for transferring the workpiece to the second traversable spindle or for pressing the tailstock against the workpiece (see the smart.Turn and DIN Programming User's Manual)
- The G925 function makes it possible to define and monitor the maximum contact force for an axis. This function can be applied to use the opposing spindle as a mechatronic tailstock, for example (see the smart.Turn and DIN Programming User's Manual)
- Controlled parting using servo-lag monitoring (G917) can now be activated to prevent collisions caused by incomplete parting processes (see the smart.Turn and DIN Programming User's Manual)

- The spindle synchronization option G720 synchronizes the shaft speeds of two or more spindles so that they rotate synchronously with a gear ratio or a defined offset (see the smart.Turn and DIN Programming User's Manual)
- In combination with the synchronization (G720) of main spindle and tool spindle, the new "Hobbing" cycle (G808) is available for milling external teeth and profiles (see the smart.Turn and DIN Programming User's Manual)
- With G924, a "fluctuating speed" can now be programmed to prevent resonance (see the smart.Turn and DIN Programming User's Manual)

New functions of software 688945-03 and 688946-01

- In the **Organization** mode of operation, you can grant or restrict access to the control by using the EXTERNAL ACCESS soft key (see also "Organization mode of operation" on page 520)
- The pocket calculator can now be activated in each application and also remains active after a change in operating modes. The **Get current value** and **Load current value** soft keys enable you to fetch numerical values from an active input field or to transfer them to an active input field (see also "Integrated calculator" on page 56)
- Tool touch probes can now be calibrated in the Machine Setup menu (see also "Calibrating the tool touch probe" on page 96)
- The workpiece datum can now also be set in the direction of the Z axis using a touch probe (see also "Machine setup" on page 89)
- In Teach-in mode, the oversizes RI and RK for the workpiece blank were introduced for finishing in the recess-turning cycles (see also "Recess turning, radial finishing—expanded" on page 236)
- On machines with a B axis it is now also possible to drill, bore, and mill in oblique planes. In addition to this, the B axis enables you to use tools even more flexibly during turning (see smart.Turn and DIN Programming User's Manual).
- The control now provides numerous touch probe cycles for various applications (see smart.Turn and DIN Programming User's Manual):
  - Calibrating a touch trigger probe
  - Measuring circles, circle segments, angle and position of the C axis
  - Misalignment compensation
  - Single- point and double-point measurement
  - Finding a hole or stud
  - Zero point setting in the Z or C axis
  - Automatic tool measurement

- The new TURN PLUS function automatically generates NC programs for turning and milling operations based on a fixed machining sequence (see smart.Turn and DIN Programming User's Manual).
- The G940 function now provides a way to calculate the tool lengths in the basic (definition) position of the B axis (see smart.Turn and DIN Programming User's Manual).
- For machining operations that require rechucking, you can define a separation point on the contour description with G44 (see the smart.Turn and DIN Programming User's Manual).
- The G927 function enables you to convert tool lengths to the reference position of the tool (B axis = 0) (see smart.Turn and DIN Programming User's Manual).
- Recesses that were defined with G22 can now be machined with the new Cycle 870 ICP Recessing (see smart.Turn and DIN Programming User's Manual).

New functions of software 68894x-02

- The "Zero point shift" miscellaneous function was introduced in ICP (see also "Zero point shift" on page 375)
- In ICP contours, fit dimensions and inside threads can now be calculated using an input form (see also "Fits and inside threads" on page 370)
- The "Copy Linear, Circular and Mirror" miscellaneous functions were introduced in ICP (see also "Copying a contour section in linear series" on page 375)
- The system time can now be set using an input form (see also "Displaying operating times" on page 97)
- The parting cycle G859 was expanded by the parameters K, SD and U (see also "Parting" on page 253)
- The ICP recess turning cycles now allow defining an approach and departure angle (see also "ICP recess turning, radial finishing" on page 244)
- With TURN PLUS you can now create programs for machining with an opposing spindle and for multipoint tools (see the smart.Turn and DIN Programming User's Manual)
- In the G797 Area Milling function, milling contours can now be selected (see the smart.Turn and DIN Programming User's Manual)
- The G720 function was expanded by the Y parameter (see the smart.Turn and DIN Programming User's Manual)
- The G860 function was expanded by the O and U parameters (see the smart.Turn and DIN Programming User's Manual)

About this manual

The symbols used in this manual are described below.



This symbol indicates that important information about the function described must be considered.



This symbol indicates that there is one or more of the following risks when using the described function:

- Danger to workpiece
- Danger to fixtures
- Danger to tool
- Danger to machine
- Danger to operator



This symbol indicates that the described function must be adapted by the machine tool builder. The function described may therefore vary depending on the machine.



This symbol indicates that you can find detailed information about a function in another manual.

Do you want any changes, or have you found any errors?

We are continuously striving to improve our documentation for you. Please help us by sending your requests to the following e-mail address: **tnc-userdoc@heidenhain.de**.

Contents

Introduction and fundamentals 1
Basics of operation 2
Machine mode of operation 3
Teach-in mode 4
ICP programming 5
Graphic simulation 6
Tool and technology database 7
Organization mode of operation 8
Tables and overviews 9
Overview of cycles 10

1 Introduction and fundamentals 33

1.1 The CNC PILOT 34
1.2 Configuration 35
Slide position 35
Tool carrier systems 35
The C axis 35
The Y axis 36
Full-surface machining 37
1.3 Features 38
Configuration 38
Operating modes 38
1.4 Data backup 40
1.5 Explanation of terms 41
1.6 CNC PILOT design 42
1.7 Fundamentals 43
Position encoders and reference marks 43
Axis designations 43
Coordinate system 44
Absolute coordinates 44
Incremental coordinates 45
Polar coordinates 45
Machine zero point 45
Workpiece zero point 46
Units of measure 46
1.8 Tool dimensions 47
Tool length 47
Tool compensation 47
Tool-tip radius compensation (TRC) 48
Milling cutter radius compensation (MCRC) 48

2 Basics of operation 49

2.1 General information on operation 50
Operation 50
Setup 50
Programming – Teach-in mode 50
Programming – smart.Turn 50
2.2 The CNC PILOT screen 51
2.3 Operation and data input 52
Operating modes 52
Menu selection 53
Soft keys 53
Data input 54
smart.Turn dialogs 54
List operations 54
Alphanumeric keyboard 55
2.4 Integrated calculator 56
Calculator functions 56
Adjusting the position of the calculator 58
2.5 Types of programs 59
2.6 The error messages 60
Display of errors 60
Opening the error window 60
Closing the error window 60
Detailed error messages 61
"Details" soft key 61
Clearing errors 62
Error log file 62
Keystroke log file 63
Saving service files 63
2.7 TURNguide context-sensitive help system 64
Application 64
Working with the TURNguide 65
Downloading current help files 69

3 Machine mode of operation 71

3.1 Machine mode of operation 72
3.2 Switch-on / Switch-off 73
Switch-on 73
Monitoring EnDat encoders 73
Traversing the reference marks 74
Switch-off 75
3.3 Machine data 76
Input of machine data 76
Machine data display 78
Cycle statuses 81
Axis feed rate 82
Spindle 82
3.4 Setting up a tool list 83
Machine with turret 83
Machine with multifix 83
Tools in different quadrants 84
Filling the turret list from the database 85
Filling the turret list 86
Tool call 87
Driven tools 87
Tool life monitoring 88
3.5 Machine setup 89
Defining the workpiece zero point 90
Homing the axes 91
Setting the protection zone 92
Defining the tool change position 93
Setting C-axis values 94
Setting up machine dimensions 95
Calibrating the tool touch probe 96
Displaying operating times 97
Setting the system time 98
3.6 Tool measurement 99
Touch off 100
Touch probe (tool touch probe) 101
Optical gauge 102
Tool compensation 103
3.7 "Manual Operation" operating mode 104
Tool change 104
Spindle 104
Handwheel operation 104
Manual direction keys 105
Teach-in cycles in Manual mode 105

3.8 Teach-in mode 106
Teach-in mode 106
Programming Teach-in cycles 106
3.9 Program Run mode 107
Loading a program 107
Comparing a tool list 108
Before executing a program 108
Finding a start block 109
Program execution 110
Entering compensation values during program run 111
Program execution in "dry run" mode 114
3.10 Graphic simulation 115
3.11 Program management 116
Program selection 116
File manager 117
Project management 118
3.12 Conversion into DIN format 119
Making a conversion 119
3.13 Units of measure 120



4 Teach-in mode 121

4.1 Working with cycles 122
Cycle starting point 122
Help graphics 123
DIN macros 123
Graphical test run (simulation) 123
Contour follow-up in Teach-in mode 124
Cycle keys 124
Switching functions (M functions) 125
Comments 125
Cycle menu 126
Addresses used in many cycles 128
4.2 Workpiece blank cycles 129
Bar/tube blank 130
ICP workpiece blank contour 131
4.3 Single cut cycles 132
Rapid traverse positioning 133
Approaching the tool change position 134
Linear machining, longitudinal 135
Linear machining, transverse 136
Linear machining at angle 137
Circular machining 139
Chamfer 141
Rounding arc 143
M functions 145

4.4 Turning cycles 146
Tool position 147
Roughing, longitudinal 149
Cut, transverse 151
Roughing, longitudinal—expanded 153
Roughing, transverse—expanded 155
Finishing cut, longitudinal 157
Finishing cut, transverse 158
Finishing cut, longitudinal—expanded 159
Finishing cut, transverse—expanded 161
Cut, longitudinal plunge 163
Cut, transverse plunge 165
Cut, longitudinal plunging—expanded 167
Cut, transverse plunging—expanded 169
Cut, longitudinal finishing plunge 171
Cut, transverse finishing plunge 172
Cut, longitudinal finishing plunge—expanded 174
Cut, transverse finishing plunge—expanded 176
Cut, ICP contour-parallel, longitudinal 178
Cut, ICP contour-parallel, transverse 180
Cut, ICP contour-parallel, longitudinal finishing 182
Cut, ICP contour-parallel, transverse finishing 184
ICP cutting, longitudinal 186
ICP cutting, transverse 188
ICP longitudinal finishing cut 190
ICP transverse finishing cut 192
Examples of turning cycles 194



4.5 Recessing cycles 198
Cutting and infeed directions for recessing cycles 198
Undercut position 199
Contour forms 199
Recessing, radial 200
Recessing, axial 202
Recessing, radial—expanded 204
Recessing, axial—expanded 206
Recessing radial, finishing 208
Recessing axial, finishing 210
Recessing radial, finishing—expanded 212
Recessing axial, finishing—expanded 214
ICP recessing cycles, radial 216
ICP recessing cycles, axial 218
ICP recessing, radial finishing 220
ICP recessing, axial finishing 222
Recess turning 224
Recess turning, radial 225
Recess turning, axial 226
Recess turning, radial—expanded 228
Recessing turning, axial—expanded 230
Recess turning, radial finishing 232
Recess turning, axial finishing 234
Recess turning, radial finishing—expanded 236
Recess turning, axial finishing—expanded 238
ICP recess turning, radial 240
ICP recess turning, axial 242
ICP recess turning, radial finishing 244
ICP recess turning, axial finishing 246
Undercutting type H 248
Undercutting type K 250
Undercutting type U 251
Parting 253
Examples of recessing cycles 255

4.6 Thread and undercut cycles 257
Thread position, undercut position 257
Handwheel superimposition 258
Feed angle, thread depth, proportioning of cuts 258
Thread run-in / thread run-out 259
Last cut 260
Thread cycle (longitudinal) 261
Thread cycle (longitudinal)—expanded 263
Tapered thread 265
API thread 267
Recut (longitudinal) thread 269
Recut (longitudinal) thread—expanded 271
Recut tapered thread 273
Recut API thread 275
Undercut DIN 76 277
Undercut DIN 509 E 279
Undercut DIN 509 F 281
Examples of thread and undercut cycles 283
4.7 Drilling cycles 285
Drilling, axial 286
Radial drilling 288
Deep-hole drilling, axial 290
Deep-hole drilling, radial 293
Tapping, axial 295
Tapping, radial 297
Thread milling, axial 299
Examples of drilling cycles 301
4.8 Milling cycles 303
Rapid positioning milling 304
Slot, axial 305
Figure, axial 307
ICP contour, axial 311
Face milling 314
Slot, radial 317
Figure, radial 319
ICP contour, radial 323
Helical-slot milling, radial 326
Milling direction for contour milling 328
Milling direction for pocket milling 329
Example of milling cycle 330
Engraving, axial 331
Engraving, radial 333
Engraving, axial/radial 335

4.9 Drilling and milling patterns 336
Linear drilling pattern, axial 337
Milling pattern linear, axial 339
Drilling pattern circular, axial 341
Milling pattern circular, axial 343
Drilling pattern linear, radial 345
Linear milling pattern, radial 347
Drilling pattern circular, radial 349
Circular milling pattern, radial 351
Examples of pattern machining 353
4.10 DIN cycles 356
DIN cycle 356

5 ICP programming 359

5.1 ICP contours 360
Loading contours 360
Form elements 361
Machining attributes 361
Calculation of contour geometry 362
5.2 ICP editor in cycle mode 363
Editing contours for cycles 363
File organization with the ICP editor 364
5.3 ICP editor in smart.Turn 365
Editing a contour in smart.Turn 366
5.4 Creating an ICP contour 368
Entering an ICP contour 368
Absolute or incremental dimensioning 369
Transitions between contour elements 369
Fits and inside threads 370
Polar coordinates 371
Angular input 371
Contour graphics 372
Selection of solutions 373
Colors in contour graphics 373
Selection functions 374
Zero point shift 375
Copying a contour section in linear series 375
Copying a contour section in circular series 376
Copying a contour section by mirroring 376
Inverting 376
Contour direction (cycle programming) 377
5.5 Editing ICP contours 378
Superimposing form elements 378
Adding contour elements 378
Editing or deleting the last contour element 379
Deleting a contour element 379
Editing contour elements 380
5.6 The zoom function in the ICP editor 385
Changing the view 385
5.7 Defining the workpiece blank 386
"Bar" blank 386
"Tube" blank 386
5.8 Contour elements of a turning contour 387
Basic elements of a turning contour 387
Contour form elements 391

5.9 Contour elements on face 398
Starting point of face contour 398
Vertical lines on face 399
Horizontal lines on face 400
Line at angle on face 401
Circular arc on face 402
Chamfer/rounding arc on the face 403
5.10 Contour elements on lateral surface 404
Starting point of lateral surface contour 404
Vertical lines on lateral surface 406
Horizontal lines on lateral surface 406
Line at angle on lateral surface 407
Circular arc on lateral surface 408
Chamfers/rounding arcs on a lateral surface 409
5.11 C and Y axis machining in smart.Turn 410
Reference data, nested contours 411
Representation of the ICP elements in the smart.Turn program 412
5.12 Face contours in smart.Turn 413
Reference data for complex face contours 413
TURN PLUS attributes 414
Circle on face 414
Rectangle on face 415
Polygon on face 416
Linear slot on face 417
Circular slot on face 417
Hole on face 418
Linear pattern on face 419
Circular pattern on face 420
5.13 Lateral surface contours in smart.Turn 421
Reference data of lateral surface 421
TURN PLUS attributes 422
Circle on lateral surface 423
Rectangle on lateral surface 424
Polygon on lateral surface 425
Linear slot on lateral surface 426
Circular slot on lateral surface 427
Hole on lateral surface 428
Linear pattern on lateral surface 429
Circular pattern on lateral surface 430

5.14 Contours in the XY plane 432
Reference data in XY plane 432
Starting point of contour in XY plane 433
Vertical lines in XY plane 433
Horizontal lines in XY plane 434
Line at angle in XY plane 435
Circular arc in XY plane 436
Chamfer/rounding arc in XY plane 437
Circle in XY plane 438
Rectangle in XY plane 439
Polygon in XY plane 440
Linear slot in XY plane 441
Circular slot in XY plane 442
Hole in XY plane 443
Linear pattern in XY plane 444
Circular pattern in XY plane 445
Single surface in XY plane 446
Centric polygon in XY plane 447
5.15 Contours in the YZ plane 448
Reference data in YZ plane 448
TURN PLUS attributes 449
Starting point of contour in YZ plane 450
Vertical lines in YZ plane 450
Horizontal lines in YZ plane 451
Line at angle in YZ plane 452
Circular arc in YZ plane 453
Chamfer/rounding arc in YZ plane 454
Circle in YZ plane 455
Rectangle in YZ plane 456
Polygon in YZ plane 457
Linear slot in YZ plane 458
Circular slot in YZ plane 458
Hole in YZ plane 459
Linear pattern in YZ plane 460
Circular pattern in YZ plane 461
Single surface in YZ plane 462
Centric polygons in YZ plane 463
5.16 Loading existing contours 464
Integrating cycle contours in smart.Turn 464
DXF contours (option) 465



6 Graphic simulation 467

6.1 Simulation mode of operation 468
Using the graphic simulation 469
The miscellaneous functions 470
6.2 Simulation window 471
Setting up the views 471
Single-window view 472
Multiple window view 472
6.3 Views 473
Traverse path display 473
Tool depiction 474
Material-removal graphic 474
3-D view 475
6.4 The zoom function 476
Adjusting the visible section 476
6.5 Simulation with mid-program startup 478
Startup block with smart.Turn programs 478
Mid-program startup in cycle programs 479
6.6 Time calculation 480
Showing the machining times 480
6.7 Saving the contour 481
Saving the generated contour in the simulation 481

7 Tool and technology database 483

7.1 Tool database 484
Tool types 484
Multipoint tools 485
Tool life management 485
7.2 Tool editor 486
Tool list 486
Editing the tool data 487
Tool texts 488
Editing multipoint tools 489
Editing tool-life data 491
Manual change systems 492
7.3 Tool data 497
General tool parameters 497
Standard turning tools 500
Recessing tools 501
Thread-cutting tools 502
Twist drills and indexable-insert drills 503
NC center drill 504
Centering tool 505
Counterbore 506
Countersink 507
Tap 508
Standard milling tools 509
Thread milling tools 510
Angle cutters 511
Milling pins 512
Touch probes 513
7.4 Technology database 514
Technology editor 515
Editing a workpiece material or cutting material list 516
Displaying/editing cutting data 517

8 Organization mode of operation 519

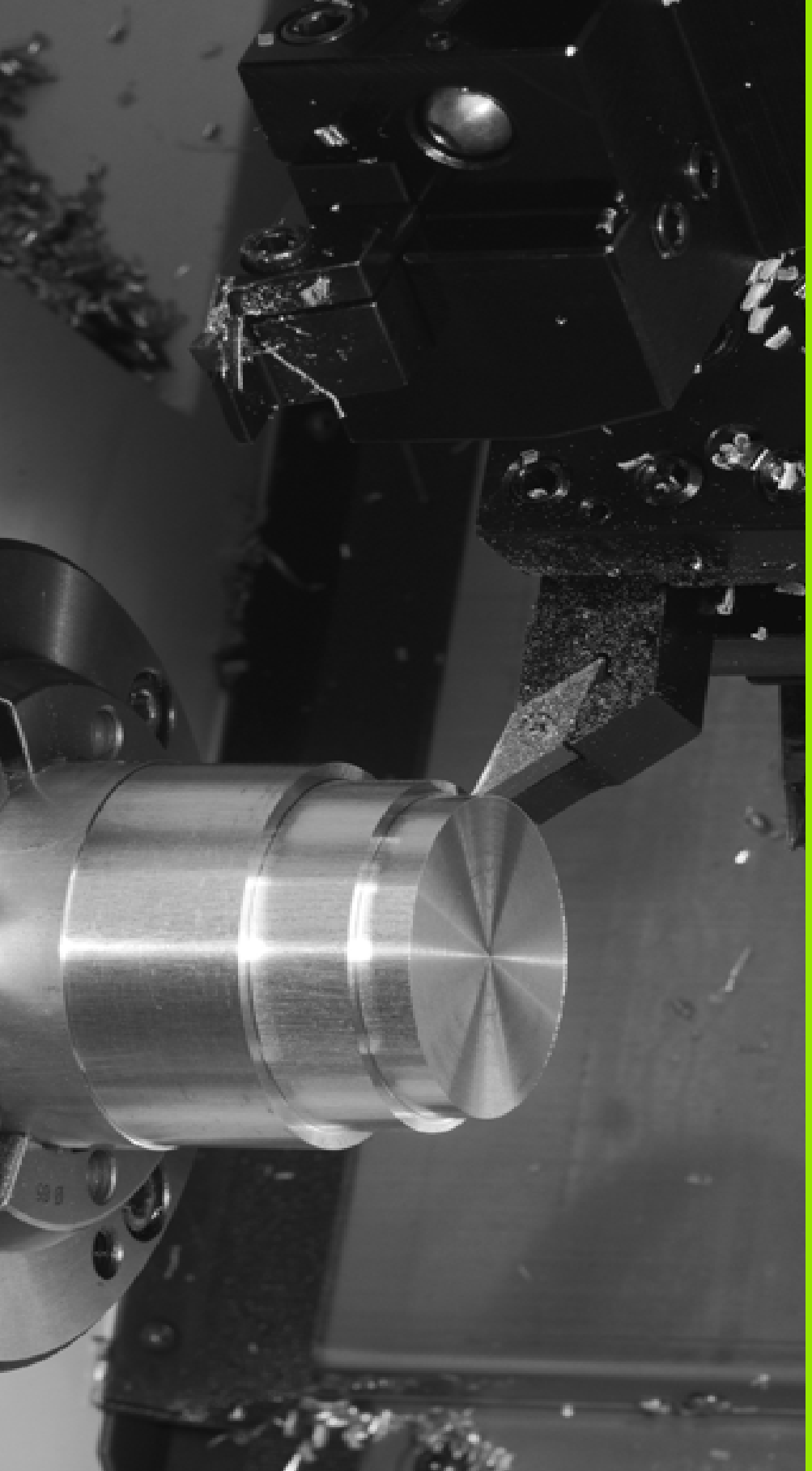
8.1 Organization mode of operation 520
8.2 Parameters 521
Parameter editor 521
List of user parameters 523
Machining parameters (processing) 527
General settings 527
Thread cutting 542
8.3 Transfer 546
Data backup 546
Data exchange with TNCremo 546
External access 546
Connections 547
Ethernet interface CNC PILOT 620 548
Ethernet interface CNC PILOT 640 549
USB connection 556
Data transfer options 557
Transferring programs (files) 558
Transferring parameters 560
Transferring tool data 561
Service files 562
Creating a data backup file 563
Importing NC programs from predecessor controls 564
8.4 Service pack 568
Installing a service pack 568

9 Tables and overviews 571

9.1 Thread pitch 572
Thread parameters 572
Thread pitch 573
9.2 Undercut parameters 579
DIN 76—undercut parameters 579
DIN 509 E – undercut parameters 581
DIN 509 F – undercut parameters 581
9.3 Technical information 582
9.4 Compatibility in DIN programs 591
Syntax elements of the CNC PILOT 640 593

10 Overview of cycles 605

10.1 Workpiece blank cycles, single cut cycles 606
10.2 Turning cycles 607
10.3 Recessing and recess-turning cycles 608
10.4 Thread cycles 609
10.5 Drilling cycles 610
10.6 Milling cycles 611



1

Introduction and fundamentals

1.1 The CNC PILOT



The CNC PILOT was conceived for CNC lathes. It is suitable for horizontal and vertical lathes. The CNC PILOT supports lathes with tool turrets. The tool carrier of horizontal lathes can be located in front of or behind the workpiece.

The CNC PILOT supports lathes with spindle, one slide (X and Z axis), C axis or positionable spindle, driven tool and machines with a Y axis.

Regardless of whether you are turning simple parts or complex workpieces, the CNC PILOT provides you with the benefits of graphical contour input and convenient programming with smart.Turn. Programming with variables, controlling special machine components, or using externally created programs, etc. is no problem: Switch to DINplus. This programming mode helps you solve all your special tasks.

The MANUALplus also offers the powerful Teach-in mode. It enables you to perform simple machining, rework or repair operations without writing NC programs.

The CNC PILOT supports operations with the C axis in cycle, smart.Turn and DIN programming. In the Y axis, the CNC PILOT supports operations with smart.Turn and DIN programming.

1.2 Configuration

In the standard version, the control is equipped with the axes X and Z and a main spindle. Optionally, a C axis, a Y axis, and a driven tool can be configured.

Slide position

The machine tool builder configures the CNC PILOT. These are the available possibilities:

- Z axis **horizontal** with tool slide behind the workpiece
- Z axis **horizontal** with tool slide in front of the workpiece
- Z axis **vertical** with tool slide to the right of the workpiece

The menu symbols, help graphics and graphic representations during ICP and simulation consider the slide position.

The representations in this User's Manual assume a lathe with tool carrier behind the workpiece.

Tool carrier systems

The CNC PILOT supports turrets with a number n of tool mounts as tool carriers.

The C axis

With a C axis you can drill and mill a workpiece on its face and lateral surfaces.

When the C axis is used, one axis interpolates linearly or circularly with the spindle in the given working plane, while the third axis interpolates linearly.

The CNC PILOT supports part program creation with the C axis in:

- Teach-in mode
- smart.Turn programming
- DINplus programming



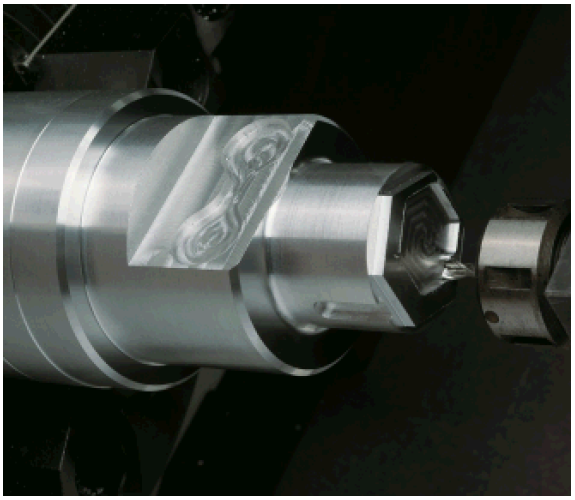
The Y axis

With a Y axis you can drill and mill a workpiece on its face and lateral surfaces.

During use of the Y axis, two axes interpolate linearly or circularly in the given working plane, while the third axis interpolates linearly. This enables you to machine slots or pockets, for example, with plane floors and perpendicular edges. By defining the spindle angle, you can determine the position of the milling contour on the workpiece.

The CNC PILOT supports program creation with the Y axis in:

- Teach-in mode
- smart.Turn programs
- in DINplus programs

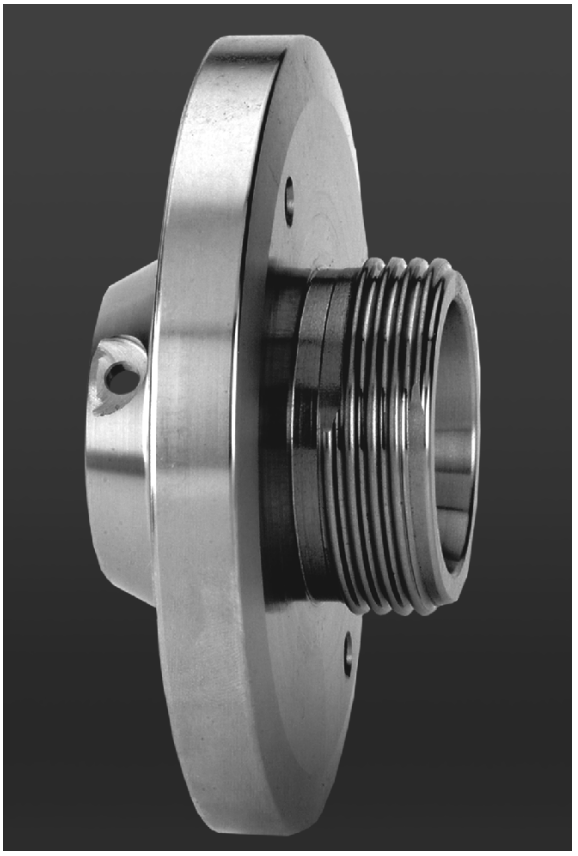


Full-surface machining

Functions like angle-synchronous part transfer with rotating spindle, traversing to a stop, controlled parting, and coordinate transformation ensures efficient machining as well as simple programming of full-surface machining.

The CNC PILOT supports full-surface machining for all common machine designs.
Examples: Lathes with

- Rotating gripper
- Movable opposing spindle
- Several spindles and tool carriers



1.3 Features

Configuration

- Basic version: X and Z axis, spindle
- Positionable spindle and driven tool
- C axis and driven tool
- Y axis and driven tool
- B axis for machining a tilted plane
- Digital current and speed control

Operating modes

Manual operation

Manual slide movement through axis-direction keys or electronic handwheels.

Graphic support for entering and running Teach-in cycles without saving the machining steps in alternation with manual machine operation.

Thread reworking (thread repair in a second workpiece setup).

Teach-in mode

Sequential linking of Teach-in cycles, where each cycle is run immediately after input, or is graphically simulated and subsequently saved.

Program run

All are possible in single-block and full-sequence modes

- DINplus programs
- smart.Turn programs
- Teach-in programs

Setup functions

- Workpiece datum setting
- Definition of tool-change position
- Definition of protection zone
- Tool measurement through touch-off, touch probe or optical gauge

Programming

- Teach-in programming
- Interactive Contour Programming (ICP)
- smart.Turn programming
- Automatic program creation with TURN PLUS
- DINplus programming

Graphic simulation

- Graphic depiction of the sequence of smart.Turn or DINplus programs and graphic depiction of a Teach-in cycle or Teach-in program
- Simulation of the tool paths as wire-frame or cutting-path graphics, special identification of the rapid-traverse paths
- Machining simulation (2-D material-removal graphic)
- Side or face view, or 2-D view of cylindrical surface
- Display of programmed contours
- Shifting and magnifying functions

Tool system

- Database for 250 tools, optionally 999 tools
- Description can be entered for every tool
- Optional support of multipoint tools (tools with multiple reference points or multiple cutting edges)
- Turret or multifix system

Technology database

- The cutting data are entered in the cycle or in the UNIT as default values.
- 9 workpiece-material/tool-material combinations (144 entries)
- Optionally 62 workpiece-material/tool-material combinations (992 entries)

Interpolation

- Linear: In 2 principal axes (max. ± 100 m)
- Circular: in 2 axes (radius max. 999 m)
- C axis: Interpolation in the linear axes X and Z with the C axis
- Y axis: Linear or circular interpolation of two axes in the given plane. The respective third axis can simultaneously perform linear interpolation.
  - G17: XY plane
  - G18: XZ plane
  - G19: YZ plane
- B axis: Drilling, boring and milling operations in oblique planes

1.4 Data backup

HEIDENHAIN recommends saving new programs and files created on a PC at regular intervals.

HEIDENHAIN provides a backup function for this purpose in the data transfer software TNCremoNT. Your machine tool builder can provide you with a copy.

You additionally need a data medium on which all machine-specific data, such as the PLC program, machine parameters, etc., are stored. Please contact your machine tool builder.

1.5 Explanation of terms

- **Cursor:** In lists, or during data input, a list item, an input field or a character is highlighted. This "highlight" is called a cursor. Entries and operations, like copying, deleting, inserting a new item, etc., refer to the current cursor position.
- **Arrow keys:** The cursor is moved with the horizontal and vertical arrow keys and with the PG UP / PG DN keys.
- **Page keys:** The PG UP / PG DN keys are also called "Page keys."
- **Navigating:** Within a list or an input box, you can move the cursor to any position you would like to check, change, delete or add to. In other words, you "navigate" through the list.
- **Active/ inactive windows, functions, menu items:** Of all windows that are displayed on the screen, only one is active. That means, any data you type on the keyboard or keypad are entered in the active window only. In the active window the title bar is shown in color. In the inactive windows, the title bar appears dimmed. Inactive function keys or menu keys also appear dimmed.
- **Menu, menu key:** The CNC PILOT arranges the available functions and function groups in a 9-field box. This box is called a menu. Each symbol in the menu is a menu key.
- **Editing:** Editing is changing, deleting and adding to parameters, commands, etc. within programs, tool data or parameters.
- **Default value:** If the parameters of cycles or DIN commands are preassigned values, these values are referred to as default values. These values are used if you do not enter the parameters.
- **Byte:** The capacity of storage media is measured in bytes. Since the CNC PILOT features an internal memory, the individual program lengths are expressed in bytes.
- **Extension:** File names consist of the actual name and the extension. The name part and the extension part are separated by a dot ("."). The extension indicates the type of file. Examples:
  - *.NC "DIN programs"
  - *.NCS "DIN subprograms (DIN macros)"
- **Soft key:** Soft keys are the unmarked keys along the side of the screen. The meaning of each key is shown on the screen.
- **Form:** The individual pages of a dialog are shown as easy-to-fill forms.
- **UNITS:** A UNIT is a group of function united into a dialog in smart.Turn.

1.6 CNC PILOT design

The dialog between machinist and control takes place via:

- Screen
- Soft keys
- Data input keypad
- Machine operating panel

The entered data can be displayed and checked on the screen. With the soft key directly below the screen, you can select functions, capture position values, confirm entries, and a lot more.

With the ERR key you can call error and PLC information.

The data input keyboard (operating panel) serves for the input of machine data, positioning data, etc. The CNC Pilot has an alphanumeric keyboard for easy input of tool descriptions, program descriptions or comments in an NC program. The machine operating panel contains all necessary controls for manual operation of the lathe.

Cycle programs, ICP contours and NC programs are stored in memory in the CNC PILOT.

For data exchange and data backup, you can use the **Ethernet interface** and the **USB interface.**

1.7 Fundamentals

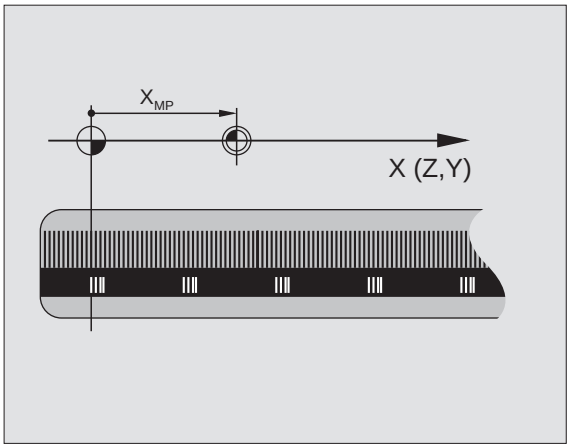
Position encoders and reference marks

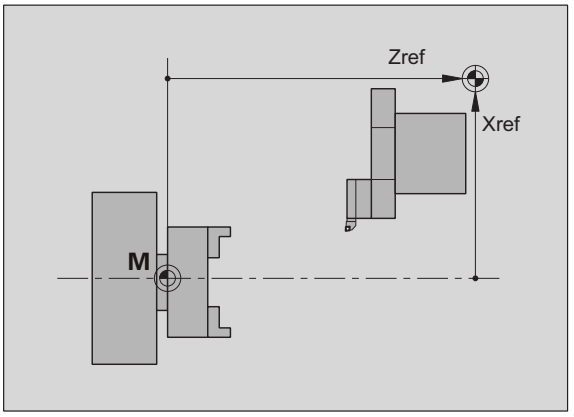
The machine axes are equipped with position encoders that register the positions of the slide or tool. When a machine axis moves, the corresponding position encoder generates an electrical signal. The control evaluates this signal and calculates the precise actual position of the machine axis.

If there is a power interruption, the calculated position will no longer correspond to the actual position of the machine slide. To recover this association, incremental position encoders are provided with reference marks. The scales of the position encoders contain one or more reference marks that transmit a signal to the control when they are crossed over. This enables the CNC PILOT to re-establish the assignment of the displayed position to the current machine position. For linear encoders with distance-coded reference marks, you only need to move each axis a maximum of 20 mm (0.8 in.) for these, and a maximum of 20° for angle encoders.

If incremental encoders are without reference marks, fixed reference positions have to be traversed after switch-on. The control knows the exact distance between these reference points and the machine datum (see figure).

With absolute encoders, an absolute position value is transmitted to the control immediately upon switch-on. In this way the assignment of the actual position to the machine slide position is re-established directly after switch-on.





Axis designations

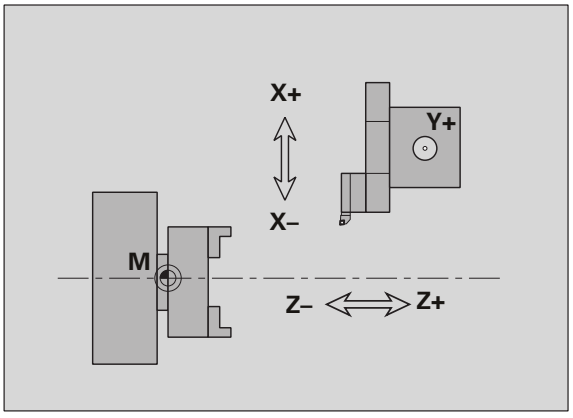
The cross slide is referred to as the **X axis** and the saddle as the **Z axis.**

All X-axis values that are displayed or entered are regarded as **diameters.**

Lathes with **Y axis**: The Y axis is perpendicular to the X axis and Z axis (Cartesian system).

When programming paths of traverse, remember to:

- Program a **positive value** to depart the workpiece.
- Program a **negative value** to approach the workpiece.



Coordinate system

The meanings of the coordinates X, Y, Z, and C are specified in DIN 66 217.

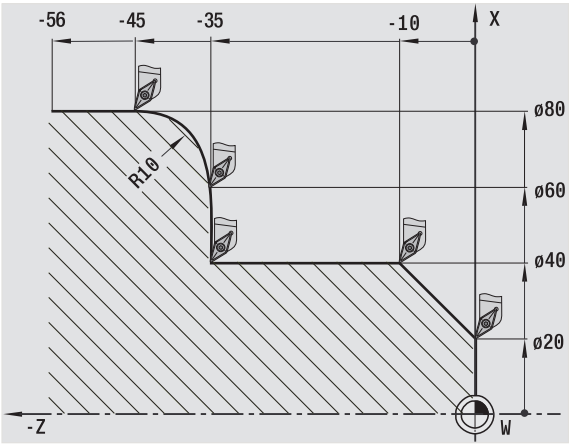
The coordinates entered for the **principal axes** X and Z are referenced to the workpiece zero point. The angles entered for the rotary axis (C axis) are referenced to the datum of the C axis.

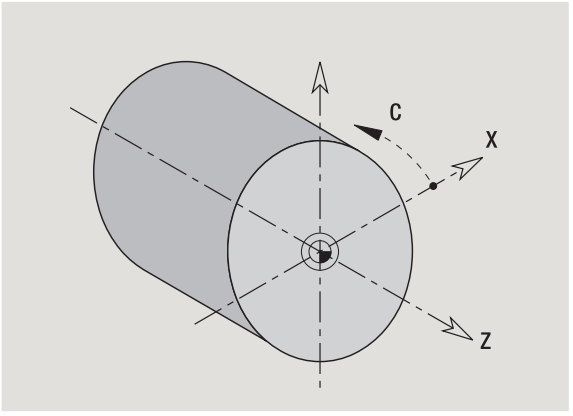
The axis designations X and Z describe positions in a two-dimensional coordinate system. As you can see from the figure to the center right, the position of the tool tip is clearly defined by its X and Z coordinates.

The CNC PILOT can connect points by linear and circular paths of traverse (interpolations). Workpiece machining is programmed by entering the coordinates for a succession of points and connecting the points by linear or circular paths of traverse.

Like the paths of traverse, you can also describe the complete contour of a workpiece by defining single points through their coordinates and connecting them by linear or circular paths of traverse.

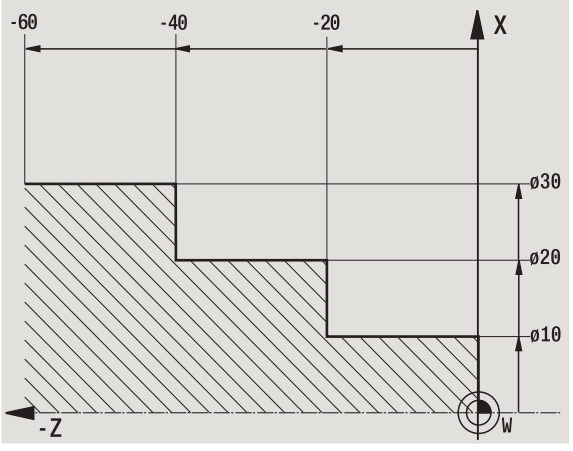
Positions can be programmed to an accuracy of 1 µm (0.001 mm). This is also the accuracy with which they are displayed.





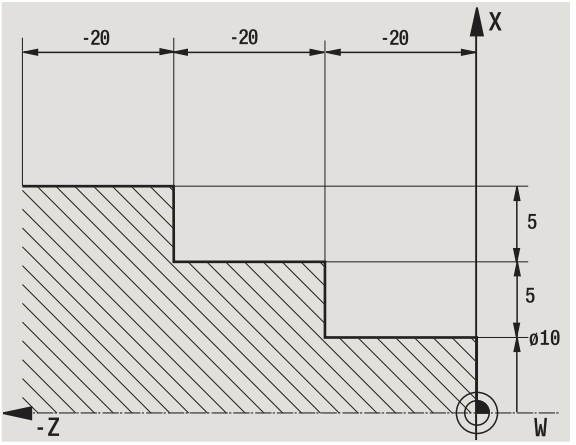
Absolute coordinates

If the coordinates of a position are referenced to the workpiece datum, they are referred to as absolute coordinates. Each position on a workpiece is clearly defined by its absolute coordinates (see figure).



Incremental coordinates

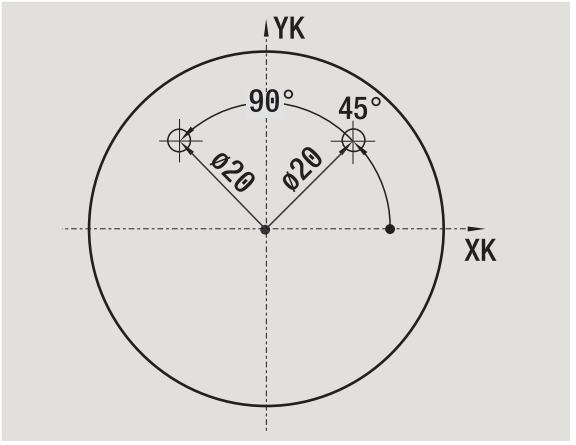
Incremental coordinates are always given with respect to the last programmed position. They specify the distance from the last active position and the subsequent position. Each position on a workpiece is clearly defined by its incremental coordinates (see figure).



Polar coordinates

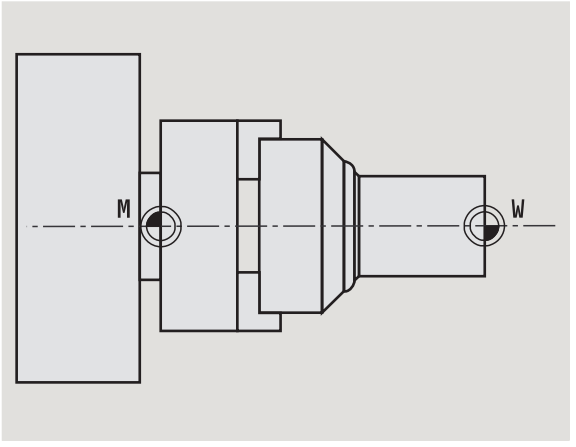
Positions located on the face or lateral surface can either be entered in Cartesian coordinates or polar coordinates.

When programming with polar coordinates, a position on the workpiece is clearly defined by the entries for diameter and angle (see figure).



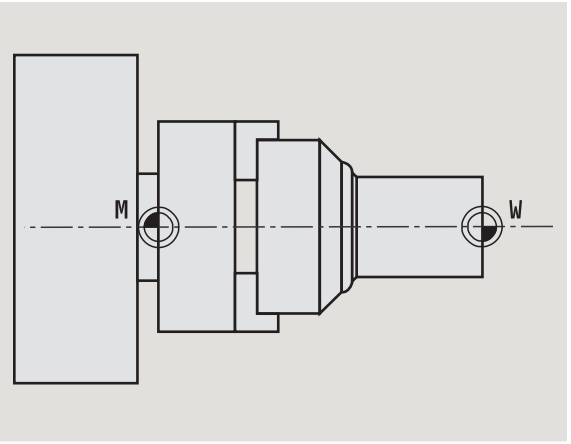
Machine zero point

The point of intersection of the X and Z axes is called the **machine zero point.** On a lathe, the machine zero point is usually the point of intersection of the spindle axis and the spindle surface. It is designated with the letter "M" (see figure).



Workpiece zero point

To machine a workpiece, it is easier to enter all input data with respect to a zero point located on the workpiece. By programming the zero point used in the workpiece drawing, you can take the dimensions directly from the drawing, without further calculation. This point is the **workpiece zero point.** It is designated with the letter "W" (see figure).



Units of measure

You can program the CNC PILOT either in the metric or inch system. The units of measurement listed in the table below apply to all inputs and displays.

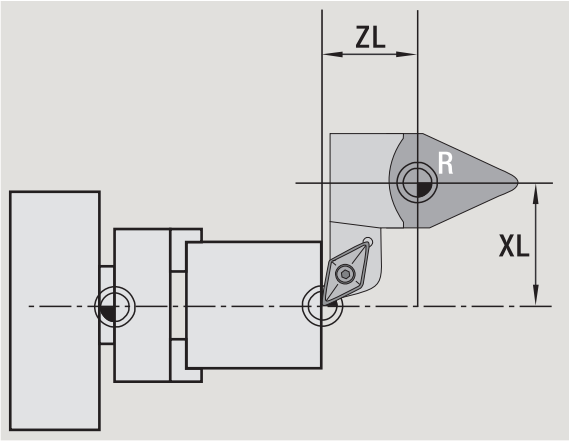
Dimensions	Metric	Inches
Coordinates	mm	inch
Lengths	mm	inch
Angle	Degrees	Degrees
Spindle speed	rpm	rpm
Cutting speed	m/min	ft/min
Feed per revolution	mm/rev	inch/rev
Feed per minute	mm/min	inch/min
Acceleration	m/s^2	ft/s^2

1.8 Tool dimensions

The CNC PILOT requires information on the specific tools for a variety of tasks, such as calculating the cutting radius compensation or the proportioning of cuts.

Tool length

All programmed and displayed position values are given with respect to the distance between the tool tip and workpiece zero point. Since the control only knows the absolute position of the tool carrier (slide), The CNC PILOT needs the dimensions XL and ZL (see figure) to calculate and display the position of the tool tip.



Tool compensation

The tool tip is subjected to wear during machining processes. To compensate for this wear, the CNC PILOT uses compensation values. The compensation values are managed independent of the values for length. The system automatically adds the compensation values to the values for length.

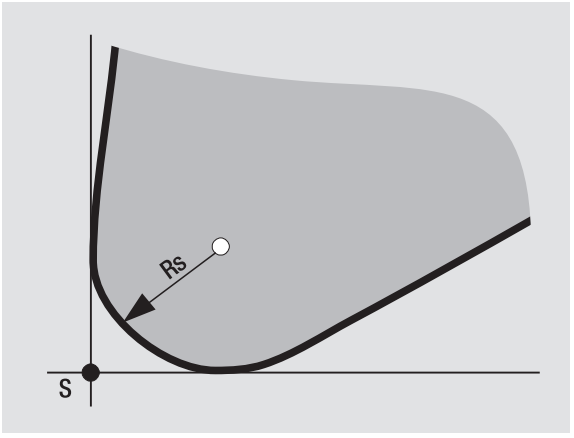
Tool-tip radius compensation (TRC)

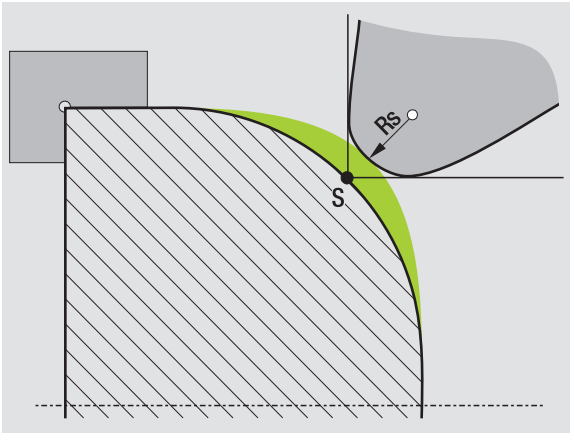
The tip of a lathe tool has a certain radius. When machining tapers, chamfers and radii, this results in inaccuracies which the CNC PILOT compensates with its cutting radius compensation function.

Programmed paths of traverse are referenced to the theoretical tool tip S. With non-paraxial contours, this will lead to inaccuracies during machining.

The TRC function compensates for this error by calculating a new path of traverse, the **equidistant line** (see figure).

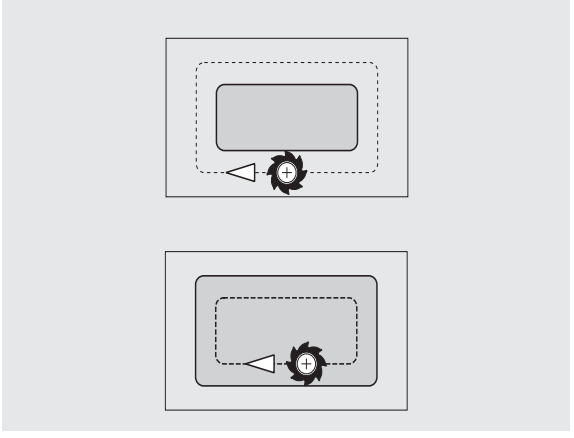
The CNC PILOT calculates the TRC for cycle programming. The smart.Turn and DIN programming feature also takes the TRC for clearance cycles into account. During DIN programming with single paths, you can also enable/disable TRC.

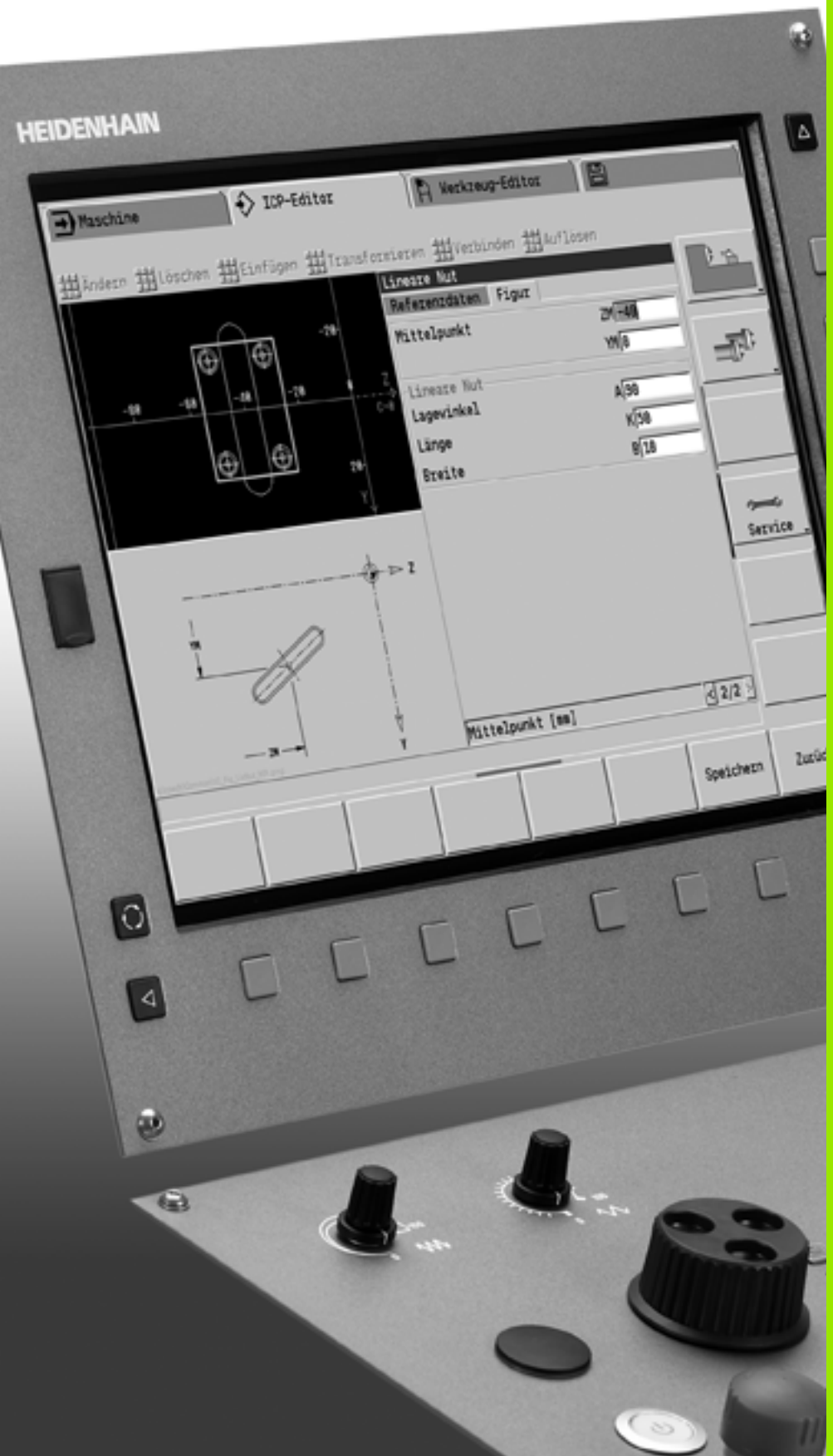




Milling cutter radius compensation (MCRC)

In milling operations, the outside diameter of the milling cutter determines the contour. When the MCRC function is not active, the system defines the center of the cutter as reference point. The TRC function compensates for this error by calculating a new path of traverse, the **equidistant line.**





2

Basics of operation

2.1 General information on operation

Operation

- Select the desired operating mode with the corresponding operating mode key.
- Within the operating mode, you can change the mode through the soft keys.
- With the numeric keypad you can select the function within the menus.
- Dialogs can consist of multiple pages.
- Besides with the soft keys, dialogs can be concluded positively with "INS" or negatively with "ESC."
- Changes made in lists are effective immediately. They are also saved if the list is closed with "ESC" or "Cancel."

Setup

- You will find all setup functions in the machine mode in "Manual mode."
- All preparatory work can be performed through the "setup" menu item and "Set S,F,T."

Programming – Teach-in mode

- Select **Teach-in** in the "machine" mode and use the **Program list** soft key to open a new cycle program.
- Activate the cycle menu through the **Add cycle** soft key. Here you select the operation and enter the details.
- Then press the **Input finished** soft key. Now you can start the simulation and check the machining process.
- Start the operation in the machine with "Cycle on."
- Save that cycle after the operation is completed.
- Repeat the last steps for each new operation.

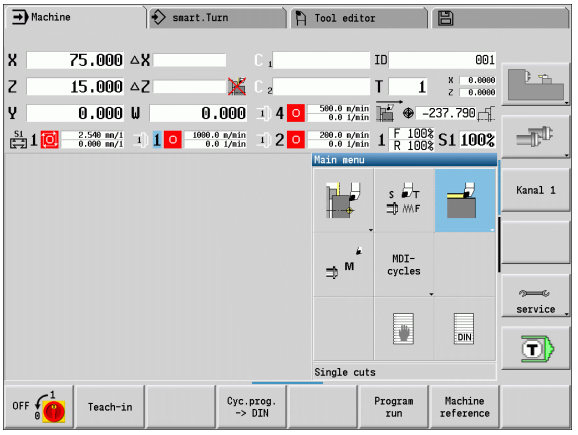
Programming – smart.Turn

- Convenient programming with UNITS in a structured NC program.
- Combinable with DIN functions.
- Contour definition is graphically possible.
- Contour follow-up when used with a workpiece blank.
- Conversion of cycle programs to smart.Turn programs with the same functions.

2.2 The CNC PILOT screen

The CNC PILOT shows the data to be displayed in **windows**. Some windows only appear when they are needed, for example, for typing in entries.

In addition, the control shows the **type of operation**, the **soft-key display** and the **PLC soft key display** on the screen. Each function that appears in a field of the soft-key row is activated by pressing the soft key directly below it.



Operating mode line
The operating mode tabs (at the top of the screen) shows the four operating modes as well as the submodes.

Machine display
The machine display field (beneath the operating mode tabs) is configurable. It shows all important information on axis positions, feed rates, rotational speeds, and tool.

Other windows used:

- **List and program window**
  Display of program lists, tool lists, parameter lists, etc. To select specific elements from the list, simply move the highlight to the desired element with the **arrow keys.**
- **Menu window**
  Display of menu symbols. This window only appears on the screen in the Teach-in and Manual modes.
- **Input window/Dialog window**
  For entering the parameters of a cycle, ICP element, DIN command, etc. Look over the existing data, then delete or edit them in the dialog window.
- **Graphic support window**
  Input data (such as cycle parameters, tool data, etc.) are explained with graphics. The **switchover key** (the key with three rotating arrows at the left edge of the screen) allows you to switch between the help graphics for internal and external machining (only for cycle programming).
- **Simulation window**
  The simulation window shows a graphic representation of the contour elements and a simulation of the tool movements. This enables you to check cycles, entire cycle programs, and DIN programs.
- **ICP contour graphics**
  Display of the contour during ICP programming.
- **DIN editing window**
  Display of the DIN program during DIN programming.
- **Error window**
  Display of occurred errors and warnings.

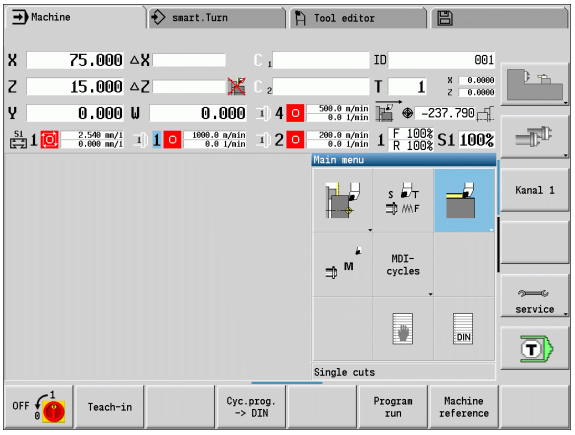
```
V:\nc_prog\ncps\WELLE.NC
N  18   G96 S200 G95 F0.5 M3
N  19   G0 X62 Z2
N  20   G47 P2
N  21   G810 ID"welle" P3 I0.5 K0.2 D0
N  22   G14 Q0
N  23   G47
N  24 END_OF_UNIT
      [// ICP-Schlichten längs]
N  25 UNIT ID"G890_ICP"
N  26   [<unit ID="G890_ICP" T="5" TID="" FK="welle...
N  27   T5
N  28   G96 S200 G95 F0.5 M3
N  29   G0 X62 Z2
N  30   G58 P0
N  31   G47 P2
N  32   G890 ID"welle" Q0 H3 D1
N  33   G14 Q0
```

2.3 Operation and data input

Operating modes

The active mode of operation is highlighted in the operating-mode tab. The CNC PILOT differentiates between the following operating modes:

- Machine—with the submodes:
  - Manual (display: "Machine")
  - Teach-in (Teach-in mode)
  - Program Run
- Programming—with the submodes:
  - smart.Turn
  - Simulation
  - ICP
  - TURN PLUS: Automatic working plan generation (AWG)
- Tool management—with the submodes:
  - Tool editor
  - Technology editor
- Organization—with the submodes:
  - User parameters
  - Transfer
  - User login



You can use the operating mode keys to switch between the modes. The selected submode and the current menu position remain during the mode change.

If you press the operating mode key in a submode, the CNC PILOT switches back to the main level of the mode.

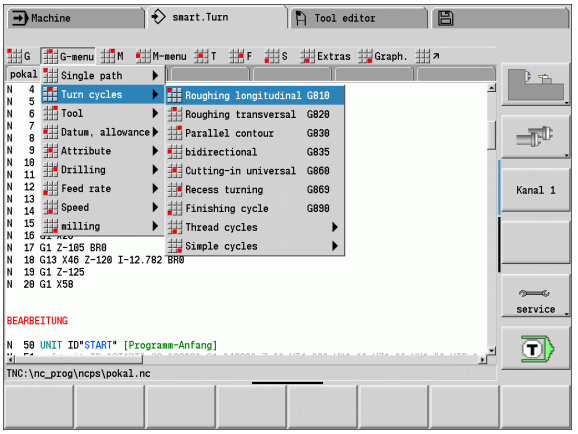
At some places, a dialog has to be ended in order to switch modes (e.g. in the tool editor).

Menu selection

The numerical keypad is used for activating a menu and for entering data. They are displayed differently depending on the operating mode.

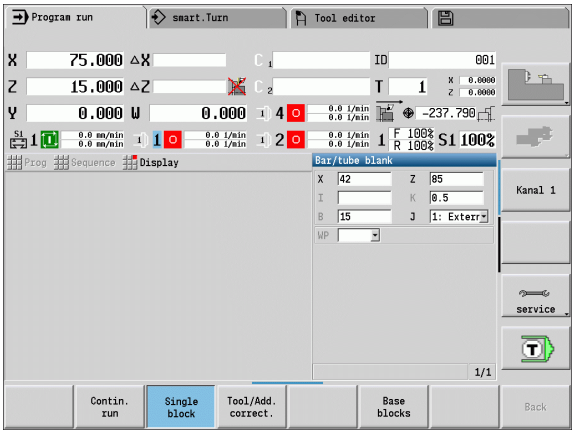
- During setup, Teach-in mode etc., the functions are shown in a 9-field box, the **menu window**. The meaning of the selected symbol / menu item is described in the footer.
- In other operating modes, the keypad symbol is shown with the position of the function marked (see figure).

Press the corresponding numerical key, or move the highlight with the arrow keys to the symbol on the screen and press the **ENT** key.



Soft keys

- With some system functions, the available functions are arranged on several soft-key levels.
- Some soft keys work like "toggle switches." A function is active when the associated field in the soft-key row is highlighted in color. The setting remains in effect until the function is switched off again.
- With functions like **Take over position** you do not have to enter values manually. The data are automatically written into the appropriate input fields.
- Data entries are not concluded until the **Save** or **Input finished** soft key has been pressed.
- The **Back** soft key takes you back to the previous operating level.



Data input

Input windows comprise several **input fields.** You can move the cursor to the desired input field with the vertical arrow keys. The CNC PILOT shows the function of the selected field in the footer of the window.

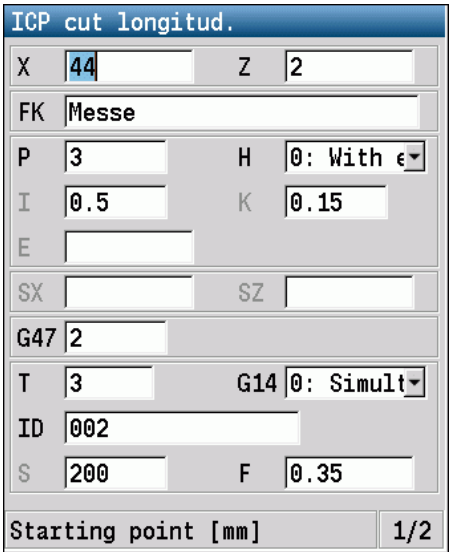
Place the highlight on the desired input field and enter the data. Existing data are overwritten. With the horizontal arrow keys, you can move the cursor **within** the input field and place it on the position where you want to delete, copy or add characters.

To confirm the data you entered in a field, press a vertical arrow key or the ENTER key.

If there are more input fields than a window can show, a second input window is used. You will recognize this through the symbol in the bottom line of the input window. To switch back and forth between the windows, press the **PG UP/PG DN keys.**



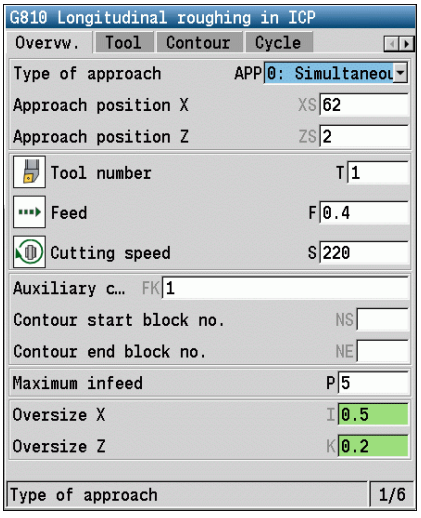
Data entry is concluded when you press the **OK** or **Input finished** or **Save** soft key. The **Back** or **Cancel** soft key discards input or changes.



smart.Turn dialogs

The unit dialog is divided into fillable forms and the forms are divided again into groups. The forms are identified by tabs and fine lines divide each tab into groups. You can navigate between the forms and groups with the **smart keys**.

smart keys	
	Go to the next form
 	Next/previous group



List operations

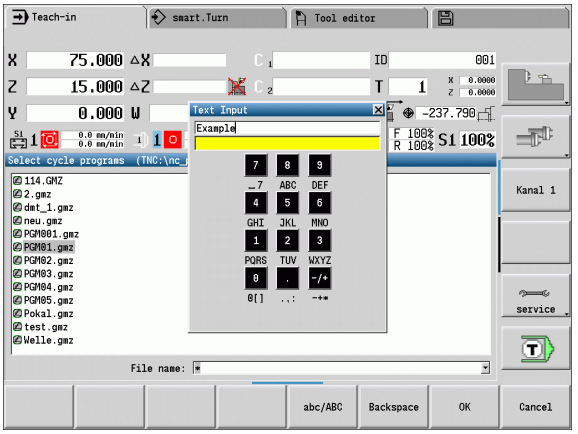
Cycle programs, DIN programs, tool lists, etc. are displayed as lists. You can scroll through a list with the arrow keys to check data or to highlight elements for operations like deleting, copying, editing, etc.

Alphanumeric keyboard

You enter letters and special characters with the screen keypad or (if available) with a PC keyboard connected over the USB port.

Enter the text with the screen keyboard

- Press the "Alphabetic keyboard" soft key or the GOTO key to enter a text, for example a program name.
- The CNC PILOT opens the "text entry" window.
- Just as on a cell phone, you press the numerical keys a few times to get the desired letters or special characters.
- Wait until the selected character is transferred to the entry field before you enter the next character.
- Use the OK soft key to load the text into the open dialog field.
- Use the **abc/ABC** soft key to select upper or lower case.
- To delete individual characters, use the Backspace soft key.

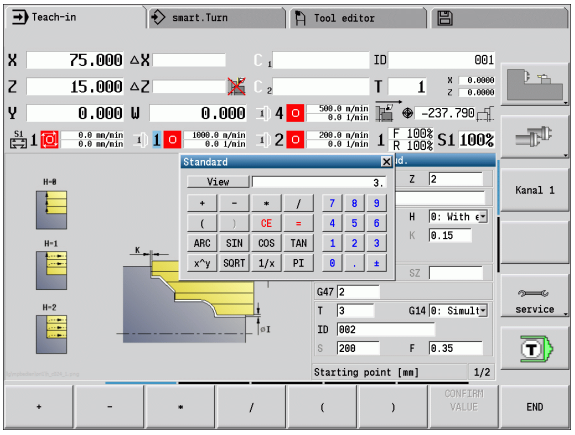


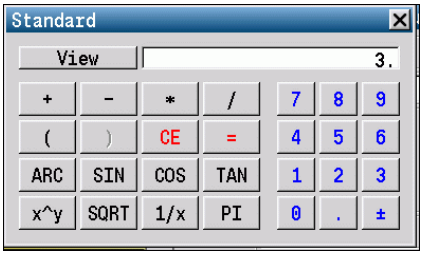
2.4 Integrated calculator

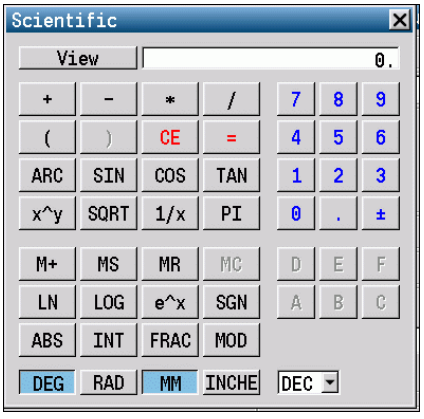
Calculator functions

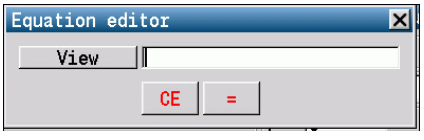
The calculator can be selected only from open dialogs in cycle programming or smart.Turn programming. You can use the calculator in the following three **views** (see figures at right):

- Scientific
- Standard
- Equation editor. Here you can type in multiple calculations in immediate sequence (for example 17*3+5/9).









> The calculator remains in effect even after a change in operating modes. Press the END soft key to close the calculator.
>
> The GET CURRENT VALUE soft key enables you to transfer a numerical value from the active input field to the calculator. The LOAD CURRENT VALUE soft key enables you to load the current value from the calculator to the active input field.

Using the calculator

▶ Use the arrow keys to select the input field.



▶ Use the **CALC** key to activate and deactivate the calculator.



▶ Shift the soft-key menu until the desired function appears.

▶ Perform the calculation.



▶ Press the soft key. The CNC PILOT transfers the value into the active input box and closes the calculator.

Switching the view of the calculator

▶ Shift the soft-key menu until the **VIEW** soft key appears.



▶ Press the **View** soft key until the desired view is set.

Mathematical function	Shortcut (soft key)
Addition	+
Subtraction	-
Multiplication	*
Division	/
Calculations in parentheses	()
Arc cosine	ARC

Mathematical function	Shortcut (soft key)
Sine	SIN
Cosine	COS
Tangent	TAN
Powers of values	X^Y
Square root	SQRT
Inversion	1/x
pi (3.14159265359)	PI
Add value to buffer memory	M+
Save the value to buffer memory	MS
Recall from buffer memory	MR
Delete buffer memory contents	MC
Natural logarithm	LN
Logarithm	LOG
Exponential function	e^x
Check the algebraic sign	SGN
Form the absolute value	ABS
Truncate decimal places	INT
Truncate places before the decimal point	FRAC
Modulus operator	MOD
Select view	View
Delete value	DEL
Unit of measure	MM or INCH
Display mode for angle values	DEG (degree) or RAD (radian measure)
Display mode of the numerical value	DEC (decimal) or HEX (hexadecimal)

Adjusting the position of the calculator

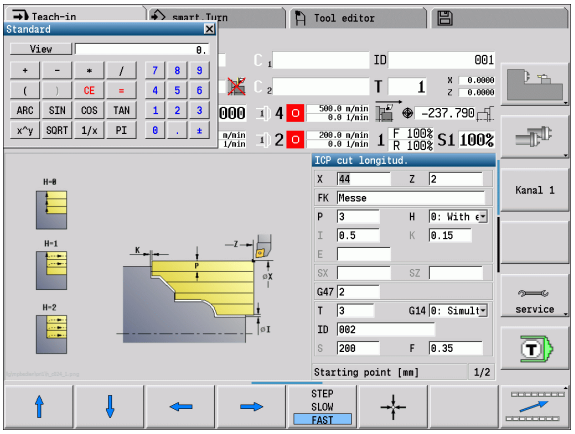
You can move the calculator as follows:

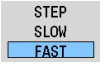
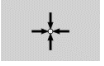
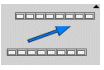


- Shift the soft-key menu until the **Additional functions** soft key appears.



- Select "additional functions."
- Position the calculator with the soft keys (see table at right).



Soft keys for positioning the calculator	
	Move window in the direction of the arrow
	
	Adjust the increment for movement
	Move window to center
	Back by one menu level

2.5 Types of programs

The CNC PILOT supports the following programs/contours:

- **Teach-in programs** (cycle programs) are used in the "Teach in" mode of operation.
- **smart.Turn** and **DIN main programs** are written in the smart.Turn mode of operation.
- **DIN subprograms** are written in the smart.Turn operating mode and are used in cycle programs and smart.Turn main programs.
- **ICP contours** are generated during Teach-in in the Teach-in or Manual mode of operation. The extension depends on the contour described.

In smart.Turn the contours are saved directly in the main program.

Program type	Folders	Extension
Teach-in programs (cycle programs)	"nc_prog\gtz"	"*.gmz"
smart.Turn and DIN main programs	"nc_prog\ncps"	"*.nc"
DIN subprograms	"nc_prog\ncps"	"*.ncs"
ICP contours	"nc_prog\gti"	
Turning contours		"*.gmi"
Contours of workpiece blanks		"*.gmr"
Contours on face		"*.gms"
Lateral surface contours		"*.gmm"

2.6 The error messages

Display of errors

The CNC PILOT generates error messages when it detects problems such as:

- Incorrect data input
- Logical errors in the program
- Contour elements that are impossible to machine

When an error occurs, it is displayed in red type in the header. Long and multi-line error messages are displayed in abbreviated form. If an error occurs in a background mode, the error symbol is shown in the operating mode tab. Complete information on all pending errors is shown in the error window.

If a rare "processor check error" should occur, the CNC PILOT automatically opens the error window. You cannot remove such an error. Shut down the system and restart the CNC PILOT.

The error message is displayed in the header until it is cleared or replaced by a higher-priority error.

An error message that contains the block number of an NC program was caused by an error in the indicated block or in the preceding block.

Opening the error window



▶ Press the ERR key. The CNC PILOT opens the error window and displays all accumulated error messages.

Closing the error window



▶ Press the END soft key—or



▶ Press the ERR key. The CNC PILOT closes the error window.

Detailed error messages

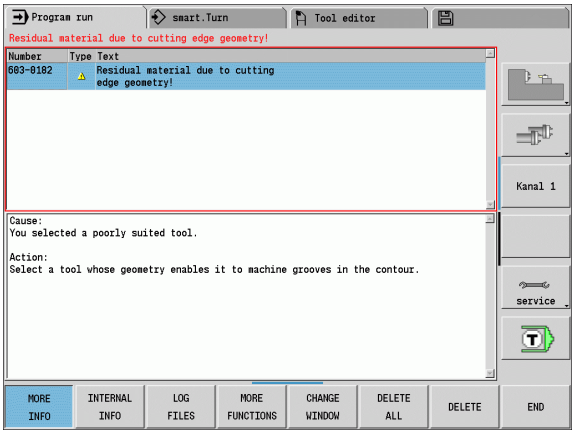
The CNC PILOT displays possible causes of the error and suggestions for solving the problem:

Information on error causes and remedies:

- Open the error window.

Info

- Position the cursor on the error message and press the soft key. The CNC PILOT opens the window with information on the error cause and corrective action.
- To exit the info, press the **Info** soft key again.



"Details" soft key

The **DETAILS** soft key supplies information on the error message. This information is only required if servicing is needed.

- Open the error window.

Details

- Position the cursor on the error message and press the soft key. The CNC PILOT opens the window with internal information about the error.
- To leave Details, press the **DETAILS** soft key again.

Clearing errors

Clearing errors outside of the error window:

▶ Open the error window.



▶ To clear the error/note in the header: Press the CE button.

In some operating modes (such as the Editing mode), the CE button cannot be used to clear the error, since the button is reserved for other functions.

Clearing more than one error:

▶ Open the error window.

Delete

▶ To delete individual errors: Position the cursor on the error message and press the soft key.

Delete all

▶ To delete all errors: Press **Delete all**.

If the cause of the error has not been removed, the error message cannot be deleted. In this case, the error message remains in the window.

Error log file

The CNC PILOT stores errors and important events (e.g. system startup) in an error log file. The capacity of the error log file is limited. If the log file is full, it switches to the next one, etc. If the last log file is full, the first one is overwritten by a new one, etc. If necessary, switch the log file to see the history. The are 5 log files available.

▶ Open the error window.

Log

▶ Press the **Log file** soft key.

Error log

▶ Open the log file.

Previous file

▶ Select previous log file, if needed.

Current file

▶ Select current log file, if needed.

The oldest entry is at the beginning of the log file, and the most recent entry is at the end.

Keystroke log file

The CNC PILOT stores keystrokes and important events (e.g. system startup) in the keystroke log file. The capacity of the keystroke log file is limited. If the log file is full, it switches to the next one, etc. If the last log file is full, the first one is overwritten by a new one, etc. If necessary, switch the log file to see the history. 10 log files are available.

▶ Open the keystroke log file.



▶ Press the **Log file** soft key.



▶ Open the log file.



▶ Select previous log file, if needed.



▶ Select current log file, if needed.

The CNC PILOT saves each key pressed during operation in the keystroke log file. The oldest entry is at the beginning of the log file, and the most recent entry is at the end.

Saving service files

If necessary, you can save the "Current status of the CNC PILOT," and make it available to a service technician for evaluation. A group of service files is saved that contain information about the current status of the machine and the machining. See "Service files" on page 562.

The information is summarized in a service files data record as a zip file.

TNC:\SERVICEx.zip

The "x" designates a consecutive serial number. The CNC PILOT always generates the service file with the number 1, and all existing files are renamed to the numbers 2 to 5. An existing file with the number 5 is deleted.

Saving service files

▶ Open the error window.



▶ Press the **Log file** soft key.



▶ Press the **Service Files** soft key.

2.7 TURNguide context-sensitive help system

Application

Before you can use the TURNguide, you need to download the help files from the HEIDENHAIN home page (see "Downloading current help files" on page 69).

The **TURNguide** context-sensitive help system includes the user documentation in HTML format. The TURNguide is called with the Info key, and the TNC often immediately displays the information specific to the condition from which the help was called (context-sensitive call). Even if you are editing in a cycle and press the Info key, you are usually brought to the exact place in the documentation that describes the corresponding function.

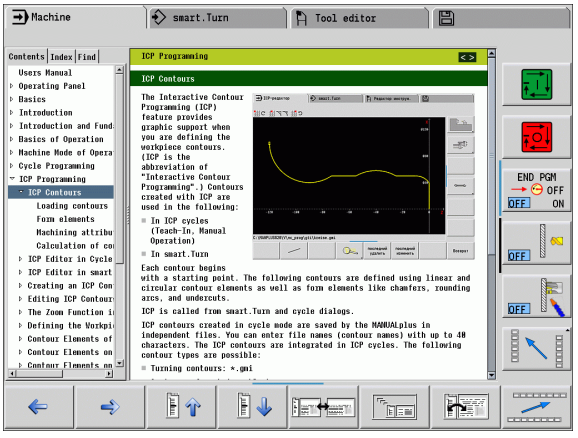
The control always tries to start the TURNguide in the language that you have selected as the conversational language on your control. If the files with this language are not yet available on your control, it automatically opens the English version.

The following user documentation is available in the TURNguide:

- User's Manual (**BHBoperating.chm**)
- smart.Turn and ISO Programming (**smartTurn.chm**)
- List of All Error Messages **(errors.chm)**

In addition, the **main.chm** "book" file is available, with the contents of all existing .chm files.

As an option, your machine tool builder can embed machine-specific documentation in the **TURNguide**. These documents then appear as a separate book in the **main.chm** file.

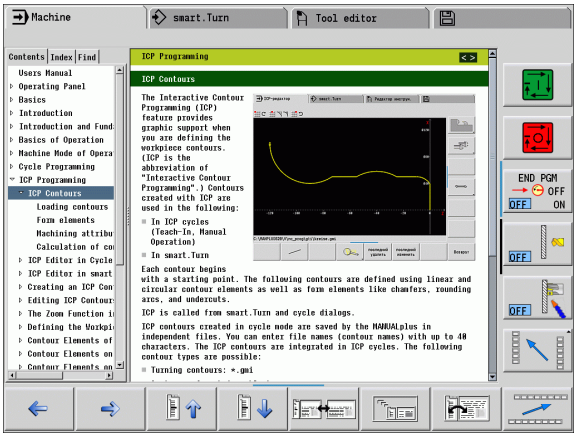


Working with the TURNguide

Calling the TURNguide

There are several ways to start the TURNguide:

- Press the Info key if the control is not already showing an error message
- Click the help symbol at the lower right of the screen beforehand, then click the appropriate soft keys



If one or more error messages are waiting for your attention, the control shows the help directly associated with the error messages. To start the **TURNguide,** you first have to acknowledge all error messages.

When the help system is called on the programming station, the control starts the internally defined standard browser (usually the Internet Explorer), or otherwise an adapted browser.

For many soft keys there is a context-sensitive call through which you can go directly to the description of the soft key's function. This functionality requires using a mouse. Proceed as follows:

- Select the soft-key row containing the desired soft key
- Click with the mouse on the help symbol that the control displays just above the soft-key row: The mouse pointer turns into a question mark.
- Move the question mark to the soft key for which you want an explanation, and click: The control opens the TURNguide. If no specific part of the help is assigned to the selected soft key, the control opens the book file **main.chm,** in which you can use the search function or the navigation to find the desired explanation manually.

Even if you are editing a cycle, context-sensitive help is available:

- Select any cycle.
- Press the Info key: The control start the help system and shows a description for the active function (does not apply to miscellaneous functions or cycles that were integrated by your machine tool builder)

Navigating in the TURNguide

It's easiest to use the mouse to navigate in the TURNguide. A table of contents appears on the left side of the screen. By clicking the rightward pointing triangle you open subordinate sections, and by clicking the respective entry you open the individual pages. It is operated in the same manner as the Windows Explorer.

Linked text positions (cross references) are shown underlined and in blue. Clicking the link opens the associated page.

Of course you can also operate the TURNguide through keys and soft keys. The following table contains an overview of the corresponding key functions.



The key functions described below are only available on the control hardware, and not on the programming station.

Function	Soft key
■ If the table of contents at left is active: Select the entry above it or below it ■ If the text window at right is active: Move the page downward or upward if texts or graphics are not shown completely	 
■ If the table of contents at left is active: Open a branch of the table of contents. If the branch is at its end, jump into the window at right ■ If the text window at right is active: No function	
■ If the table of contents at left is active: Close a branch of the table of contents ■ If the text window at right is active: No function	
■ If the table of contents at left is active: Use the cursor key to show the selected page ■ If the text window at right is active: If the cursor is on a link, jump to the linked page	
■ If the table of contents at left is active: Switch the tab between the display of the table of contents, display of the subject index, and the full-text search function and switching to the screen half at right ■ If the text window at right is active: Jump back to the window at left	
■ If the table of contents at left is active: Select the entry above it or below it ■ If the text window at right is active: Jump to the next link	 

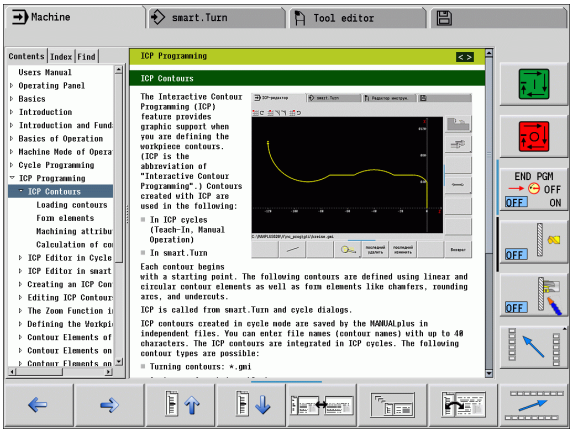
Function	Soft key
Select the page last shown	
Page forward if you have used the "select page last shown" function	
Move up by one page	
Move down by one page	
Display or hide table of contents	
Switch between full-screen display and reduced display. With the reduced display you can see some of the rest of the control window.	
The focus is switched internally to the control application so that you can operate the control when the TURNguide is open. If the full screen is active, the control reduces the window size automatically before the change of focus.	
Exit TURNguide	

Subject index

The most important subjects in the Manual are listed in the subject index (**Index** tab). You can select them directly by mouse or with the cursor keys.

The left side is active.

- Select the **Index** tab
- Activate the **Keyword** input field
- Enter the word for the desired subject and the control synchronizes the index and creates a list in which you can find the subject more easily, or
- Use the arrow key to highlight the desired keyword
- Use the ENT key to call the information on the selected keyword



You can enter the search word only with a keyboard connected via USB.

Full-text search

In the **Find** tab you can search the entire TURNguide for a specific word.

The left side is active.

- Select the **Find** tab
- Activate the **Find:** input field
- Enter the desired word and confirm with the ENT key: the control lists all sources containing the word
- Use the arrow key to highlight the desired source
- Press the ENT key to go to the selected source

You can enter the search word only with a keyboard connected via USB.

The full-text search only works for single words.

If you activate the **Search only in titles** function (by mouse or by using the cursor and the space key), the control searches only through headings and ignores the body text.

Downloading current help files

You'll find the help files for your control software on the HEIDENHAIN homepage **www.heidenhain.de**. Help files for most conversational language are at:

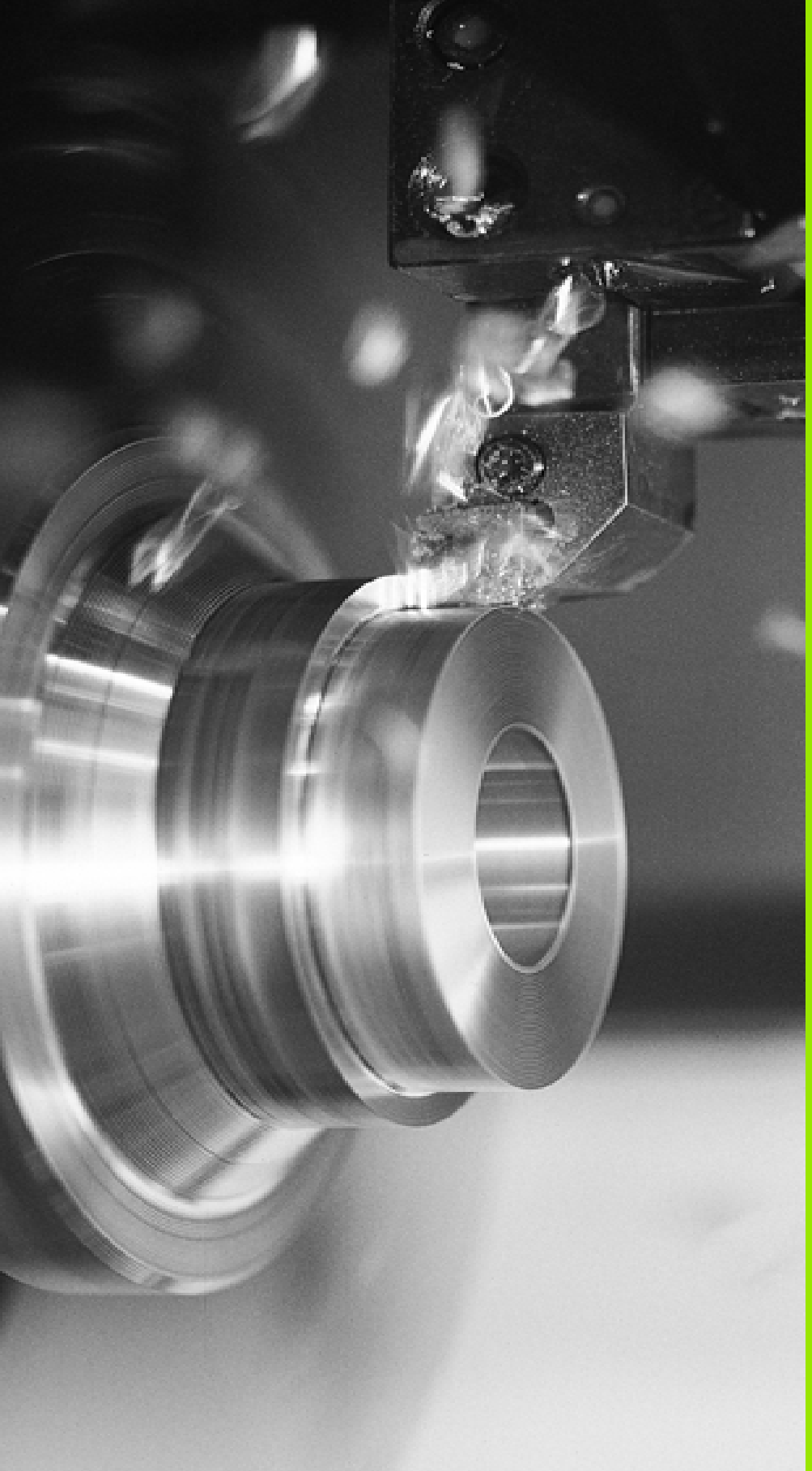
- Services and Documentation
- Software
- CNC PILOT help system
- NC software number of your control, for example **34056x-02**
- Select the desired language, for example English: You will see a ZIP file with the appropriate help files
- Download the ZIP file and unzip it
- Move the unzipped CHM files to the control in the **TNC:\tncguide\en** directory or into the respective language subdirectory (see also the following table)



If you want to use TNCremoNT to transfer the CHM files to the control, then in the **Extras**>**Configuration**>**Mode**>**Transfer in binary format** menu item you have to enter the extension **.CHM**.

Language	TNC directory
German	**TNC:\tncguide\de**
English	**TNC:\tncguide\en**
Czech	**TNC:\tncguide\cs**
French	**TNC:\tncguide\fr**
Italian	**TNC:\tncguide\it**
Spanish	**TNC:\tncguide\es**
Portuguese	**TNC:\tncguide\pt**
Swedish	**TNC:\tncguide\sv**
Danish	**TNC:\tncguide\da**
Finnish	**TNC:\tncguide\fi**
Dutch	**TNC:\tncguide\nl**
Polish	**TNC:\tncguide\pl**
Hungarian	**TNC:\tncguide\hu**
Russian	**TNC:\tncguide\ru**
Chinese (simplified)	**TNC:\tncguide\zh**
Chinese (traditional)	**TNC:\tncguide\zh-tw**

Language	TNC directory
Slovenian (software option)	**TNC:\tncguide\sl**
Norwegian	**TNC:\tncguide\no**
Slovak	**TNC:\tncguide\sk**
Latvian	**TNC:\tncguide\lv**
Korean	**TNC:\tncguide\kr**
Estonian	**TNC:\tncguide\et**
Turkish	**TNC:\tncguide\tr**
Romanian	**TNC:\tncguide\ro**
Lithuanian	**TNC:\tncguide\lt**



3

Machine mode of operation

3.1 Machine mode of operation

The Machine mode of operation includes all functions for machine setup, workpiece machining, and Teach-in program definition.

- **Machine setup:** For preparations like setting axis values (defining workpiece zero point), measuring tools or setting the protection zone.
- **Manual mode:** Machine a workpiece manually or semi-automatically.
- **Teach-in mode:** "Teach-in" a new cycle program, change an existing program, or graphically simulate cycles.
- **Program run:** Graphically simulate existing cycle programs or smart.Turn programs and use them for the production of parts.

A **Teach-in cycle** is a machining step that has already been programmed for you. This can be any machining operation from a single cut through to a complex machining task like thread cutting. In any case, a cycle is always a complete machining step that is immediately executable once you have defined a few parameters that describe the workpiece to be machined.

In Manual mode, the cycles that you program are **not stored.** In Teach-in mode, each machining step is executed with a cycle and then stored and integrated into a complete **Teach-in program.** You can subsequently use this **program** in parts production by repeating it as often as desired in the Program Run mode.

In **ICP programming,** any contour can be defined using linear/circular elements and transition elements (chamfers, rounding arcs, undercuts). You include the contour description in ICP cycles (see "ICP contours" on page 360).

You write **smart.Turn** and **DIN programs** in the smart.Turn mode of operation. The DIN programming feature provides your with commands for simple traversing movements, DIN cycles for complex machining tasks, switching functions, mathematical operations and programming with variables.

You can either create "independent" programs that already contain all necessary switching and traversing commands and are executed in the Program Run mode, or program **DIN subprograms** that are integrated in Teach-in cycles. The commands that you use in a DIN subprogram depend on the job at hand. DIN subprograms support the complete range of commands that is available for DIN programs.

You can also **convert** Teach-in programs to smart.Turn programs. This enables you to make use of straightforward Teach-in programming, and then convert the part program to DIN format for subsequent optimization or completion.

3.2 Switch-on / Switch-off

Switch-on

The CNC PILOT displays the startup status. When the system has completed all tests and initializations, it switches to the Machine mode of operation. The tool display shows the tool that was last used.

If errors are encountered during system start, the control displays the **error symbol** on the screen. You can check these error messages as soon as the system is ready (see "The error messages" on page 60).



After system start, the CNC PILOT assumes that the tool which was last used is still inserted in the tool holder. If this is not the case, you must inform the control of the tool change.

Monitoring EnDat encoders

If EnDat encoders are used, the control saves the axis positions during switch-off of the machine. During switch-on, the CNC PILOT compares for each axis the position during switch-on with the position saved during switch-off.

If there is a difference, one of the following messages appears:

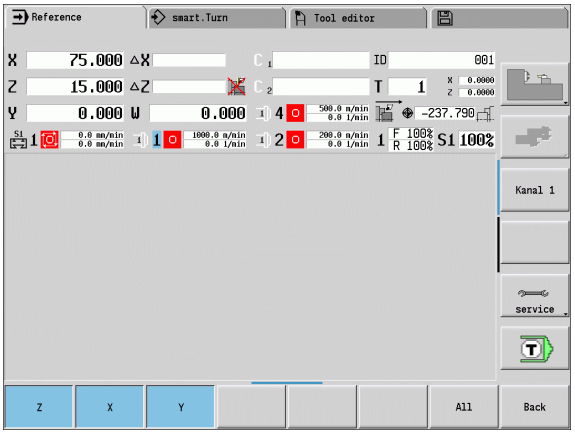
- "S-RAM error: Saved position of the axis is invalid."
  This message is correct if the control has been switched on for the first time, or if the encoder or other control components involved were exchanged.
- "Axis was moved after power-off. Position difference: xx mm or degrees"
  Check the current position and confirm it if the axis was in fact moved.
- "HW parameter changed: Saved position of the axis is invalid."
  This message is correct if configuration parameters were changed.

The cause for one of the above listed messages can also be a defect in the encoder or control. Please contact your machine supplier if the problem recurs.

Traversing the reference marks

Whether a **reference run** is necessary depends on the encoders used:

- EnDat encoder: Reference run is not necessary.
- Distance-coded encoders: The position of the axes is ascertained after a short reference run.
- Standard encoder: The axes move to known, machine-based points. As soon as a reference mark is traversed, a signal is transmitted to the control. The control knows the distance between the reference mark and the machine zero point and can now establish the precise position of the axis.



REFERENCE RUN

Z

Press the **Z reference** soft key

X

Press the **X reference** soft key

All

Or press **all** soft keys

Press **Cycle start** for the control to traverse the reference marks

The CNC PILOT activates the position display and switches to the **main menu**.

In case you traverse the reference marks separately for the X and Z axes, you only traverse in either the X or the Z axis.

Switch-off

Proper switch-off is recorded in the error log file.

SWITCH-OFF

Go to the main level of the Machine mode of operation

Activate the error window

Additional functions

Press the **MORE FUNCTIONS** soft key

OFF

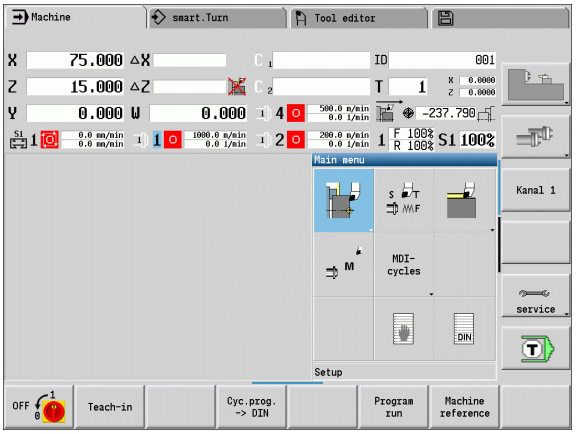
Press the **OFF** soft key

The CNC PILOT displays a confirmation request.

YES

Press the **Enter key** or the **YES** soft key. The software shuts down

Wait until the CNC PILOT requests you to switch off the machine.



3.3 Machine data

Input of machine data

In Manual mode, you enter the information for tool, spindle speed and feed rate/cutting speed in the TSF dialog box (**Set T, S, F** input window). In Teach-in programs the tool information and technology data are included in the cycle parameters, and in smart.Turn programs they are part of the NC program.

In the T, S, F dialog box you also define the "maximum speed" and the "stopping angle."

You can save the cutting data (cutting speed, feed rate) in the technology database as a function of the workpiece material, the tool's cutting material and the type of operation. With the **Proposed technology** soft key the data are taken into the dialog.

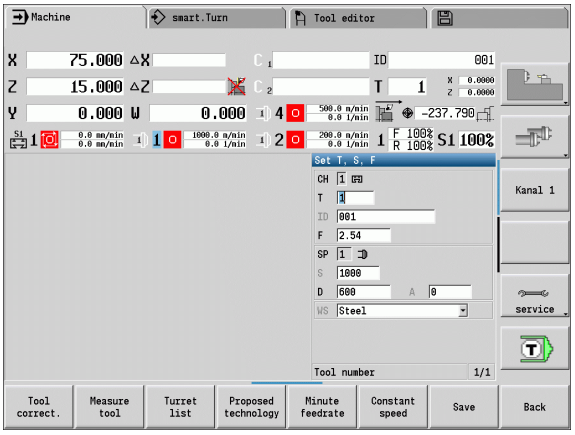
The **Tool list** soft key opens the tool list (turret list). This list indicates the current assignment of the tool carrier. There is a place in the table for every tool holder. During setup, each tool (ID number) is assigned to a tool holder.

If your machine is equipped with a driven tool, you use the spindle-change key to select the spindle for which the your entries are to apply. The selected spindle is indicated in the display. For this reason there are two versions of the TSF dialog box:

- **Without driven tool** (shown in upper figure): The parameters S, D and A apply to the main spindle
- **With driven tool** (shown in lower figure): The parameters S, D and A apply to the selected spindle

Meaning of the parameters:

- S: Spindle speed / constant speed
- D: Max. spindle speed
- A: Stopping angle
- BW: Angle in the B axis (machine-dependent function)
- CW: C tilting plane angle: Position of the C axis to determine the work position of the tool (machine-dependent function)



ENTER THE TOOL DATA AND TECHNOLOGY DATA



Select **Set T, S, F** (only available in Manual mode)

Define the parameters

Save

Conclude data input



Caution. Depending on the machine, this operation might cause the turret to turn.

Selecting the workpiece spindle (machine-dependent)

If your machine is equipped with a workpiece spindle, the WP parameter is shown in the TSF form. The WP parameter allows you to select the workpiece spindle for machining in Teach-in mode and MDI.

Select the workpiece spindle for machining with **WP**:

- Main drive
- Opposing spindle for rear-face machining

The WP parameter setting is saved in the Teach-in and MDI cycles and displayed in the corresponding cycle form.

If you selected the opposing spindle for rear-face machining with the WP parameter, the cycle is mirrored (in the opposite Z direction). Use tools with suitable tool orientation.



In the TSF menu, the setting for the WP parameter is changed when you:

- run a cycle with another WP parameter setting
- select a program during program run

Soft keys for "Set T, S, F"

Tool correct.	See "Tool compensation" on page 103.
Measure tool	See "Touch off" on page 100.
Tool list	Call the "tool list": Transfer of T number from the tool list: See "Setting up a tool list" on page 83.
Proposed technology	Transfer of cutting speed and feed rate from the technology data
Minute feedrate	■ **On:** Feed per minute (mm/min) ■ **Off:** Feed per revolution (mm/rev)
Constant speed	■ **On:** Constant speed (rpm) ■ **Off**: Constant surface speed (m/min)

Machine data display

Elements of machine data display

Position display X, Y, Z, W: Distance between tool tip and workpiece zero point

- A black axis letter means the axis is enabled; white means it is not enabled.



Handwheel active



Clamping active



Position display C: Position of the C axis

- Empty box: C-axis is not active
- A black axis letter means the axis is enabled; white means it is not enabled.

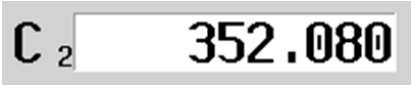


Display settings of the position display: Can be set via the MP_axesDisplayMode user parameter. The setting is shown by a letter next to the position window.

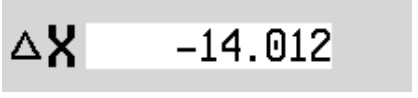


- A: Actual value (setting: REFIST)
- N: Nominal value (setting: REFSOLL)
- L: Following error (setting: SCHPF)
- D: Distance to go (setting: RESTW)

Display of the slide number and C-axis number: A numeral next to the position window of the axis shows the assigned slide or C-axis number. The numeral is only displayed if an axis was configured multiple times, e.g. a second C-axis as opposing spindle.



Distance-to-go display X, Y, Z, W: The distance remaining from the current position to the target position of the active traversing command



Distance-to-go and protection zone status: Distance-to-go display and display of status of protective zone monitoring



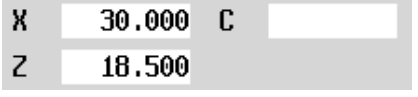
Protective zone monitoring active



Protective zone monitoring not active



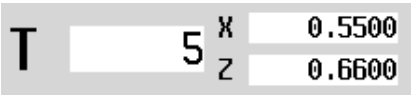
Position display for four axes: Display of position values for up to four axes. The displayed axes depend on the machine configuration.



Elements of machine data display

T number display

- T number of the inserted tool
- Tool compensation values

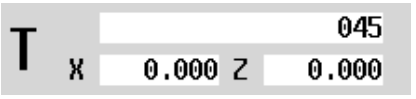


For all of T displays:

- T highlighted in color indicates a driven tool
- T number or ID highlighted in color indicates a mirrored tool holder
- A letter X/Z of the compensation highlighted in color indicates that a special compensation is active in the X/Z direction

T ID display

- ID of the inserted tool
- Tool compensation values



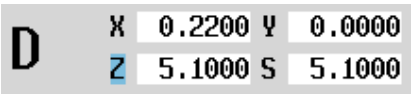
T ID display without compensation values

- ID of the inserted tool



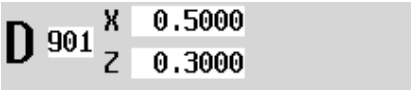
Tool compensation

- Special compensation only for recessing tools or button tools
- Special compensation value in gray means special compensation is not active
- A letter X/Z of the compensation highlighted in color indicates that a special compensation is active in the X/Z direction



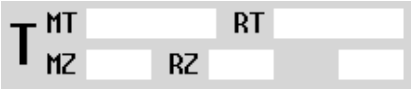
Additive compensation

- Compensation values in gray means D compensation is not active
- Compensation values in black means D compensation is active



Tool life information

- "T": Black=global tool life monitoring on; white=global tool life monitoring off
- MT, RT active: Monitoring according to tool life
- MZ, RZ active: Monitoring according to part quantity
- All fields empty: Tool without tool-life monitoring



Slide display and cycle status

- Upper field: Setting of the override control
- Lower field with white background: Actual feed rate
- Lower field with gray background: Programmed feed rate with stationary slide



Slide display and cycle status

- Upper field: Programmed feed rate
- Lower field: Actual feed rate



Elements of machine data display

Slide display and cycle status



- Upper field: Setting of the override control
- Middle field: Programmed feed rate
- Lower field: Actual feed rate

Slide display with rear-face machining



- If rear-face machining is enabled, the slide number is highlighted in blue.

Spindle display with spindle number, gear range and spindle status



- Upper field: Setting of the override control
- Lower field: Actual speed or spindle position

For all of the spindle displays:

- Spindle symbol: Black means the spindle is enabled; white means it is not.
- Numeral in spindle symbol: Gear range
- Numeral at right next to the spindle symbol: Spindle number
- If a spindle key exists, the number of the selected spindle is highlighted in color.
- Spindle status: See "Spindle" on page 82.
- Display of the programmed speed in rpm or m/min
- Display of the actual speed in rpm
- If M19 is active and the machine tool builder has made the setting, when the spindle is not turning, the display shows the spindle position instead of the spindle speed.
- If a spindle is in slave mode during synchronous operation, the value "0" is displayed instead of the programmed speed.

Spindle display with spindle number, gear range and spindle status



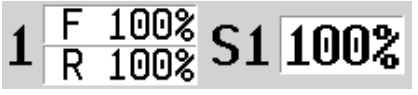
- Upper field: Programmed speed
- Lower field: Actual speed or spindle position

Spindle display with spindle number, gear range and spindle status



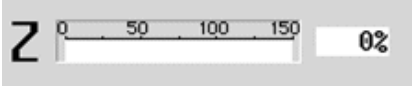
- Upper field: Setting of the override control
- Middle field: Programmed speed
- Lower field: Actual speed or spindle position

Override display of the active spindle

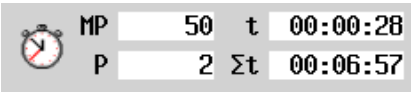
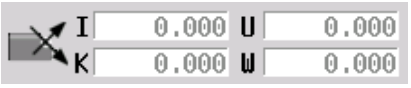


- **F**: Feed rate
- **R**: Rapid traverse
- **S**: Spindle

Motor utilization: Utilization of the drive motor with respect to the rated torque



- Digital axis and spindle motors
- Analog axis and spindle motors, if set up by the machine tool builder

Elements of machine data display	
Display of quantity: The quantity is incremented after each M30, M99 or M18 programmed counter pulse. ■ MP: Default unit quantity ■ P: Number of finished parts	
Display of quantity and job time: The quantity is incremented after each M30, M99 or M18 programmed counter pulse. ■ MP: Default unit quantity ■ P: Number of finished parts ■ t: Run time of the current program ■ Sum t: Total time	
Display of rear-face machining: Information about rear-face machining is shown in the RSM display (RSM: **R**ear **S**ide **M**achining). ■ RSM status ■ Active datum shift of the configured RSM axis	 
B-axis display: The information displayed about the status of the tilted plane varies depending on the setting of the machine parameters. ■ Programmed angular value of the B axis ■ Display of the current values I, K, U and W ■ I: Plane reference in X ■ K: Plane reference in Z ■ U: Shift in X ■ W: Shift in Z	 

The machine data display is configurable by the machine tool builder. The machine data that appear on your screen may therefore deviate from the example shown.

Cycle statuses

The CNC PILOT shows the current cycle status with the cycle symbol (see table at right).

Cycle symbols	
Status "Cycle ON" Cycle or program execution is active.	
Status "Cycle OFF" Cycle or program execution is not active.	

Axis feed rate

F is the identification letter for feed data. Depending on which mode of the **Feed rate** soft key is active, data is entered in:

- Millimeters per spindle revolution (feed per revolution)
- Millimeters per minute (feed per minute).

On the screen, you can tell the type of feed rate from the unit of measure in the input field.

You can change the feed value with the **feed compensation controller** (feed override) (range: 0 % to 150 %).

Spindle

S is the identification letter for spindle data. Depending on which mode of the **Constant speed** soft key is active, data is entered in:

- Revolutions per minute (constant speed)
- Meters per minute (constant surface speed).

The input range is limited by the maximum spindle speed. You define the speed limitation in the **Set T, S, F,** dialog box or in DIN programming with the G26 command. The speed limit remains in effect until a new speed limit value is programmed.

The speed compensation controller (speed override) allows you to change the spindle speed (range: 50 % to 150 %).

- If you are machining with a constant cutting speed, the CNC PILOT calculates the spindle speed from the position of the tool tip. The smaller the diameter of the tip, the higher the spindle speed. The **maximum spindle speed,** however, is never exceeded.
- The spindle symbols indicate the direction of spindle rotation as seen from the point of view of the machinist.
- The spindle designation is fixed by the machine tool builder (see table at right).

Spindle symbols (S display)	
Direction of spindle rotation M3	
Direction of spindle rotation M4	
Spindle stopped	
Spindle position-controlled (M19)	
C axis on spindle motor is active	

Spindle designations			
Spindle	**H**	**0**	**1**
Driven tool	**1**	**1**	**2**

3.4 Setting up a tool list

Machine with turret

The tools used are listed in the turret list. The ID number of the mounted tool is assigned to every tool holder in the turret.

In the Teach-in cycle you program the turret position as **T number**. The **tool ID number** is automatically entered under "ID."

The turret list can be set up through the **TSF menu** or directly from the cycle dialogs in the Teach-in mode.

- **T turret pocket number**
- **Tool ID** (name) is entered automatically.

Tool list

▶ Open the **Turret list**. If the cursor is on the ID input field, the CNC PILOT also automatically opens the **tool list** with the entries in the tool database.

Machine with multifix

Machines with multifix tool holders have one tool pocket in which the tools are changed manually.

- **T turret pocket number:** Always T1
- **Tool ID** (name): Select the ID number from the tool list

Tool list

▶ Open the **Tool list**.

The turret and multifix tool systems can be used together on one machine. The **machine tool builder** defines the number of the multifix pocket.

Tools in different quadrants

Example: The **principal tool carrier** of your lathe is in front of the workpiece (standard quadrant). An **additional tool holder** is behind the workpiece.

When CNC PILOT is configured, it is defined for each tool holder whether the X dimensions and the direction of rotation of circular arcs are mirrored. In the above-mentioned example the additional tool holder is assigned the attribute "mirrored."

If this method is used, all machining operations are programmed as usual—regardless of which tool holder executes the operation. The simulation also shows all machining operations in the standard quadrant.

The tools are also described and dimensioned for the standard quadrant—even if they are inserted in the additional tool holder.

Mirroring does not become effective until the machining of the workpiece, i.e. when the additional tool holder is executing the machining operation.

Filling the turret list from the database

The turret list indicates the current assignment of the tool carrier. The turret list can be set up through the **TSF menu** or directly from the cycle dialogs in the Teach-in mode.

Look at the entries in the tool database in order to move entries from the database into the turret assignment list. The CNC PILOT displays the database entries in the lower area of the screen. The cursor keys are active in this list.

TRANSFERRING TOOLS FROM THE DATABASE

Tool list

Press the **Tool list** soft key (when turret assignment is open) to open the database.

Pocket down

Select the position in the turret assignment list.

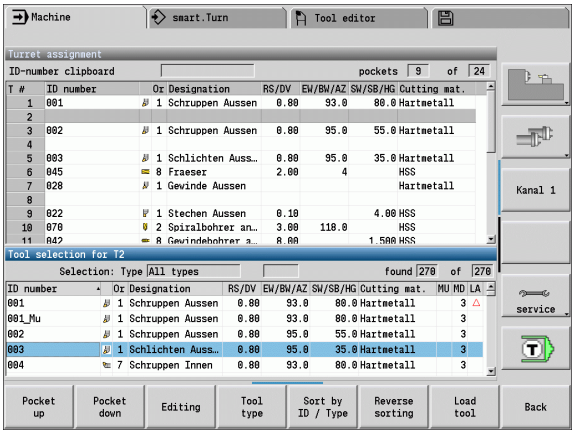
Pocket up

Select and sort the entries in the tool database (see soft-key table at right)

Use the cursor keys to select the entry in the tool database.

Load tool

Load the selected tool into the turret assignment list.



Selecting and sorting entries in the tool database

Tool type	The CNC PILOT opens the **soft-key menu** to select the desired tool type.
Sort by ID / Type	Sorts the tools in the displayed list as desired according to: ■ Tool type ■ Tool ID ■ Tool orientation Each time the soft key is pressed it changes to the next sorting mode.
Reverse sorting	Switches between ascending and descending sorting.
Editing	Not active here
Back	**Closes** the tool list.

Filling the turret list

The turret assignment indicates the current assignment of the tool carrier. When you set up a turret list, you enter the ID numbers of the tools.

The turret list can be set up through the **TSF menu** or directly from the cycle dialogs in the Teach-in mode. The desired turret pocket is selected through the cursor keys. You can also set up manual changing systems in the turret assignment (see "Setting up the holder for manual change systems" on page 496).

SETTING UP A TOOL LIST



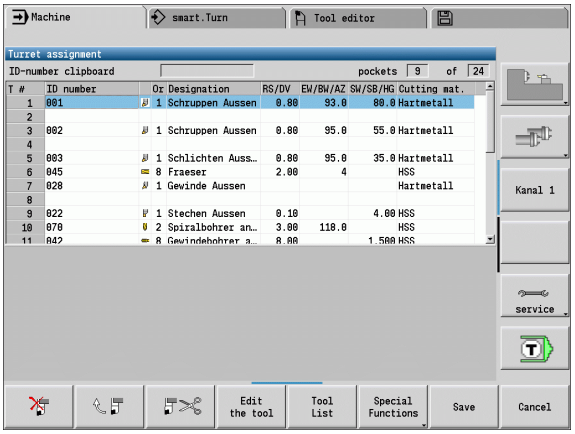
Select **Set T, S, F** (only available in Manual mode)

Initiate the cycle dialog

Tool list

Press the **Tool list** soft key to activate the turret assignment table.

Adapt the turret assignment (see soft-key table at right).



Soft keys in turret list

	Delete entry
	Paste entry from clipboard
	Cut out entry and save it in the clipboard
Tool list	Show entries in the tool database
Special Functions	Switch to next menu
Delete all	Delete the complete turret list
Back	Back by one menu level
Save	Load the T number and the tool ID number into the TSF or cycle dialog
Cancel	Close the turret list **without** loading the T number and tool ID into the dialog box. Changes in the turret list remain effective.

Tool call

T is the identification letter for the tool holder. **ID** designates the tool ID number. The tool is called by "**T**" (turret pocket number). The ID number **ID** is shown and automatically filled in the dialogs. A turret list is kept.

In Manual mode, you enter the T number in the TSF dialog box. In Teach-in mode, "T" and "ID" are cycle parameters.



If a T number is entered in the **TSF dialog box** with an ID number that is not defined in the turret list, then the turret list will be changed accordingly.

Driven tools

- Driven tools are defined in the tool description.
- The driven tool can be operated with feed per revolution if the tool spindle drive is equipped with a rotary encoder.
- If driven tools are used with constant cutting speed, the spindle speed is calculated from the tool diameter.

Tool life monitoring

If desired, you can have the CNC PILOT monitor tool life or the number of parts that are produced with a specific tool.

The tool life monitoring function adds the time a tool is used at feed rate. The quantity monitoring counts the number of finished parts. The count is compared with the entry in the tool data.

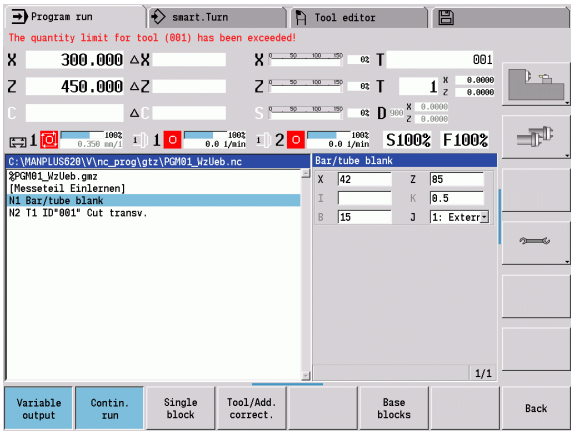
As soon as the tool life expires or the programmed quantity is reached, the CNC PILOT generates an error message and stops program execution **after** the end of the program. If you are working with program repeats (M99 in DIN programs), the system is stopped after execution of the current repeat.

- For Teach-in programs, the **simple tool life monitoring** is available. Here CNC PILOT informs you when a tool is worn out.
- In smart.Turn and DIN PLUS programs, you have the choice between the **simple tool life monitoring** or the **tool life monitoring with replacement tools** option. If you use replacement tools, the CNC PILOT automatically inserts the "sister tool" as soon as the tool is worn-out. The CNC PILOT does not stop the program run until the last tool of the tool sequence of exchange is worn out.

You activate/deactivate the tool life management in the parameter "System/General settings for automatic operation/user parameter tool life."

The type of monitoring data, the "tool life/remaining tool life" or the "maximum number of pieces/remaining number of pieces" are managed in the tool data. They are also edited and displayed in the tool data (see "Editing tool-life data" on page 491).

You can define replacement tools when setting up the turret. The "interchange chain" can contain more than one replacement tool. The interchange chain is part of the NC program (see chapter titled "Tool Programming" in the "smart.Turn and DIN Programming" User's Manual.)



You must update the data on tool life and number of pieces in the "tool management" mode when you replace the insert of a tool.

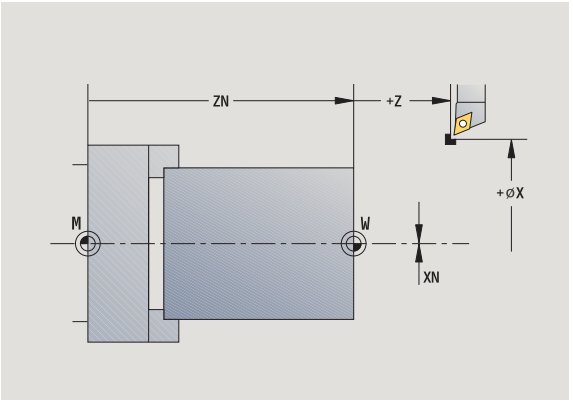
3.5 Machine setup

The machine always requires a few preparations, regardless of whether you are machining a workpiece manually or automatically. In Manual mode the following functions are subitems of the **Setup** menu item:

- Setting the axis values (defining workpiece zero point)
  - Machine reference (axis reference run)
- Setting the protection zone
- Defining the tool change position
- Setting C-axis values
- Defining machine dimensions
- Displaying operating times
- Probing

Defining the workpiece zero point

In the dialog, the distance between the machine zero point and the workpiece zero point (also know as offset) is shown as **XN** and **ZN.** If the workpiece zero point is changed, the display values will be changed accordingly.



The workpiece datum can now also be set in the direction of the Z axis using a touch probe. When setting the zero point, the control checks which type of tool is currently active. If you select the **Workpiece zero point** setup function and a touch probe is inserted, the control automatically adjusts the input form. Press NC Start to start the measuring process. Press the **Save** soft key to apply the measured value as workpiece zero point in the Z axis.

SETTING THE WORKPIECE ZERO POINT



Select **Setting up**



Select **Set axis values**

Touch the workpiece zero point (end face)

Z=0

Define this point as the "workpiece zero point Z"

Enter the distance between the tool and the workpiece zero point as "measuring point coordinate Z"

Save

The CNC PILOT calculates the workpiece zero point Z

Delete Z offset

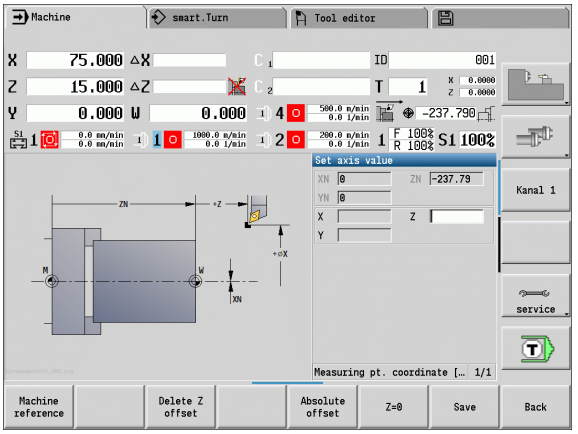
Machine zero point Z = workpiece zero point Z (offset = 0)

Absolute offset

This makes it possible to enter the zero point shift directly in ZN

Homing the axes

It is possible to home axes that have already been homed. Here you can select individual axes or all axes simultaneously.



REFERENCE RUN



Select **Setting up**



Select **Set axis values**

Machine reference

Press the **Machine reference** soft key

Z

Press the **Z reference** soft key

X

Press the **X reference** soft key

All

Or press **all** soft keys



Press **Cycle start** for the control to traverse the reference marks

The CNC PILOT refreshes the position display.

Setting the protection zone

With active protection zone monitoring, the CNC PILOT checks for every movement whether the **protection zone in –Z direction** would be damaged. If it detects such a violation, it stops the axis movement and generates an error message.

The "Setting the protection zone" setup dialog shows the distance between the machine zero point and the protection zone in **–ZS**.

The status of the protection zone monitoring is shown in the machine display if it has been configured by the machine manufacturer (see table).

SETTING THE PROTECTION ZONE/SWITCHING OFF THE MONITORING FUNCTION



Select **Setting up**



Select **Set protection zone**

Move the tool with the jog keys or handwheel until it reaches the protection zone

Take over position

Use the **Take over position** soft key to load this position as protection zone

Enter the position of the protection zone relative to the workpiece zero point (field: "Meas. pt. coordin.–Z")

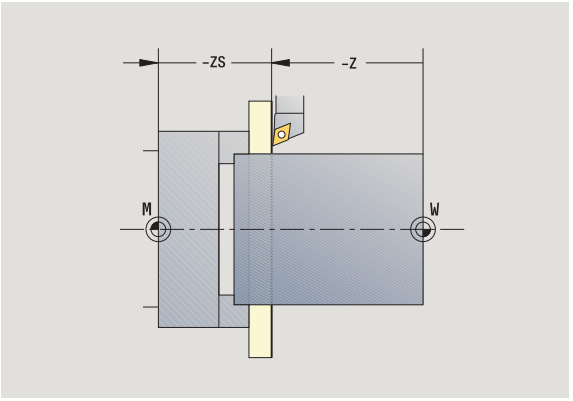
Save

Use the **Save** soft key to load the entered position as protection zone

Protectn off

Switch off protective zone monitoring

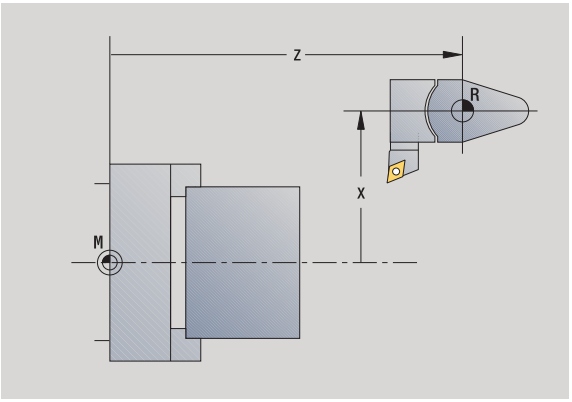
- Protective zone monitoring is not active if the **Set protect zone** dialog box is open.
- In DIN programming, you deactivate protective zone monitoring with **G60 Q1** and reactivate it with **G60.**



Protection zone status	
Protective zone monitoring active	
Protective zone monitoring not active	

Defining the tool change position

With the cycle **Move to tool change position** or the DIN command **G14,** the slide moves to the tool change point. Program the tool change point far enough away from the workpiece so that the turret can rotate without collision and the tools do not damage the workpiece during tool change.



DEFINING THE TOOL CHANGE POSITION



Select **Setting up**



Select the **tool change point**

Move to the tool change position

Take over position

Move to the tool change point using the jog keys or the handwheel and load this position as tool change point

Enter the tool change position directly

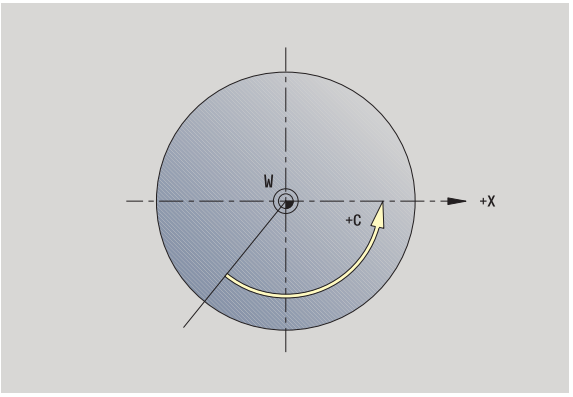
Enter the desired tool change position in the X and Z input fields in machine coordinates (X = radius dimension).

The coordinates of the tool change position are entered and displayed as the distance between machine zero point and tool carrier zero point. It is recommended to move to the tool change point and load the position with the **Take over position** soft key.

Setting C-axis values

The "Set C-axis values" enables you to define a datum shift for the workpiece spindle:

- CN: Position value of the workpiece spindle (display)
- C: Datum shift of the C-axis



DEFINING THE ZERO POINT OF THE C AXIS



Select **Setting up**



Select **Set C axis values**

Position the C axis

C=0

Define the position as the **C axis zero point**

Enter the zero point shift of the C axis:

Save

Confirm entry for CNC PILOT to calculate the **C axis zero point**

Delete C offset

Delete the zero point shift of the C axis

Expanded form view for machines with opposing spindle

If your machine is equipped with an opposing spindle, the CA parameter is shown. The CA parameter enables you to specify for which workpiece spindle (main spindle or opposing spindle) entries for the "Set C-axis value" function are effective.

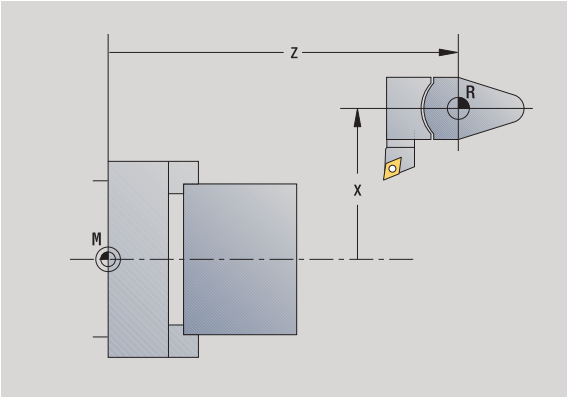
The active angle offset is shown in the CV parameter. An angle offset is activated with G905 to match the position of main and opposing spindle to each other. This may be necessary if both spindles need to be synchronized for a part transfer. The "Delete CV offset" soft key enables you to reset an active angle offset.

Additional parameters for machines with opposing spindle:

- CV: Display of active angle offset
- CA: Selection of C-axis (main spindle or opposing spindle)

Setting up machine dimensions

The "Set up machine dimensions" function allows you to save any positions to use these in NC programs.



SETTING UP MACHINE DIMENSIONS



Select **Setting up**



Select **Set up machine dimensions**

Enter the number for the machine dimensions

Transfer X

Assume position of a single axis as machine dimensions

Take over position

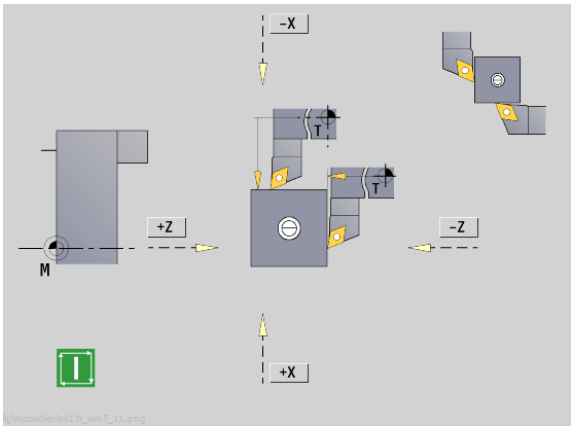
Assume position of all axes as machine dimensions

Save

Save machine dimensions

Calibrating the tool touch probe

The "Calibrate the tool touch probe" function enables you to determine the exact position values of the tool touch probe.



MEASURING THE TOUCH PROBE POSITION

Insert an exactly measured tool or reference tool



Select **Setting up**



Select **touch probe**



Select **tool touch probe**

Pre-position the tool for the first direction of measurement

+/-

Set the positive or negative traverse direction

-Z

Press the soft key for this direction (e.g. –Z direction)



Press **Cycle START**. The tool moves in the direction of measurement. The position of the touch probe is measured and saved when the tool touch probe releases a trigger signal. The tool returns to the starting point.

-X

Press the "Back" soft key to terminate the calibration process. The calibration values measured are saved, or

Pre-position the tool for the next measuring direction and repeat the procedure (max. 4 measuring directions)

Displaying operating times

In the Service menu, you can view different operating times:

Operating time	Meaning
Control on	Operating time of the control since being put into service
Machine on	Operating time of the machine tool since being put into service
Program run	Duration of controlled operation since being put into service

The machine tool builder can provide further operating time displays. The machine manual provides further information.

DISPLAY OPERATING TIMES



Select **Setting up**



Select **Service**



Select **Display operating times**

Setting the system time

With the "Adjust system time" function, you can set the date and time on your control.

You will need a mouse to navigate the **Adjust system time** input form.

Use the Month and Year soft keys to increment or decrement the respective settings.

To use an NTP server for setting the time, select a server from the server list first.

ADJUST SYSTEM TIME



Select **Setting up**



Select **Service**



Select **Adjust system time**

Select **Synchronize the time over NTP server** (if available)

Select **Set the time manually**

Select **Date**

Select **Time**

Select **Time zone**

Press the **OK** soft key

3.6 Tool measurement

The CNC PILOT supports tool calibration

- By touch-off. The setup dimensions are determined by comparing a tool with an already measured tool.
- By touch probe (stationary of swiveling in the working space; installed by the machine tool builder).
- By optical gauge (installed by the machine tool builder).

Calibration by touch-off is always available. If a touch probe or an optical gauge is installed, select these measuring methods by soft key.

If the tool dimensions are already known, you can enter the setup dimensions directly in the "tool management" mode of operation.



- The compensation values are deleted during tool measurement.
- Please note that for drilling and milling tools the center is measured.
- The tools' type and orientation determine how they are measured. Note the help graphics.

Touch off

You measure the dimensions relative to a calibrated tool by "touching the tool off."

FINDING THE TOOL DIMENSIONS BY TOUCH-OFF WITH THE TOOL

In the tool table, enter the tool you want to measure



Insert the reference tool and enter the T number in the **TSF dialog box**

Turn an end face and define this coordinate as the workpiece zero point



Return to the **TSF dialog box** and insert the tool to be measured

Measure tool

Activate **Measure tool**

Touch the face with the tool

Transfer Z

Enter the value 0 for the **measuring point coordinate Z** (workpiece zero point) and save it

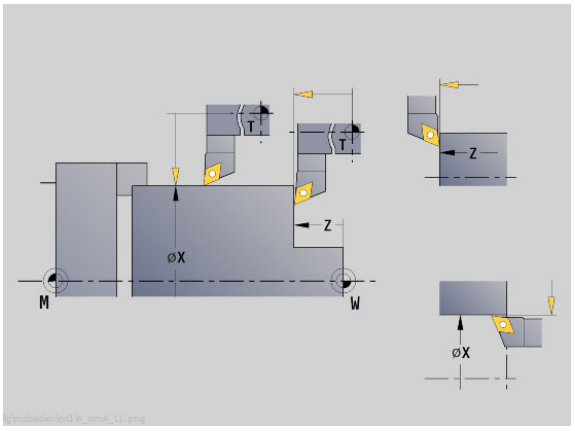
Turn a measuring diameter

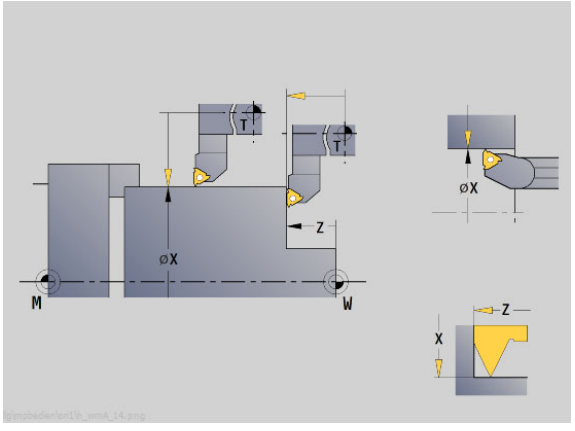
Transfer X

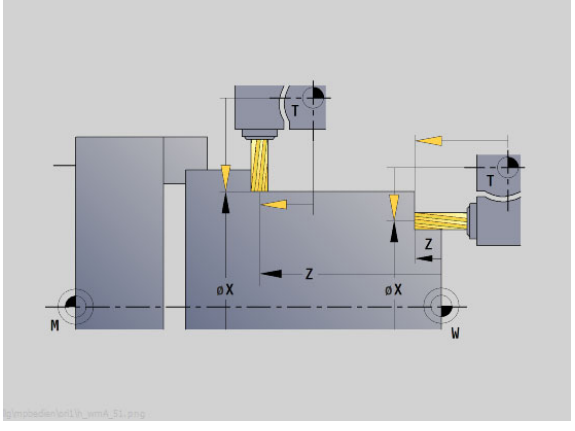
Enter the diameter value as **measuring point coordinate X** and save it

Save radius

For turning tools, enter the cutting edge radius and load it into the tool table







Touch probe (tool touch probe)

FINDING THE TOOL DIMENSIONS BY USING A TOUCH PROBE

In the tool table, enter the tool you want to measure



Insert a measured tool and enter the T number in the **TSF dialog box**

Measure tool

Activate **Measure tool**

Measure probe

Activate **Touch probe**

Pre-position the tool for the first direction of measurement

+/-

Set the positive or negative traverse direction

-Z

Press the soft key for this direction (e.g. –Z direction)



Press **Cycle START**. The tool moves in the direction of measurement. When it contacts the touch probe, the control calculates and saves the set-up dimensions. The tool returns to the starting point.

Pre-position the tool for the second direction of measurement

-X

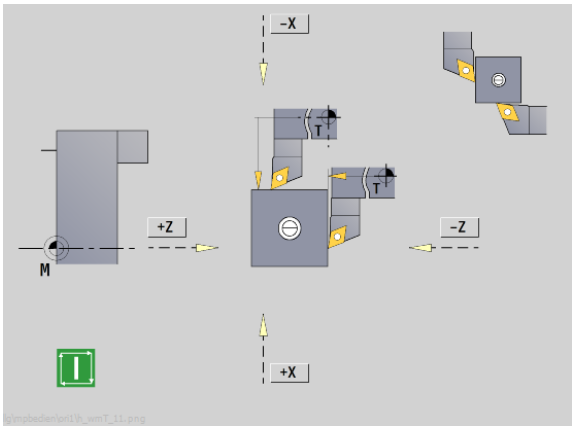
Press the soft key for this direction (e.g. –X direction)

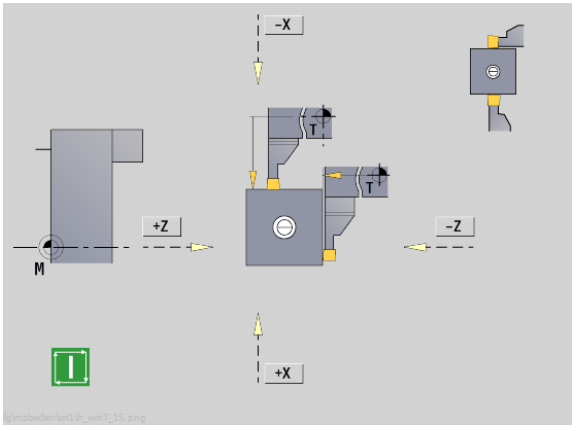


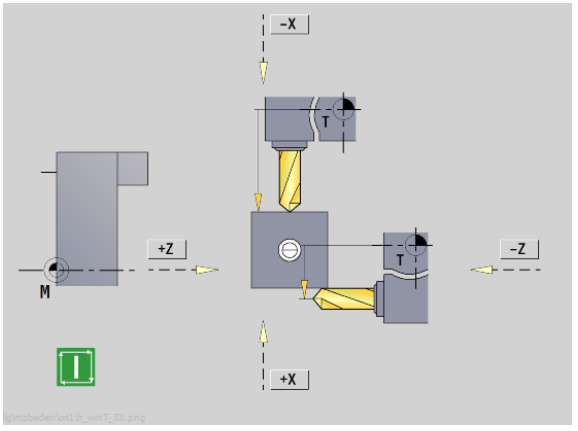
Press **Cycle START**. The tool moves in the direction of measurement. When it contacts the touch probe, the control calculates and saves the set-up dimensions.

Save radius

For turning tools, enter the cutting edge radius and load it into the tool table







Optical gauge

FINDING THE TOOL DIMENSIONS BY USING AN OPTICAL GAUGE

In the tool table, enter the tool you want to measure



Insert a measured tool and enter the T number in the **TSF dialog box**

Measure tool

Activate **Measure tool**

Measure optical

Activate **optical gauge**

Position the tool at the cross hairs of the optical gauge by using the jog keys or the handwheel

Transfer Z

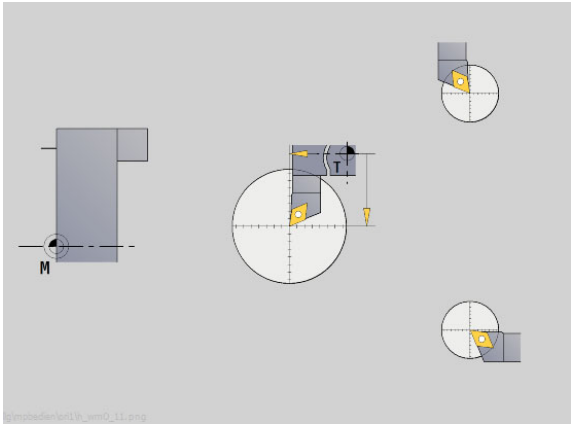
Save the tool dimension in Z

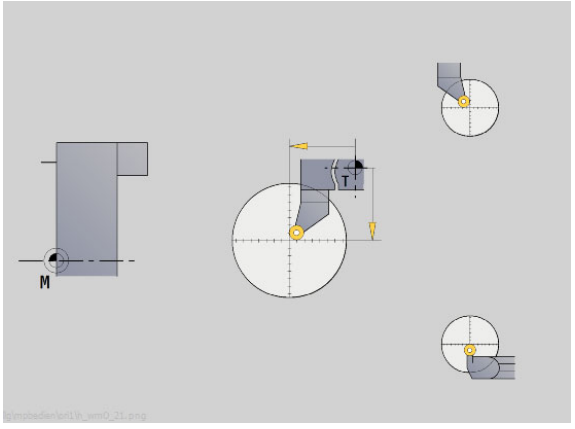
Transfer X

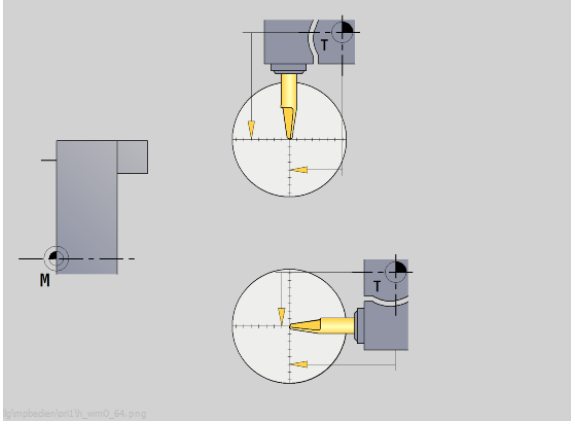
Save the tool dimension in X

Save radius

For turning tools, enter the cutting edge radius and load it into the tool table







Tool compensation

The tool compensation in X and Z as well as the special compensation for recessing tools and button tools compensate for wear of the cutting edge.

A compensation value must not exceed +/–10 mm.

DEFINING TOOL COMPENSATION



Select **Set T, S, F** (only available in Manual mode)

Tool correct.

Press the **Tool correct.** soft key

X compns. of tool

Press the **X offset for tool** (or Z offset)

The compensation values that you determine per handwheel are now shown in the distance-to-go display

Save

Transfer the compensation value to the tool table

- The T display shows the new compensation value
- The distance-to-go display is canceled

DELETING TOOL COMPENSATION VALUES



Select **Set T, S, F** (only available in Manual mode)

Tool correct.

Press the **Tool correct.** soft key

Delete

Press the **Delete** soft key

Delete X compens.

Delete compensation value in X (or Z)

3.7 "Manual Operation" operating mode

With **manual workpiece machining,** you move the axes with the handwheels or manual direction keys. You can also use Teach-in cycles for machining more complex contours (semi-automatic mode). The paths of traverse and the cycles, however, are **not stored.**

After switch-on and traversing the reference marks, the CNC PILOT is always in Manual mode. This mode remains active until you select **`Teach-in`** or **`Program Run.`** "Machine" displayed in the header indicates that you are in Manual mode.



Define the workpiece zero point and enter the machine data before you start machining.

Tool change

Enter the **T number/tool ID** in the **`Set T, S, F`** dialog box. Check the tool parameters.

T0 does not define a tool. This also means that it does not contain any data on tool length, cutting radius, etc.

Spindle

Enter the spindle speed in the **`Set T, S, F`** dialog box. To start and stop spindle rotation, press the spindle keys on the machine operating panel. The **`stopping angle A`** in the **`TSF dialog box`** makes the spindle always stop at this position.



Pay attention to the maximum speed (can be defined with **`TSF dialog box`**).

Handwheel operation

Refer to the machine manual.

Manual direction keys

With the manual direction keys, you can move the axes at the programmed feed rate or at rapid traverse. Enter the feed rate in the **TSF dialog box**.

- **Feed rate**
  - If the **spindle is rotating**: Feed per revolution [mm/rev]
  - If the **spindle is not rotating**: Feed per minute [m/min]
- Feed rate in **rapid traverse** : Feed per minute [m/min]

Teach-in cycles in Manual mode

- Set the spindle speed
- Set the feed rate
- Insert tool, define T number and check tool data (T0 is not permitted)
- Approach cycle start point
- Select the cycle and enter cycle parameters
- Graphically check the cycle run
- Run the cycle

The entries last made in a cycle dialog remain in memory until a new cycle is selected.

3.8 Teach-in mode

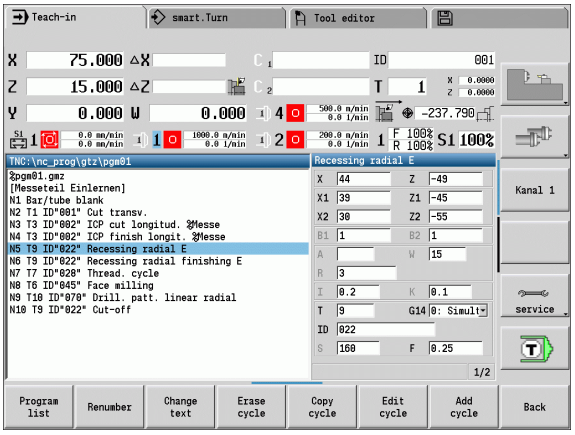
Teach-in mode

In the **Teach-in mode** you machine a workpiece step by step with the help of Teach-in cycles. The CNC PILOT "memorizes" how the workpiece was machined and stores the working steps in a cycle program, which you can call up again at any time. The **Teach-in** mode can be switched on by soft key and is displayed in the header.

Each Teach-in program is given a name and a short description. The individual cycles of a cycle program are listed as blocks and are numbered in ascending order. The block number has no meaning for the program run. The cycles are run after each other. When the cursor is located on a cycle block, CNC PILOT displays the cycle parameters.

The cycle block contains:

- Block number
- Tool used (turret pocket number and tool ID)
- Cycle designation
- Number of ICP contour or of DIN subprogram (after "%")



Programming Teach-in cycles

When creating a new Teach-in program, you program each cycle in the following sequence of actions "Enter—Simulate—Execute—Save." The individual cycles form the cycle program.

You can change Teach-in programs by simply editing the necessary cycle parameters, and delete or add cycles as required.

When you exit the **Teach-in** mode or switch off the machine, the Teach-in program remains as it was programmed or edited.

When you call an ICP cycle, the control displays a soft key for switching to the ICP contour editor (see "ICP editor in cycle mode" on page 363).

DIN subprograms are programmed in the smart.Turn editor and then integrated in a DIN cycle. You can use either the **DIN edit** soft key to go into the smart.Turn editor when you select the DIN cycle, or the operating-mode key.

Soft keys	
Program list	Switch to the "select cycle programs" function.
Renumber	Renumber the block numbers of the cycles.
Change text	Enter/edit the program description. Open the alphabetic keyboard.
Erase cycle	Delete the selected cycle.
Copy cycle	Copy the cycle parameters into a buffer memory. (Example: Adopt the parameters of the roughing cycle for the finishing cycle).
Insert	Transfer the cutting data from the tool memory. (Soft key appears only after **Copy cycle**.)
Edit cycle	Change the cycle parameter or mode. The cycle type, however, cannot be changed.
Add cycle	Insert a new cycle below the highlighted block.

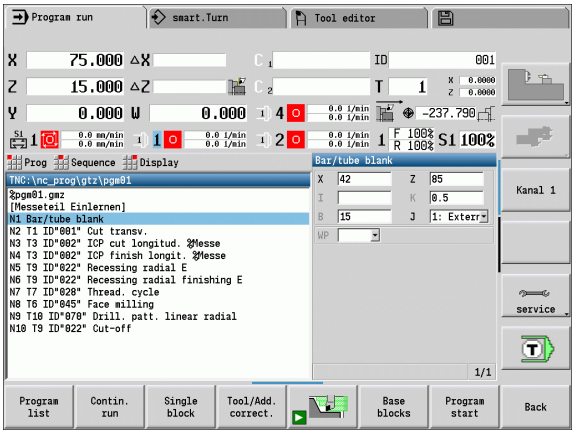
3.9 Program Run mode

Loading a program

In Program Run mode, you use Teach-in and DIN programs for parts production. You cannot change the programs in this mode. The graphic simulation feature, however, allows you to check the programs **before** you run them. The CNC PILOT also offers the **Single Block** and the **Continuous Run** mode with which you can machine step by step the first workpiece of a whole batch.

The **smart.Turn** programs are selected as DIN programs (*.nc).

Program Run automatically loads the most recently used program. To load another program, proceed as follows:



LOAD THE TEACH-IN PROGRAM OR NC PROGRAM

Program list	Open the program list—the CNC PILOT displays the Teach-in programs
DIN	Display the DIN program

Select the Teach-in program or DIN program

Open	Display the DIN program

You can start a Teach-in or smart.Turn program at any desired block to resume a machining operation after an interruption (mid-program startup).

The **Program Run** mode can be switched on with the soft key and is displayed in the header.

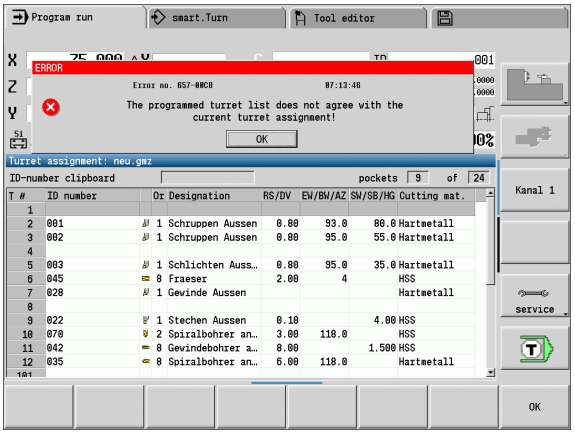
If you press **Program Run,** the CNC PILOT reads in the program that was last active in this mode or in the editing mode. Alternately, you can select another program with **Program list** (see "Program management" on page 116).

Comparing a tool list

While a program is being loaded, the CNC PILOT compares the current tools in the turret with the tool list of the program. If tools are used in the program that are not in the current turret list or are located in another pocket, an error message is displayed.

After the error message is confirmed, the program-dependent tool list is shown for checking.

You can transfer the programmed tool table with the **Load tool** soft key, or cancel the program selection with **Cancel**.



Danger of collision!

- Load the **programmed tool list** only if it matches the actual turret assignment.
- A program start is not possible unless the programmed tool list **matches** the turret list that was set up.

Before executing a program

Faulty programs
The CNC PILOT checks the programs during loading up to the **MACHINING** section. If it detects an error (for example, an error in the contour description), it displays the error symbol in the screen headline. You can then press the **Info key** for detailed information on the error.

The machining section of a program and therefore all traverse is not interpreted until after **Cycle on**. If there is an error here, the machine stops with an error message.

- **Testing the cycles and cycle parameters**
  The CNC PILOT displays the Teach-in program or the DIN program in the list window. With Teach-in programs, the parameters of the cycle on which the cursor is placed are displayed.
- **Graphic control**
  You can monitor program run with the graphic simulation feature (see "Simulation mode of operation" on page 468).

Danger of collision!

Before starting the program, check it in the simulation to find errors in the programming or the syntax used.

Finding a start block

The CNC PILOT must be prepared by the machine tool builder for the mid-program startup function (PLC).

Mid-program startup means entering into an NC program at a selected point. In smart.Turn programs you can start the program at any NC block.

The CNC PILOT starts program run from the cursor position. The starting position is not changed by a previous graphic simulation.

In the search for a start block, the CNC PILOT brings the machine into the situation in which it would be in a normal program run just before the startup block. First the tool is selected, then the axes are positioned in the configured sequence, and finally the spindle is switched on.

- In the **End mid-program startup after start block** machine parameter (601810), you can define whether program execution after a mid-program startup will start at the selected NC block or at the subsequent NC block.
- HEIDENHAIN recommends starting at an NC block immediately after a T command.

Please note:

- Position the slide with the following conditions:
  - The turret can tilt without collision.
  - The axes must be able to move to the last programmed position without collision.

The mid-program startup feature can vary depending on the individual machine tool. If the machine parameter 601810 is set to start program execution at the selected NC block, remember the following:

- If you use a T command as the starting block, the turret moves first to the previous tool and then to the tool selected in the starting block.

Program execution

The selected Teach-in or DIN program is executed as soon as you press **Cycle start.** You can interrupt machining at any time by pressing **Cycle stop.**

During program run, the cycle (or DIN block) that is presently being executed is highlighted. With Teach-in programs, the parameters of the cycle currently being run are displayed in the input window.

You can influence the program run with the soft keys listed in this table.



In the Program Run menu, the number of parts to be machined can be defined in the MP parameter (machine-dependent function). If this parameter is defined, you can only run the program until the specified quantity is reached. The control will then display a message and will not allow any further machining operations. With the Delete qty. Parts soft key, you can reset the workpiece counter.

In the P input field, you can enter the actual count. This is useful, for example, if you have already machined a number of parts.

Soft keys	
Program list	Select Teach-in program or smart.Turn program.
Contin. run	Teach-in program: ■ **On:** Cycles are run continuously, one after the other, up to the next tool change to be acknowledged. ■ **Off:** Stop after every cycle. Start the following cycle with **Cycle start**. smart.Turn program: ■ **On:** Program execution without any interruption. ■ **Off:** Stop before command M01.
Single block	■ **On:** The simulation stops after each path of traverse (basic block) Start the next path: **Cycle start**. (Recommendation: Single block should be used together with the basic-block display.) ■ **Off:** Cycles / DIN commands are executed without any interruption
Tool/Add. correct.	Input of tool compensation values or additive correction values, see "Entering compensation values during program run" on page 111
	Switch the graphic simulation on.
Base blocks	■ **On**: The traversing and switching commands are shown in DIN format (basic blocks). ■ **Off:** Teach-in or DIN program is displayed
Program start	The cursor returns to the first block of the Teach-in program or DIN program.

Entering compensation values during program run

Tool compensation

ENTERING TOOL COMPENSATION VALUES

Tool/Add. correct.

Activate the tool compensation

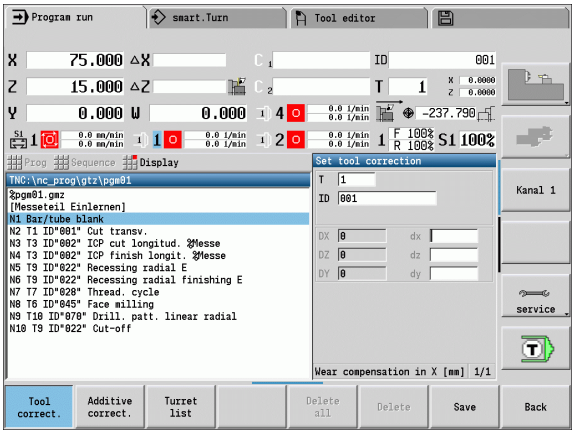
Tool correct.

Enter the tool number or select a tool from the tool list

Enter the compensation values

Save

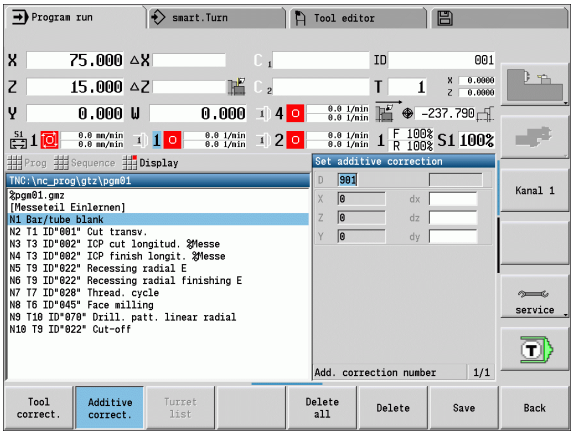
Press the **Save** soft key for the valid compensation data to be displayed in the input window and saved



- Entered values are **added** to the existing compensation values and are effective immediately.
- To delete a compensation, enter the current compensation value with the opposite algebraic sign.

Additive compensation

The CNC PILOT manages 16 additive compensation values. You edit the compensation values in the Program Run mode and activate it with **G149** in a smart.Turn program or in ICP **finishing** cycles.



ENTERING ADDITIVE COMPENSATION

Tool/Add. correct.

Activate the additive compensation

Additive correct.

Enter the number of the additive compensation

Enter the compensation values

Save

Press the **Save** soft key for the valid compensation data to be displayed in the input window and saved

READING ADDITIVE COMPENSATION

Tool/Add. correct.

Activate the additive compensation

Additive correct.

Enter the number of the additive compensation

Place the cursor in the next input field—CNC PILOT displays the valid compensation values

DELETING ADDITIVE COMPENSATION

Tool/Add. correct.

Additive correct.

Activate the additive compensation

Enter the number of the additive compensation

Delete

Press the **Delete** soft key—these compensation values are deleted

Delete all

Press the **Delete all** soft key—all compensation values are deleted

- Entered values are **added** to the existing compensation values and are effective immediately.
- The compensation values are saved in an internal table and are available in any program.
- Delete all additive compensation values when you set up the machine again.

Program execution in "dry run" mode

The dry run mode is used for fast program execution up to a point at which machining is to resume. The prerequisites for a dry run are:

- The CNC PILOT must be prepared by the machine tool builder for dry run. (The function is activated with a keylock switch or a key.)
- The **Program Run** mode must be activated.

In dry run, all feed paths (except thread cuts) are traversed at the rapid rate. You can reduce the traversing speed with the feed rate override. Do not use the dry run feature for anything other than "cutting air."

When dry run is activated, the spindle status or spindle speed is "frozen." After deactivation of the dry run, the CNC PILOT returns to the programmed feed rates and spindle speeds.



Use the dry run feature only to "cut air."

3.10 Graphic simulation

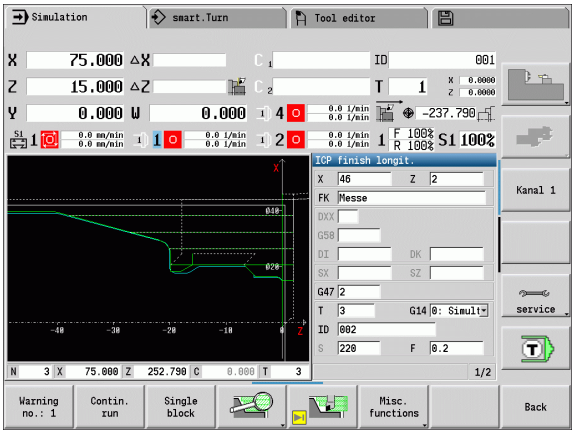
The graphic simulation feature enables you to check the machining sequence, the proportioning of cuts and the finished contour **before** actual machining.

In the **Manual Operation** and **Teach-in** modes, this function simulates the execution of a single Teach-in cycle—in **Program Run** mode it simulates a complete Teach-in cycle or DIN program.

A programmed workpiece blank is displayed in the simulation graphics. The CNC PILOT also simulates machining operations that are executed with a traversable spindle or the C axis on the face or lateral surface. This allows you to check the complete machining process.

In Manual mode and Teach-in mode, the Teach-in cycle you are currently working on is simulated. In the Program Run mode the simulation begins starting with the cursor position. smart.Turn and DIN programs are simulated from program start.

For more details on the use and operation of the simulation, refer to the chapter "Simulation mode of operation" on page 468.

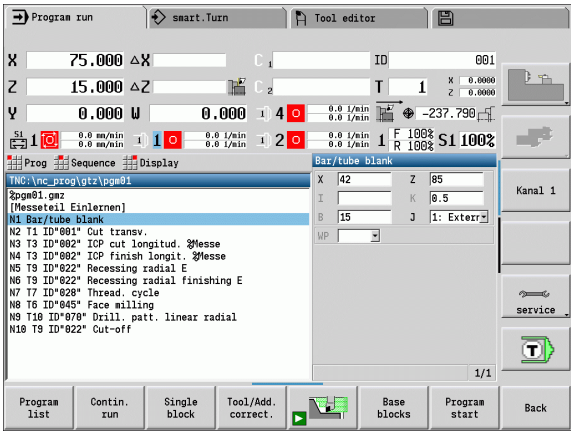


3.11 Program management

Program selection

Program Run automatically loads the most recently used program.

In the program selection the programs available in the control are listed. You select the desired program, or use the **ENTER** key to go to the **File name** input field. In this input field you limit the selection or enter the program name directly.



Program list

▶ Open the **program list**. Use the soft keys for program selection and sorting (see following table).

Soft keys in the program selection dialog

Details	Display the file attributes: **size, date, time**
DIN	Switch between **Teach-in and DIN/smart.Turn** programs
Organi-zation	Open the **program organization** soft-key menu (see page 117)
Sorting	Open the **sorting functions** soft-key menu (see following table)
Project	Open the **project organization** soft key menu (see "Project management" on page 118)
Alphabetic keyboard	Open the **alphabetic keyboard** (see "Alphanumeric keyboard" on page 55)
Open	**Open** the program for the automatic start
Cancel	**Close** the program selection dialog box. The program previously active in the Program Run mode remains effective.

Soft keys for sorting functions

Details	Display the file attributes: **size, date, time**
Sort by file name	Sort the programs by file name
Sort by size	Sort the programs by file size

Soft keys for sorting functions	
Sort by date	Sort the programs by change date
Reverse sorting	Reverse the sorting direction
Open	**Open** the program for the automatic start
Back	Return to program selection dialog

File manager

With the functions of the program organization you can copy, delete and otherwise manipulate files. You can select the program type (Teach-in programs, smart.Turn or DIN programs) before calling the program organization.

Soft keys file manager	
Paths / files	Alternate between directory and file window
Cut out	Cut marked file
Copy	Copy marked file
Insert	Insert the file stored in the buffer memory
Rename	Rename marked files
Delete	Delete marked file following confirmation prompt
Details	Display details
Mark everything	Mark all files
Sorting	Sort files
Write protection	Activate/deactivate write protection for the marked program

Soft keys file manager	
Alphabetic keyboard	Open the **alphabetic keyboard** (see "Alphanumeric keyboard" on page 55)
Back	Return to program selection dialog

Project management

You can make your own project folder in the project management so that you can centrally manage associated files. When you create a project, a new folder is set up with the corresponding subfolder structure in the "TNC:\Project\" directory. You can save your programs, contours and drawings to the subfolders.

The "Project" soft key activates the project management. The control shows you all existing projects in a tree structure. The control also opens a soft key menu in the project management that allows you to create, select and manage projects. To reselect the standard directory of the control, select the "TNC:\nc_prog" folder, or press the "Select standard dir." soft key.

Soft keys project	
New project	Create new project
Copy project	Copy marked project
Delete project	Delete marked project following confirmation prompt
Rename project	Rename marked project
Select project	Select marked project
Select std.directory	Select standard directory

You can select any project names. The subfolders (dxf, gti, gtz, ncps and Pictures) have predefined names which must not be changed.

All existing project folders are displayed in the project management. Use the file manager to change between subfolders.

3.12 Conversion into DIN format

The **Convert to DIN** function enables you to convert a Teach-in program to a smart.Turn program with the same functionality. You can then optimize, expand such a smart.Turn program, etc.

Making a conversion

CONVERSION INTO DIN FORMAT

Cyc.prog. -> DIN

Press the **Cycle program --> DIN** soft key (in the main menu)

Select the program to be converted

Cyc.prog. -> DIN

Press **Cycle program --> DIN** soft key (program selection menu)

The generated DIN program has the same program name as the Teach-in program.

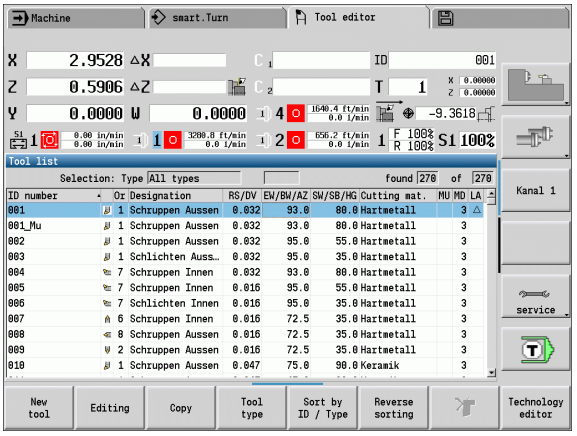
Should the CNC PILOT encounter any errors during conversion, it generates an error message and cancels conversion.

If a program with the name used is open in the smart.Turn editor, the conversion is canceled with an error message.

3.13 Units of measure

The CNC PILOT is operating in either the metric or inch system. The units and decimal places in the displays are given and entries interpreted according to the units of measure.

	inch	Metric
Units		
Coordinates, lengths, path data	inch	mm
Feed rate	in./rev or in./min	mm/rev or mm/min
Cutting speed	ft/min	m/min
Number of decimal places in displays and input		
Coordinate data and path data	4	3
Compensation values	5	3

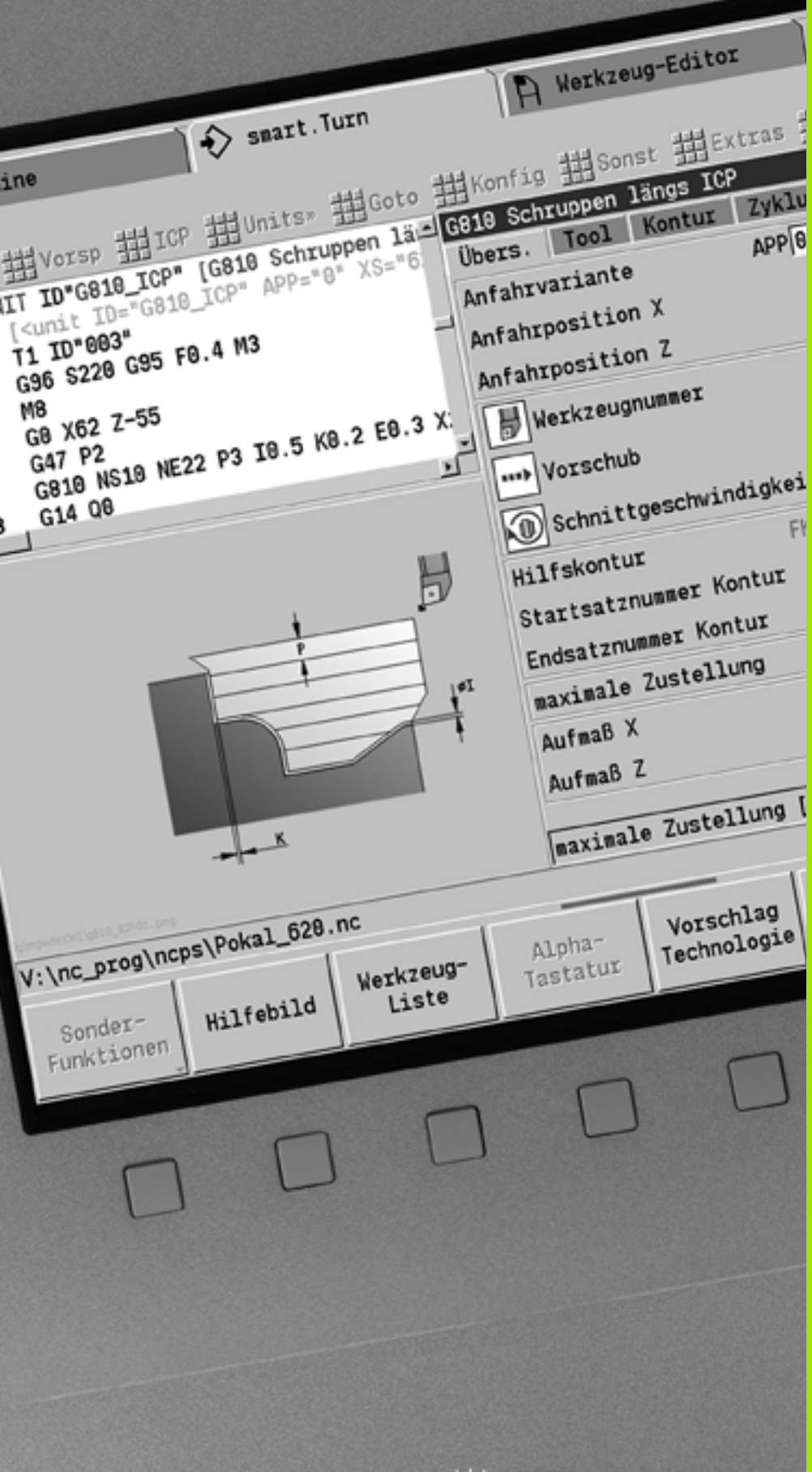


The inch/metric setting is also evaluated for the displays and entries in Tool management.

Make the metric/inch setting in the user parameter "System/Definition of unit of measure in effect for display" (Page 523). Changed metric/inch settings become effective immediately, with no need for a restart.

The basic block display also switches to inches.

- The unit is defined in all NC programs, Metric programs can be run with an active inch mode and vice versa.
- New programs are made with the selected unit.
- Refer to your machine manual if you want to know whether and how the **handwheel resolution** can be set to inches.



4

Teach-in mode

4.1 Working with cycles

Before you can use the cycles, you must set the workpiece zero point and ensure that the tools you are going to use are described. You enter the machine data (tool, feed rate, spindle speed) in Teach-in mode together with the other cycle parameters. In Manual mode, you must program these machine data before calling a cycle.

The cutting data can be taken from the technology database using the **Proposed technology** soft key. For this database access, a type of operation is permanently assigned to every cycle.

Define the individual cycles as follows:

- Position the tool tip with the handwheel or the jog keys to the starting point of the cycle (only in Manual mode)
- Select and program a cycle
- Graphically test the cycle
- Execute the cycle
- Save the cycle (only in Teach-in mode)

Cycle starting point

In Manual mode, cycles are executed from the "current tool position."

In Teach-in mode, you enter the **starting point** as one of the parameters. The CNC PILOT moves to this position at rapid traverse by the shortest path (diagonal) **before executing the cycle.**

Danger of collision!

If the tool cannot approach the next starting point on the shortest path without colliding with the workpiece, you must define an auxiliary position with the **Rapid traverse positioning** cycle.

Help graphics

The functions and parameters of the Teach-in cycles are illustrated in the graphic support window. These graphics usually show an external machining operation.



▶ The **Circle key** allows you to switch between the help graphics for internal and external machining.

Elements used in the graphic support window:

- Broken line: Rapid traverse path
- Continuous line: Feed path
- Dimension line with arrow head on one side: "Directional dimension"—the algebraic sign defines the direction
- Dimension line with arrow head on both sides: "Absolute dimension"—the algebraic sign has no effect.

DIN macros

DIN macros (DIN cycles) are DIN subprograms (see "DIN cycle" on page 356). You can integrate DIN macros in Teach-in programs. Make sure that the DIN macros do not contain any zero point shifts.



Danger of collision!

Teach-in programming: With DIN macros, the zero point shift is reset at the end of the cycle. Therefore, do not use any DIN macros with zero point shifts in Teach-in programming.

Graphical test run (simulation)

Before executing a cycle, you can graphically test the contour details and the machining sequence (see "Simulation mode of operation" on page 468).

Contour follow-up in Teach-in mode

The contour follow-up function updates the originally defined workpiece blank with every machining step. The turning cycles take the current contour of the workpiece blank into account for the calculation of infeed and machining paths. Air cuts are avoided and approach paths optimized.

In order to activate contour follow-up in Teach-in mode, program a workpiece blank and select "With contour follow-up" in the **RG** input parameter (see also "Workpiece blank cycles" on page 129).



When contour follow-up is active, you can also use modal functions, such as "Interrupted feed" or "Zero point shift."

Contour follow-up can only be used for turning operations.

Cycle run with active contour follow-up (RG: 1):

- Pressing Cycle start initiates a search for the selected cycle for mid-program startup.
- The next Cycle start executes the M commands (e.g. direction of rotation).
- The next Cycle start positions the tool to the last programmed coordinates (e.g. tool change point).
- The next Cycle start runs the selected cycle.

Cycle keys

A programmed Teach-in cycle is not executed until **Cycle start** is pressed. **Cycle stop** interrupts a running program. During thread cutting, with a **Cycle stop** the tool is lifted off the contour and then stopped. The cycle has to be **restarted.**

During a cycle interruption you can:

- Resume cycle execution with **Cycle start.** The control will always resume execution of the cycle at the point of interruption—even if you have moved the axes in the meantime.
- Move the axes with the manual direction keys or the handwheel.
- Terminate the machining process with the **Back** soft key.

Switching functions (M functions)

The CNC PILOT generates all switching functions that are necessary for running a cycle.

The direction of spindle rotation must be defined in the tool parameters. Using the tool parameters, the cycles generate spindle trigger functions (M3 or M4).

Your machine manual provides further information on automatically triggered switching functions.

Comments

You may assign a comment to an existing Teach-in cycle. The comment is inserted in brackets "[...]" below the cycle.

ADDING OR EDITING COMMENTS

Create/select a cycle

Change text

Press the **Change text** soft key

GOTO

Press the **Goto** key to show the alphabetic keyboard

Enter the comment with the on-screen alphanumeric keyboard

Save

Transfer the comment

Cycle menu

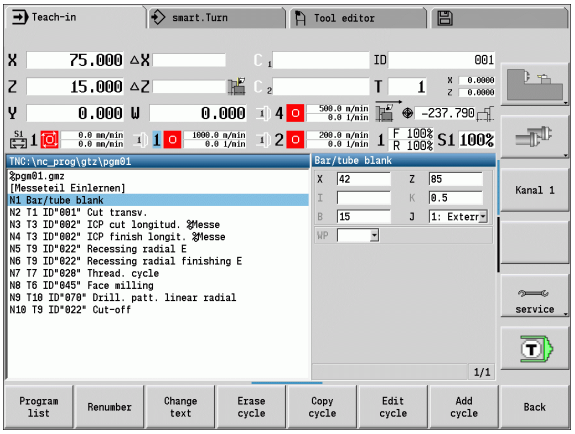
The main menu shows the cycle groups (see table below). Once a cycle group has been selected, the soft keys for the individual cycles appear.

You can use **ICP cycles** for complex contours, and **DIN macros** for technologically sophisticated machining operations. In cycle programs, the names of the ICP contours or DIN macros are at the end of the line of the cycle.

Some cycles offer **optional parameters**. That means, specific contour elements will only be machined if you set the corresponding parameters. The identification letters for optional parameters and parameters that are preassigned default values are displayed in gray.

The following parameters are only required in **Teach-in mode:**

- Starting point X, Z
- Machine data S, F, T and ID



Cycle groups	Menu key
Workpiece blank Define standard workpiece blank or workpiece blank with ICP	
Single cuts Position at rapid traverse, linear and circular single cuts, chamfers, and rounding arcs	
Turning cycles, longitudinal/transverse Turning and finishing cycles for turning and facing	
Recessing and recess-turning cycles Cycles for recessing, contour recessing, undercuts and parting	
Thread cutting Thread cycles, relief turns and thread repair	
Drilling Drilling cycles and patterns for face and lateral-surface machining	
Milling Milling cycles and patterns for face and lateral surface machining	
DIN macros Including DIN macros	

Soft keys in cycle programming: Depending on the type of cycle, you define the **variants** of the cycle by soft key (see table below).

Soft keys in cycle programming	
Edit ICP	Call the interactive contour input
T-Change approach	Move to the tool change position
Spindle Stop M19	Activate spindle positioning (M19)
With return	■ **On:** Tool returns to the cycle start point ■ **Off:** Tool remains at cycle end position
Finishing run	Switch to the finishing operation
Expanded	Switch to the expanded mode
Tool list	Open the **turret list and tool list**. You can load the tool from the list.
Take over position	Load the actual positions X and Z in Teach-in mode
Proposed technology	Accept the proposed values for feed rate and cutting velocity from the database
Constant speed	■ **On:** Constant speed (rpm) ■ **Off:** Constant surface speed (m/min)
Pattern linear	Linear drilling or milling pattern on face or lateral surface
Pattern circular	Circular drilling or milling pattern on face or lateral surface
Input finished	Transfer entered/changed values
Back	Cancel the current dialog

Addresses used in many cycles

Safety clearance G47

Safety clearances are used for approaching and departing paths. If the cycle run takes a safety clearance into account during execution you will find the address "G47" in the dialog. Proposed value: See (safety clearance G47) Page 523.

Safety clearances SCI and SCK

The safety clearances **SCI** and **SCK** are considered for approach and departure paths in drilling and milling cycles.

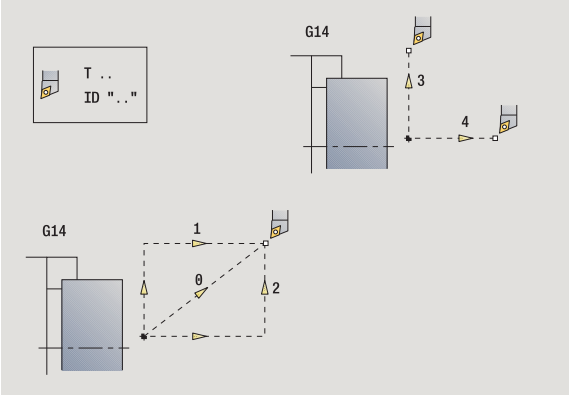
- SCI = Safety clearance in the working plane
- SCK = Safety clearance in infeed direction

Proposed value: See (safety clearance G147) Page 523.

Tool change point G14

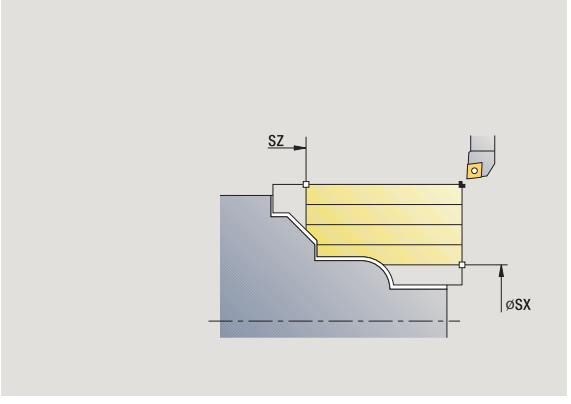
With the address G14, at the end of the cycle you program a movement of the slide to the tool change position (see "Defining the tool change position" on page 93). You can influence the approach to the tool change position as follows:

- No axis (do not approach the tool change position)
- 0: Simultaneous (default)
- 1: First X, then Z
- 2: First Z, then X
- 3: Only X
- 4:Only Z



Cutting limit SX, SZ

With the addresses **SX** and **SZ** you limit the contour area to be machined in the X and Z direction. Seen from the tool position at the beginning of the cycle, the contour to be machined is truncated at these positions.



Additive compensation Dxx

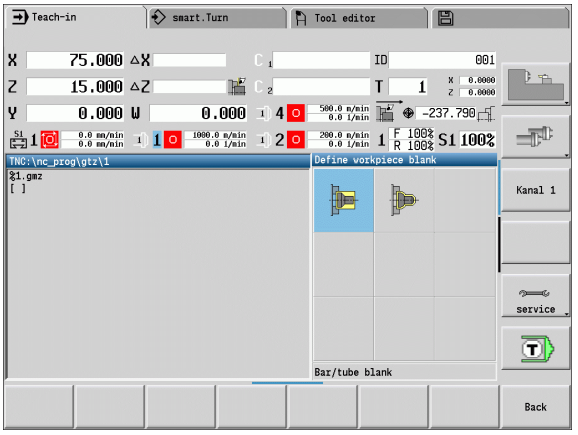
With the address **Dxx** you activate an additive compensation for the entire cycle run. The xx stands for the compensation numbers 1 to 16. The additive compensation is switched off at cycle end.

4.2 Workpiece blank cycles



The workpiece blank cycles describe the workpiece blank and the setup used. The workpiece blank cycles do not influence the machining process.

The contours of workpiece blanks are shown during the simulation of the machining process.



Workpiece blank	Symbol
Bar/tube blank Define the standard blank	
ICP workpiece blank contour Free workpiece blank description with ICP	

Bar/tube blank



Select **define the blank**



Select **bar/tube blank**

The cycle describes the workpiece blank and the setup used. This information is evaluated during the simulation.

Cycle parameters

X Outside diameter

Z Length, including transverse allowance and clamping range

I Inside diameter for workpiece blank "tube"

K Right edge (transverse allowance)

B Clamping range

J Clamp type

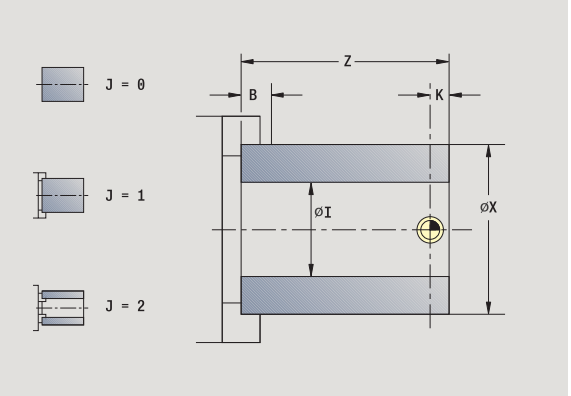
- 0: Not clamped
- 1: Externally clamped
- 2: Internally clamped

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

RG Contour follow-up for Teach-in mode (see also "Contour follow-up in Teach-in mode" on page 124):

- 0: Without contour follow-up
- 1: With contour follow-up



ICP workpiece blank contour

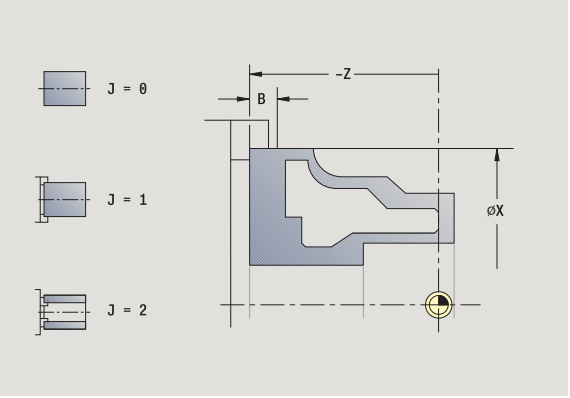


Select **define the blank**



Select **ICP workpiece blank contour**

The cycle integrates the workpiece blank defined with ICP and describes the setup used. This information is evaluated during the simulation.



Cycle parameters

X Clamp diameter

Z Clamping position in Z

B Clamping range

J Clamp type

- 0: Not clamped
- 1: Externally clamped
- 2: Internally clamped

RK ICP contour number

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

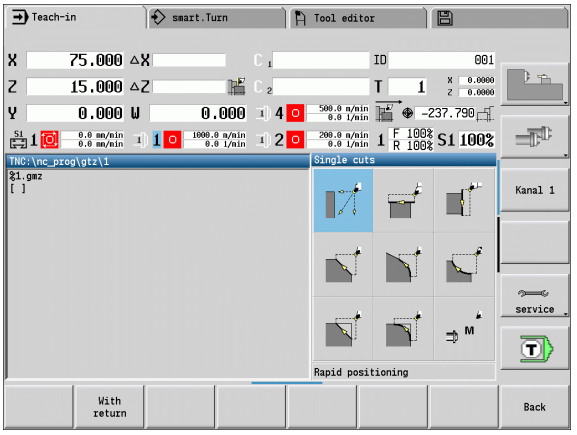
RG Contour follow-up for Teach-in mode

- 0: Without contour follow-up
- 1: With contour follow-up

4.3 Single cut cycles



In the single-cut cycles you position the tool at rapid traverse, perform linear or circular cuts, machine chamfers or rounding arcs, and enter M functions.



Single cuts	Symbol
Rapid traverse positioning	
Move to the tool change position	T-Change approach 
Linear machining, longitudinal/ transverse Single longitudinal/transverse cut	 
Linear machining at angle Single oblique cut	
Circular machining Single circular cut (for cutting direction, see menu key)	 
Machine a **chamfer**	
Machine a **rounding**	
Call an **M function**	

Rapid traverse positioning

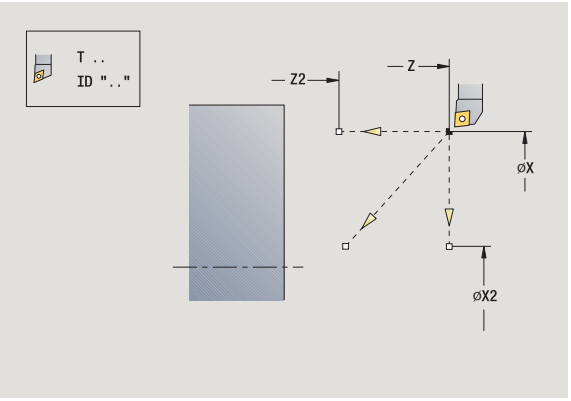


Call the **single-cut menu**



Select **rapid traverse positioning**

The tool moves at rapid traverse from the starting point to the target point.



Cycle parameters

X, Z	Starting point
X2, Z2	Target point
T	Turret pocket number
ID	Tool ID number
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent) ■ Main drive ■ Opposing spindle for rear-face machining
BW	Angle in the B axis (machine-dependent function)



If more axes are available on your machine, additional input parameters will be displayed.

Approaching the tool change position



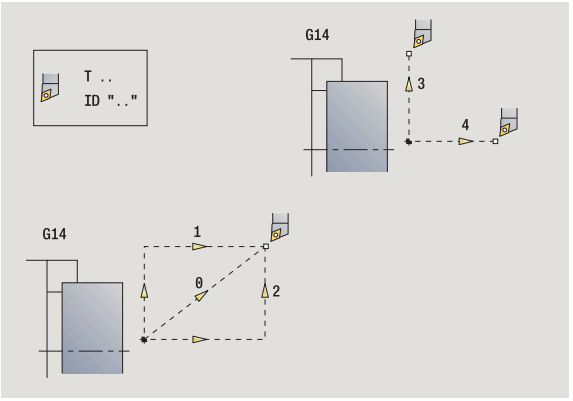
Call the **single-cut menu**



Select **rapid traverse positioning**

T-Change approach

Activate the **T-Change approach** soft key



The tool moves at rapid traverse from the current position to the tool change position (see page 128).

After reaching the tool change position, the control switches to the tool indicated in "T."

Cycle parameters

G14 Sequence (default: 0)

- 0: Simultaneous (diagonal path of traverse)
- 1: First X, then Z direction
- 2: First Z, then X direction
- 3: X direction only
- 4: Z direction only

T Turret pocket number

ID Tool ID number

MT M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Linear machining, longitudinal



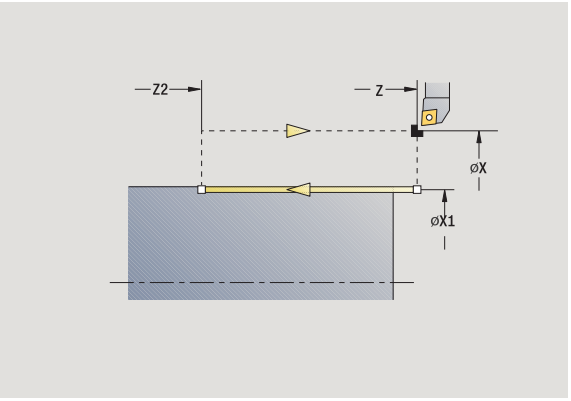
Call the **single-cut menu**

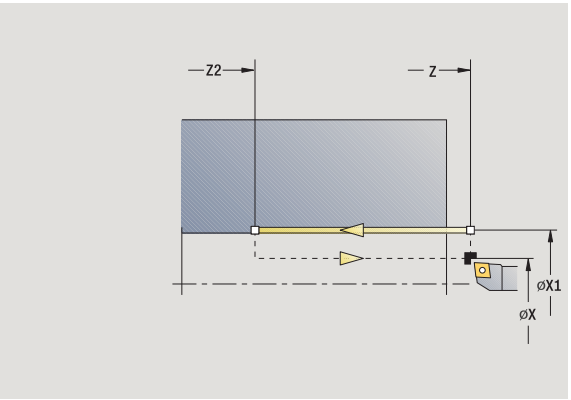


Select **longitudinal linear machining**

With return

- **Off:** When the cycle is completed, the tool remains at the cycle end position
- **On:** Tool returns to the starting point





Linear machining, longitudinal
The tool moves from the starting point to the contour **end point Z2** at the programmed feed rate and remains at the cycle end position.

Contour linear, longitudinal (with return)
The tool approaches the workpiece, executes the longitudinal cut and returns to the starting point at the end of cycle (see figures).

Cycle parameters

X, Z	Starting point
X1	Starting point of contour (if "With return" is active)
Z2	Contour end point
T	Turret pocket number
G14	Tool change point (if "With return" is active)
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent) ■ Main drive ■ Opposing spindle for rear-face machining

Type of machining for technology database access: **Finishing**

Cycle execution if "With return" is active

1. Move from the starting point to the **contour starting point X1**
2. Move to contour **end point Z2** at the programmed feed rate
3. Retract and return on paraxial path to starting point

Linear machining, transverse



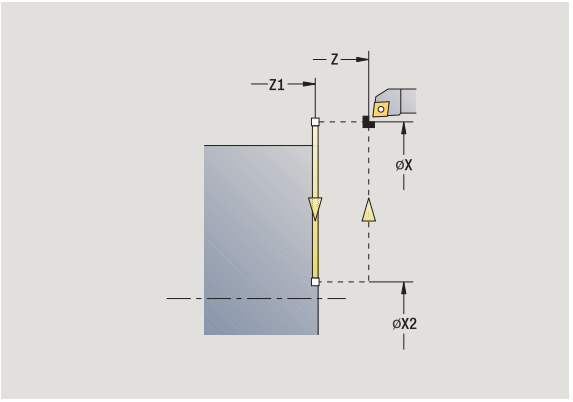
Call the **single-cut menu**

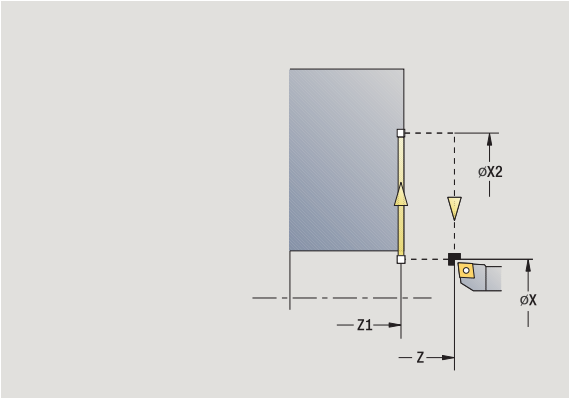


Select **transverse linear machining**

With return

- **Off:** When the cycle is completed, the tool remains at the cycle end position
- **On:** Tool returns to the starting point





Linear machining, transverse
The tool moves from the starting point to the contour **end point X2** at the programmed feed rate and remains at the cycle end position.

Contour linear, transverse (with return)
The tool approaches the workpiece, executes the transverse cut and returns to the starting point at the end of cycle (see figures).

Cycle parameters

X, Z	Starting point
Z1	Starting point of contour (if "With return" is active)
X2	Contour end point
T	Turret pocket number
G14	Tool change point (if "With return" is active)
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent) ■ Main drive ■ Opposing spindle for rear-face machining

Type of machining for technology database access: **Finishing**

Cycle execution if "With return" is active

1. Move from the starting point to the **contour starting point Z1**
2. Move to contour **end point X2** at the programmed feed rate
3. Retract and return on paraxial path to starting point

Linear machining at angle



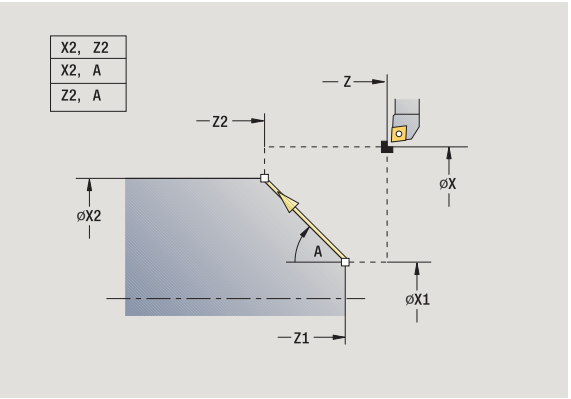
Call the **single-cut menu**



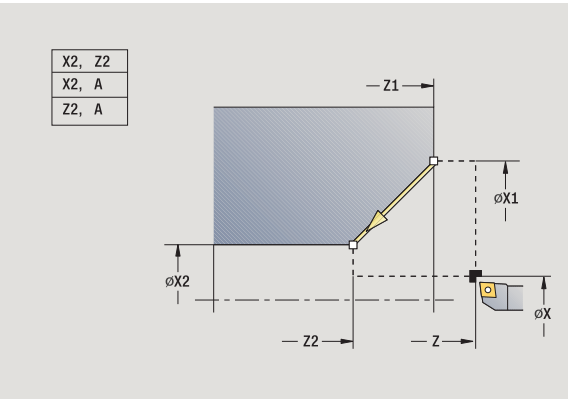
Select **linear machining at angle**

With return

- **Off:** When the cycle is completed, the tool remains at the cycle end position
- **On:** Tool returns to the starting point



Linear machining at angle
The CNC PILOT calculates the target position and moves the tool on a straight line from the starting point to the target position at the programmed feed rate. When the cycle is completed, the tool remains at the cycle end position.



Contour linear, at angle (with return)
The CNC PILOT calculates the target position. The tool then approaches the workpiece, executes the linear cut and returns to the starting point at the end of cycle (see figures). Cutter radius compensation is taken into account.

Cycle parameters

X, Z	Starting point
X1, Z1	Starting point of contour (if "With return" is active)
X2, Z2	Contour end point
A	Starting angle (range: –180° < A < 180°)
G47	Safety clearance (if "With return" is active)
T	Turret pocket number
G14	Tool change point (if "With return" is active)
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent) ■ Main drive ■ Opposing spindle for rear-face machining

Type of machining for technology database access: **Finishing**

Parameter combinations for defining the target point: see help graphic

Cycle execution if "With return" is active

1. Calculate the target position
2. Move from the starting point to the **`contour starting point X1, Z1`**
3. Move to target position at programmed feed rate
4. Retract and return on paraxial path to starting point

Circular machining



Call the **single-cut menu**



Select **circular machining** (counterclockwise)

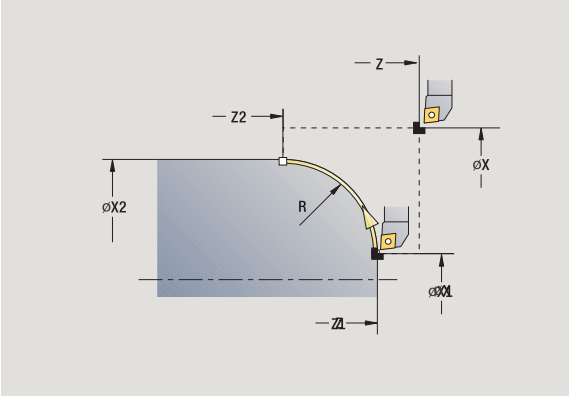


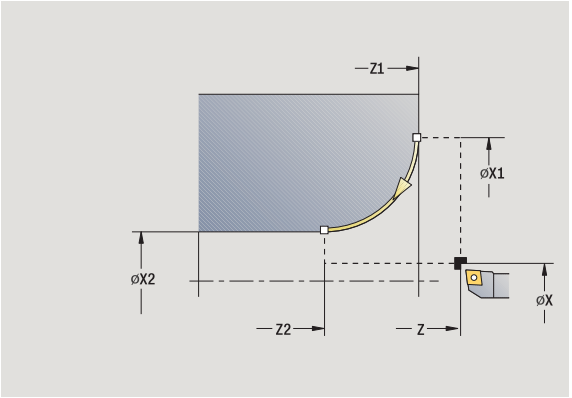
Select **circular machining** (clockwise)

With return

- **Off:** When the cycle is completed, the tool remains at the cycle end position
- **On:** Tool returns to the starting point

Circular machining
The tool moves on a circular path from the **starting point X, Z** to contour **end point X2, Z2** and stops at the end of the cycle.





Contour circular (with return)
The tool approaches the workpiece, executes the circular cut and returns to the starting point at the end of cycle (see figures). Cutter radius compensation is taken into account.

Cycle parameters

X, Z	Starting point
X1, Z1	Starting point of contour (if "With return" is active)
X2, Z2	Contour end point
R	Radius of rounding
G47	Safety clearance (if "With return" is active)
T	Turret pocket number
G14	Tool change point (if "With return" is active)
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent) ■ Main drive ■ Opposing spindle for rear-face machining

Type of machining for technology database access: **Finishing**

Cycle execution if "With return" is active

1. Move paraxially from the starting point to the **`contour starting point X1, Z1`**
2. Move in circular arc to contour **`end point X2, Z2`** at the programmed feed rate
3. Retract and return on paraxial path to starting point

Chamfer



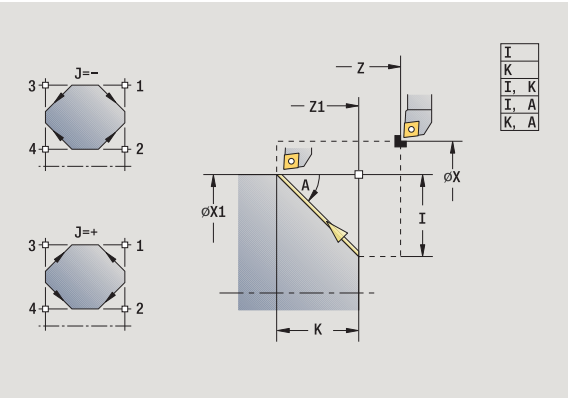
Call the **single-cut menu**



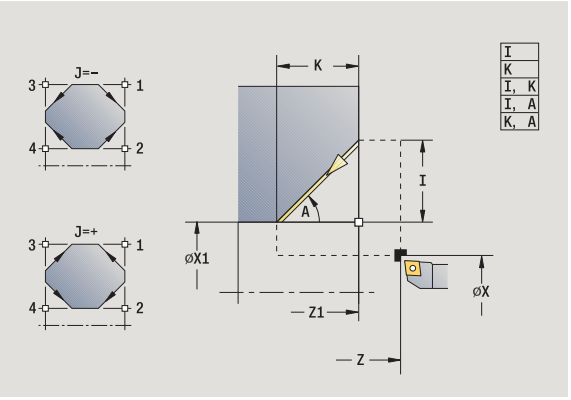
Select **chamfer**

With return

- **Off:** When the cycle is completed, the tool remains at the cycle end position
- **On:** Tool returns to the starting point



Chamfer
The cycle produces a chamfer that is dimensioned relative to the corner of the workpiece contour. When the cycle is completed, the tool remains at the cycle end position.



Contour chamfer (with return)
The tool approaches the workpiece, machines the chamfer that is dimensioned relative to the corner of the workpiece contour and returns to the starting point at the end of cycle. Cutter radius compensation is taken into account.

Cycle parameters

X, Z	Starting point
X1, Z1	Corner point of contour
A	Starting angle: Angle of the chamfer (range: 0°< A < 90°)
I, K	Chamfer width (in X, Z)
J	Element position (default: 1)—the algebraic sign determines the cutting direction (see help graphic)
G47	Safety clearance (if "With return" is active)
T	Turret pocket number
G14	Tool change point (if "With return" is active)
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent) ■ Main drive ■ Opposing spindle for rear-face machining

Type of machining for technology database access: **Finishing**

Parameter combinations for defining the chamfer:

- I or K (45° chamfer)
- I, K
- I, A or K, A

Cycle execution if "With return" is active

1. Calculate starting point and end point of chamfer
2. Move paraxially from the starting point to the "chamfer starting point"
3. Move to end point of chamfer at the programmed feed rate
4. Retract and return on paraxial path to starting point

Rounding arc



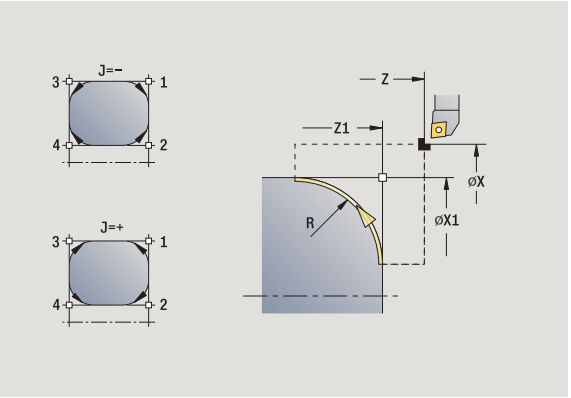
Call the **single-cut menu**



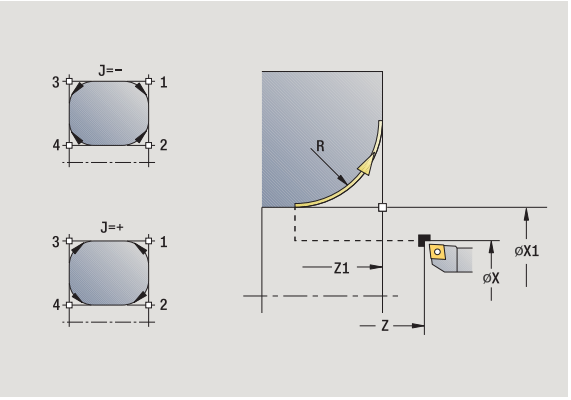
Select **rounding**

With return

- **Off:** When the cycle is completed, the tool remains at the cycle end position
- **On:** Tool returns to the starting point



Rounding arc
The cycle produces a rounding arc that is dimensioned relative to the corner of the workpiece contour. When the cycle is completed, the tool remains at the cycle end position.



Contour rounding (with return)
The tool approaches the workpiece, machines the rounding that is dimensioned relative to the corner of the workpiece contour and returns to the starting point at the end of cycle. Cutter radius compensation is taken into account.

Cycle parameters

X, Z	Starting point
X1, Z1	Corner point of contour
R	Radius of rounding
J	Element position (default: 1)—the algebraic sign determines the cutting direction (see help graphic)
G47	Safety clearance (if "With return" is active)
T	Turret pocket number
G14	Tool change point (if "With return" is active)
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent) ■ Main drive ■ Opposing spindle for rear-face machining

Type of machining for technology database access: **Finishing**

Cycle execution if "With return" is active

1. Calculate "starting point and end point of arc"
2. Move paraxially from the starting point to the "arc starting point"
3. Move in circular arc to end point of rounding at programmed feed rate
4. Retract and return on paraxial path to starting point

M functions

Machine commands (M functions) are not executed until **Cycle start** has been pressed. With the **M list** soft key you can open an overview of the available M functions. For the meaning of the M functions, refer to your machine manual.

M FUNCTION



Call the **single-cut menu**



Select **M function**

Enter the number of the M function

Input finished

Conclude entry



Press **Cycle start**

SPINDLE STOP M19 (SPINDLE POSITIONING)



Call the **single-cut menu**



Select **M function**

Spindle Stop M19

Switch M19 on

Enter the stopping angle

Input finished

Conclude entry



Press **Cycle start**

4.4 Turning cycles



Turning cycles rough and finish simple contours in **basic mode** and complex contours in **expanded mode.**

With ICP cutting cycles, you can machine contours defined with **ICP.** See "ICP contours" on page 360.

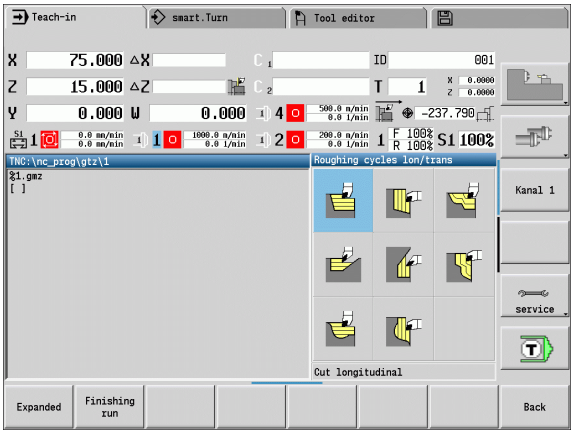


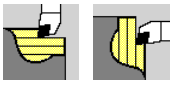
- **Proportioning of cuts:** The CNC PILOT calculates an infeed that is <=**infeed depth P.** An "abrasive cut" is avoided.
- **Oversizes:** Oversizes are considered in "expanded" mode.
- **Cutter radius compensation:** Active
- **Safety clearance** after each step:
  - Basic mode: 1 mm
  - Expanded mode: The safety clearance is set separately for internal and external machining (see "List of user parameters" on page 523).

Cutting and infeed directions for turning cycles

The CNC PILOT automatically determines the cutting and infeed directions from the cycle parameters.

- **Basic mode:** The parameters for starting point X, Z (Manual mode: current tool position) and contour starting point X1 / contour end point Z2 determine these directions.
- **Expanded mode:** The parameters for contour starting point X1, Z1 and contour end point X2, Z2 determine these directions.
- **ICP cycles:** The parameters for contour starting point X, Z (Manual mode: current tool position) and starting point of the ICP contour determine these directions.



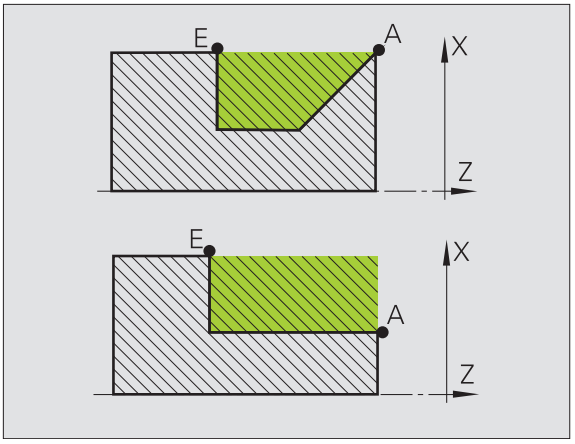
Turning cycles	Symbol
Turning, longitudinal/transverse Roughing and finishing cycle for simple contours	
Plunge-cutting, longitudinal/ transverse Roughing and finishing cycle for simple plunge-cut contours	
ICP contour-parallel, longitudinal/transverse Roughing and finishing cycle for any type of contour (cutting paths parallel to finished part)	
ICP turning, longitudinal/ transverse Roughing and finishing cycle for any type of contour	

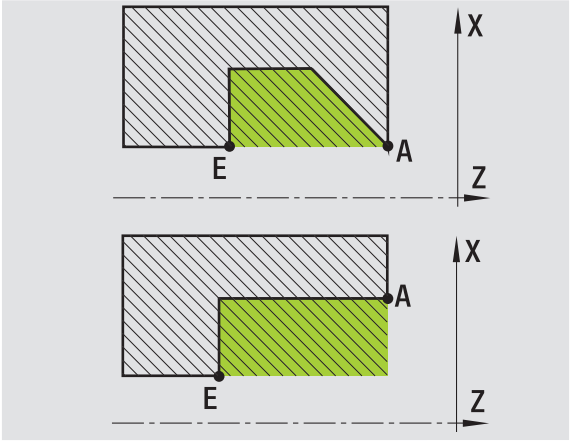
Tool position

It is important that you observe the tool positions (starting point X, Z) before executing any of the turning cycles in expanded mode. The rules also apply for all cutting and infeed directions as well as for roughing and finishing (see examples of linear cycles).

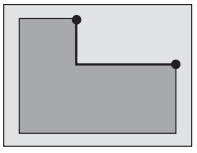
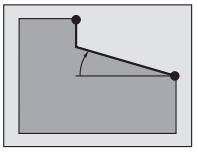
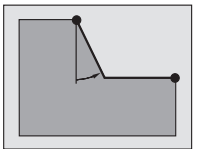
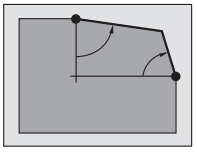
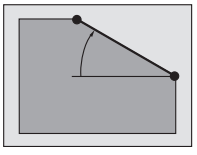
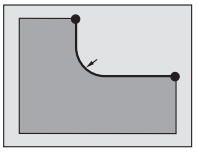
- The starting point must not be located in the shaded area.
- The area to be machined starts at the **`starting point X, Z,`** if the tool is positioned before the contour area. The control will otherwise only machine the contour area defined.
- If the **`starting point X, Z`** for internal machining is located above the turning center, only the contour area defined will be machined.

(A = contour starting point X1, Z1; E = contour end point X2, Z2)



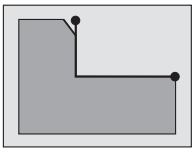


Contour elements

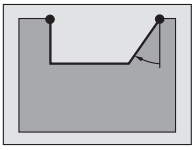
Contour elements in turning cycles	
Basic mode Machining a rectangular area	
Expanded mode Oblique cut at contour start	
Expanded mode Oblique cut at contour end	
Expanded mode Oblique cuts at contour start and end with angles > 45°	
Expanded mode **One** oblique cut (by entering the starting point of contour, end point of contour and starting angle)	
Expanded mode Rounding	

Contour elements in turning cycles

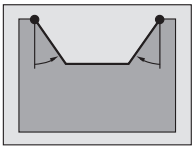
Expanded mode
Chamfer (or rounding arc) at contour end



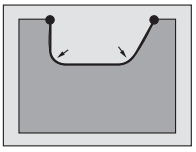
Basic mode
Machining with descending contour



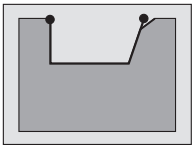
Basic mode
Oblique cut at contour end



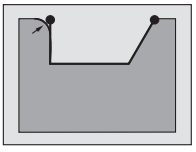
Expanded mode
Rounding in contour valley (in both corners)



Expanded mode
Chamfer (or rounding arc) at contour start



Expanded mode
Chamfer (or rounding arc) at contour end



Roughing, longitudinal

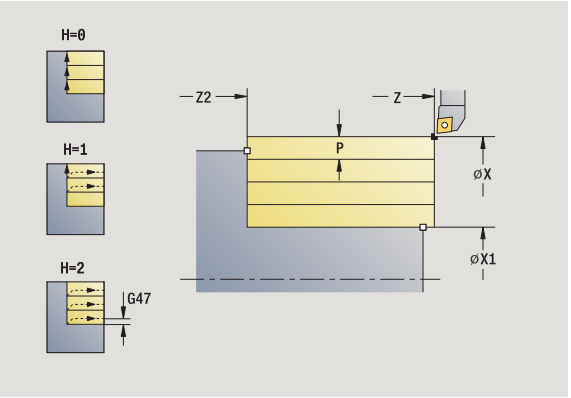


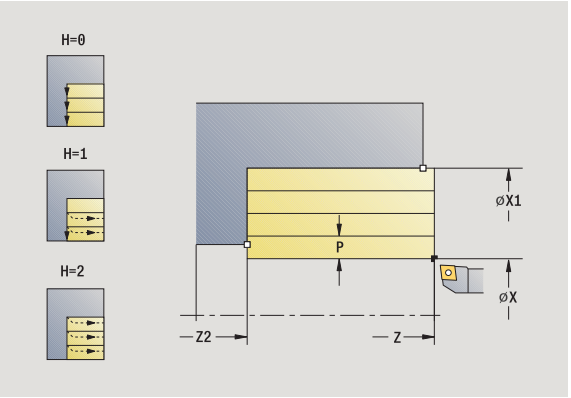
Select **cut, longitudinal/transverse**



Select **cut longitudinal**

The cycle roughs the rectangle described by the **starting point** and **contour starting point X1/contour end point Z2**.





Cycle parameters

X, Z	Starting point
X1	Contour starting point
Z2	Contour end point
P	Infeed depth: Maximum infeed depth
H	Contour smoothing
	■ 0: With every cut
	■ 1: With the last cut
	■ 2: No finishing cut
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Roughing**

Cycle run

1 Calculate the proportioning of cuts (infeed)
2 Approach the workpiece from starting point for first pass
3 Move at the programmed feed rate to the **contour end point Z2**
4 The contour is machined depending on the **contour smoothing H**:
5 Retract and approach for next pass
6 Repeat 3 to 5 until the **contour starting point X1** is reached
7 Return to starting point on diagonal path
8 Move to the tool change point according to the G14 setting

Cut, transverse

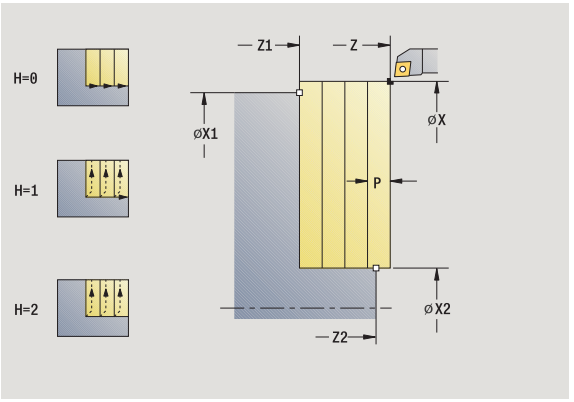


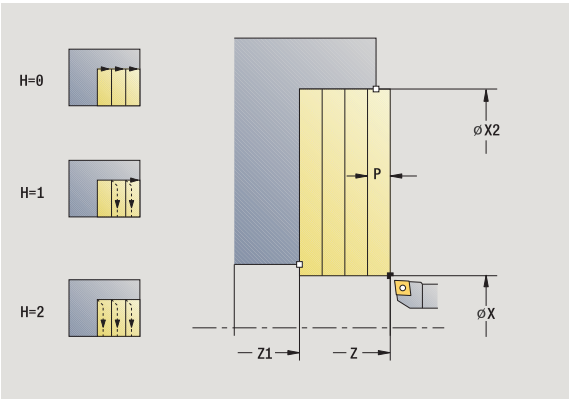
Select **cut, longitudinal/transverse**



Select **cut transverse**

The cycle roughs the rectangle described by the **starting point** and **contour starting point Z1/contour end point X2**.





Cycle parameters

X, Z	Starting point
Z1	Contour starting point
X2	Contour end point
P	Infeed depth: Maximum infeed depth
H	Contour smoothing
	■ 0: With every cut
	■ 1: With the last cut
	■ 2: No finishing cut
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Roughing**

Cycle run

1. Calculate the proportioning of cuts (infeed)
2. Approach the workpiece from starting point for first pass
3. Move at the programmed feed rate to the **contour end point X2**
4. The contour is machined depending on the **contour smoothing H**:
5. Retract and approach for next pass
6. Repeat 3 to 5 until the **contour starting point Z1** is reached
7. Return to starting point on diagonal path
8. Move to the tool change point according to the G14 setting

Roughing, longitudinal—expanded



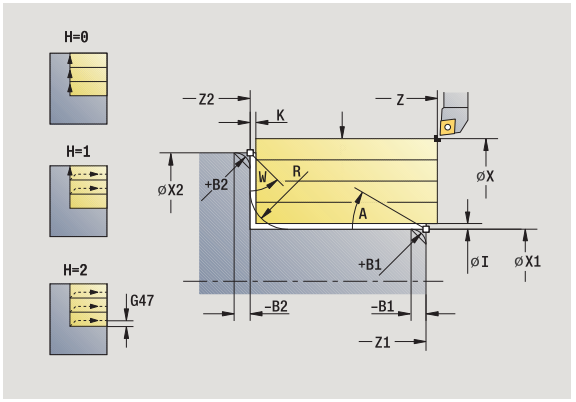
Select **cut, longitudinal/transverse**

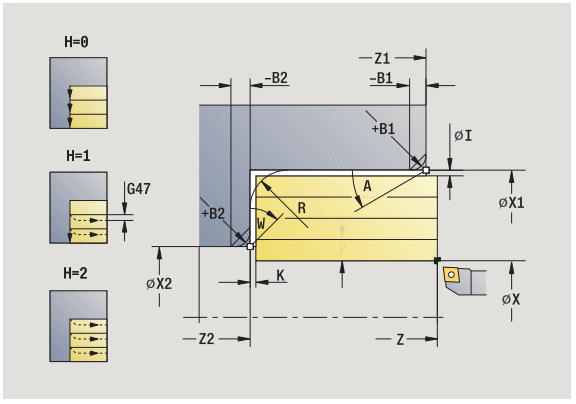


Select **cut longitudinal**

Expanded

Press the **Expanded** soft key





Taking the oversizes into account, the cycle roughs the area described by the **starting point** and **contour starting point X1/contour end point Z2**.

Cycle parameters

X, Z	Starting point
X1, Z1	Contour starting point
X2, Z2	Contour end point
P	Infeed depth: Maximum infeed depth
A	Starting angle (range: 0° <= A < 90°)
W	End angle (range: 0° <= W < 90°)
R	Rounding arc
I, K	Oversize X, Z
H	Contour smoothing ■ 0: With every cut ■ 1: With the last cut ■ 2: No finishing cut
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
B1, B2	Chamfer/rounding arc (B1 contour start; B2 contour end) ■ B>0: Rounding radius ■ B<0: Chamfer width
BP	Break duration: Time span for interruption of the feed. The chip is broken by the (intermittent) interruption of the feed.
BF	Feed duration: Time interval until the next break. The chip is broken by the (intermittent) interruption of the feed.
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Roughing**

By setting the following **optional parameters,** you can define additional contour elements:

- A:Oblique cut at contour start
- W:Oblique cut at contour end
- R:Rounding
- B1:Chamfer/Rounding at contour start
- B2:Chamfer/Rounding at contour end
- BP:Pause duration
- BF:Feed duration
- WS:Angle of the chamfer at the contour starting point (not yet implemented)
- WE:Angle of the chamfer at the contour end point (not yet implemented)

Cycle run

1. Calculate the proportioning of cuts (infeed)
2. Approach the workpiece from starting point for first pass
3. Move at the programmed feed rate to the **`contour end point Z2,`** or if defined, to one of the optional contour elements
4. The contour is machined depending on the **`contour smoothing H`**:
5. Retract and approach for next pass
6. Repeat 3 to 5 until the **`contour starting point X1`** is reached
7. Return to starting point on paraxial path
8. Move to the tool change point according to the G14 setting

Roughing, transverse—expanded



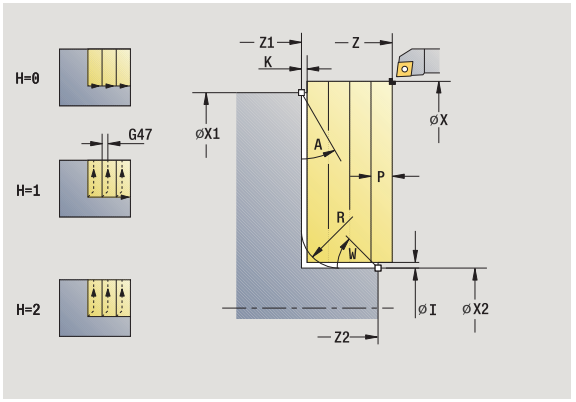
Select **cut, longitudinal/transverse**

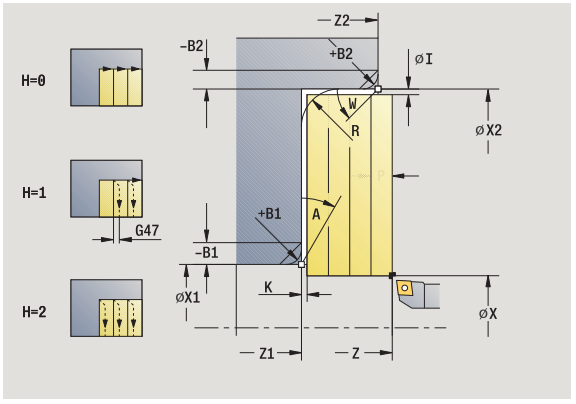


Select **cut transverse**

Expanded

Press the **Expanded** soft key





Taking the oversizes into account, the cycle roughs the area described by the **starting point** and the **contour starting point Z1/contour end point X2**.

Cycle parameters

X, Z	Starting point
X1, Z1	Contour starting point
X2, Z2	Contour end point
P	Infeed depth: Maximum infeed depth
A	Starting angle (range: 0° <= A < 90°)
W	End angle (range: 0° <= W < 90°)
R	Rounding arc
I, K	Oversize X, Z
H	Contour smoothing
	■ 0: With every cut
	■ 1: With the last cut
	■ 2: No finishing cut
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
B1, B2	Chamfer/rounding arc (B1 contour start; B2 contour end)
	■ B>0: Rounding radius
	■ B<0: Chamfer width
BP	Break duration: Time span for interruption of the feed. The chip is broken by the (intermittent) interruption of the feed.
BF	Feed duration: Time interval until the next break. The chip is broken by the (intermittent) interruption of the feed.
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Roughing**

By setting the following **optional parameters,** you can define additional contour elements:

- A:Oblique cut at contour start
- W:Oblique cut at contour end
- R:Rounding
- B1:Chamfer/Rounding at contour start
- B2:Chamfer/Rounding at contour end
- BP:Pause duration
- BF:Feed duration
- WS:Angle of the chamfer at the contour starting point (not yet implemented)
- WE:Angle of the chamfer at the contour end point (not yet implemented)

Cycle run

1. Calculate the proportioning of cuts (infeed)
2. Approach the workpiece from starting point for first pass
3. Move at the programmed feed rate to the **`contour end point X2,`** or if defined, to one of the optional contour elements
4. The contour is machined depending on the **`contour smoothing H`**:
5. Retract and approach for next pass
6. Repeat 3 to 5 until the **`contour starting point Z1`** is reached
7. Return to starting point on paraxial path
8. Move to the tool change point according to the G14 setting

Finishing cut, longitudinal



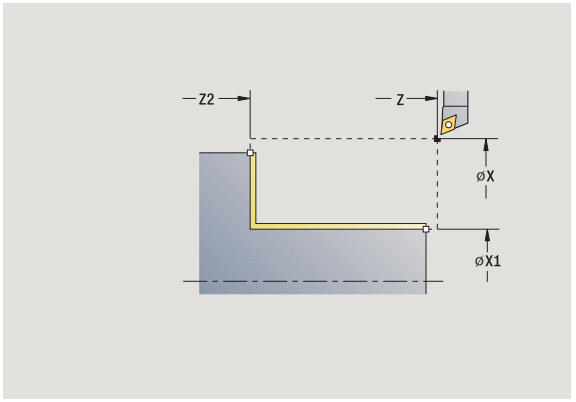
Select **cut, longitudinal/transverse**



Select **cut longitudinal**

Finishing run

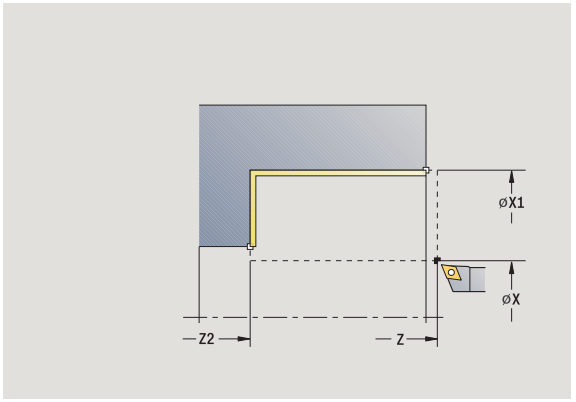
Press the **Finishing run** soft key



The cycle finishes the contour area from **contour starting point X1** to **contour end point Z2**.



At the end of the cycle, the tool returns to the starting point.



Cycle parameters

X, Z	Starting point
X1	Contour starting point
Z2	Contour end point
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent) ■ Main drive ■ Opposing spindle for rear-face machining

Type of machining for technology database access: **Finishing**

Cycle run

1. Move in transverse direction from the starting point to the **contour starting point X1**
2. Finish first in longitudinal direction, then in transverse direction
3. Return in longitudinal direction to starting point
4. Move to the tool change point according to the G14 setting

Finishing cut, transverse



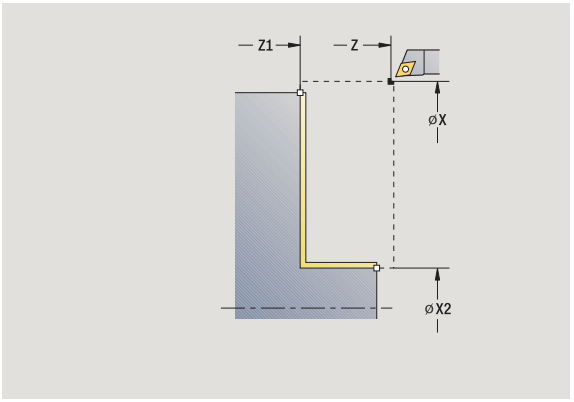
Select **cut, longitudinal/transverse**



Select **cut transverse**

Finishing run

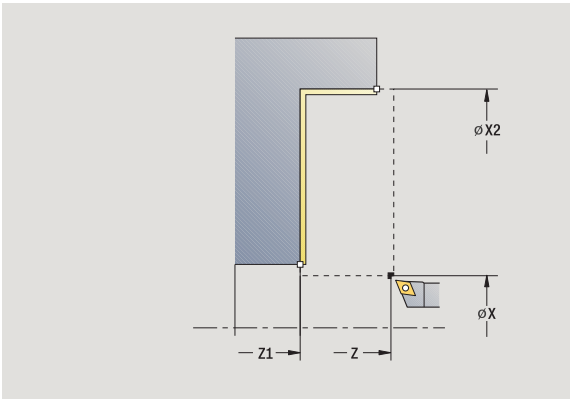
Press the **Finishing run** soft key



The cycle finishes the contour area from **contour starting point Z1** to **contour end point X2**.



At the end of the cycle, the tool returns to the starting point.



Cycle parameters

X, Z	Starting point
Z1	Contour starting point
X2	Contour end point
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent) ■ Main drive ■ Opposing spindle for rear-face machining

Type of machining for technology database access: **Finishing**

Cycle run

1. Move in longitudinal direction from the starting point to the **contour starting point Z1**
2. Finish first in transverse direction, then in longitudinal direction
3. Return in transverse direction to starting point
4. Move to the tool change point according to the G14 setting

Finishing cut, longitudinal—expanded



Select **cut, longitudinal/transverse**



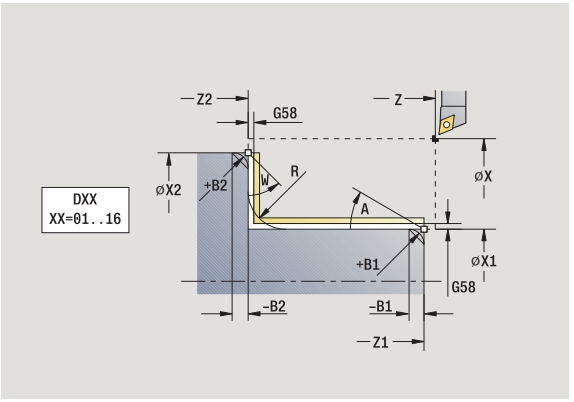
Select **cut longitudinal**

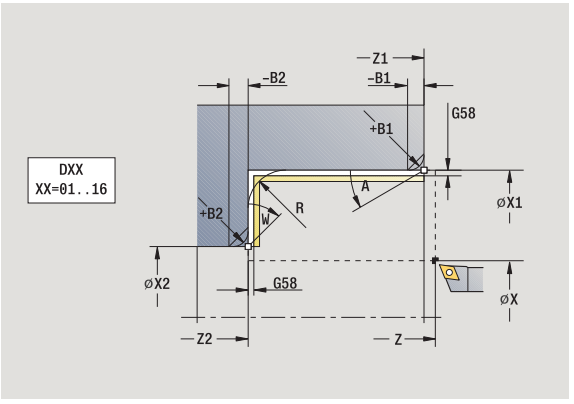
Expanded

Press the **Expanded** soft key

Finishing run

Press the **Finishing run** soft key





The cycle finishes the contour area from the **contour starting point** to the **contour end point**.



When the cycle is completed, the tool remains at the cycle end position.

Cycle parameters

X, Z	Starting point
X1, Z1	Contour starting point
X2, Z2	Contour end point
A	Starting angle (range: 0° <= A < 90°)
W	End angle (range: 0° <= W < 90°)
R	Rounding arc
DXX	Additive compensation number 1 –16 (see page 128)
G58	Contour-parallel oversize
G47	Safety clearance (see page 128)
B1, B2	Chamfer/rounding arc (B1 contour start; B2 contour end) ■ B>0: Rounding radius ■ B<0: Chamfer width
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Finishing**

By setting the following **optional parameters,** you can define additional contour elements:

- A:Oblique cut at contour start
- W:Oblique cut at contour end
- R:Rounding
- B1:Chamfer/Rounding at contour start
- B2:Chamfer/Rounding at contour end
- WS:Angle of the chamfer at the contour starting point (not yet implemented)
- WE:Angle of the chamfer at the contour end point (not yet implemented)

Cycle run

1. Move in transverse direction from the starting point to the **contour starting point X1, Z1**
2. Finish contour area from **contour starting point X1, Z1** to **end point X2, Z2,** taking optional contour elements into account
3. Move to the **tool change position** according to the **G14** setting

Finishing cut, transverse—expanded



Select **cut, longitudinal/transverse**



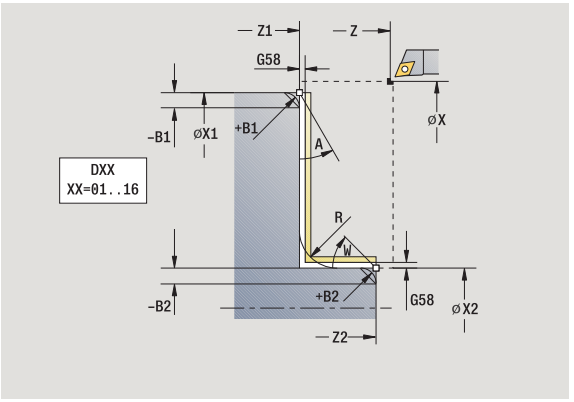
Select **cut transverse**

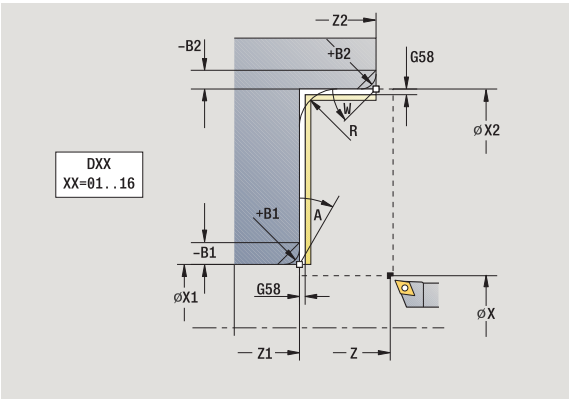
Expanded

Press the **Expanded** soft key

Finishing run

Press the **Finishing run** soft key





The cycle finishes the contour area from the **contour starting point** to the **contour end point**.

When the cycle is completed, the tool remains at the cycle end position.

Cycle parameters

X, Z	Starting point
X1, Z1	Contour starting point
X2, Z2	Contour end point
A	Starting angle (range: 0° <= A < 90°)
W	End angle (range: 0° <= W < 90°)
R	Rounding arc
DXX	Additive compensation number 1 –16 (see page 128)
G58	Contour-parallel oversize
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
B1, B2	Chamfer/rounding arc (B1 contour start; B2 contour end) ■ B>0: Rounding radius ■ B<0: Chamfer width
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Finishing**

By setting the following **optional parameters,** you can define additional contour elements:

- A:Oblique cut at contour start
- W:Oblique cut at contour end
- R:Rounding
- B1:Chamfer/Rounding at contour start
- B2:Chamfer/Rounding at contour end
- WS: Angle of the chamfer at the contour starting point (not yet implemented)
- WE: Angle of the chamfer at the contour end point (not yet implemented)

Cycle run

1. Move in longitudinal direction from the starting point to the **contour starting point X1, Z1**
2. Finish contour area from **contour starting point X1, Z1** to **end point X2, Z2,** taking optional contour elements into account
3. Move to the tool change point according to the G14 setting

Cut, longitudinal plunge



Select **cut, longitudinal/transverse**

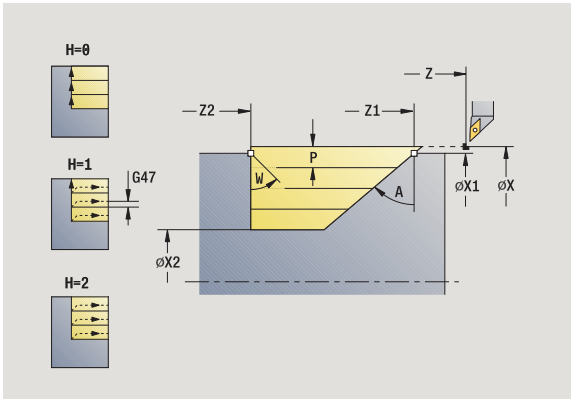


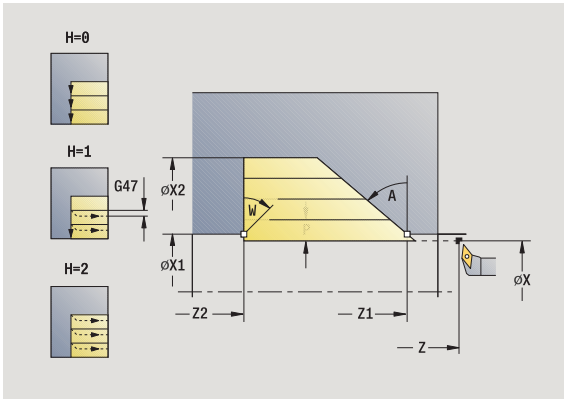
Select **plunge, longitudinal**

The cycle roughs the area described by the **contour starting point**, **contour end point** and **plunge angle**.



- The tool plunges with the maximum possible angle, leaving material remaining.
- The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50 %).





Cycle parameters

X, Z	Starting point
X1, Z1	Contour starting point
X2, Z2	Contour end point
P	Infeed depth: Maximum infeed depth
H	Contour smoothing ■ 0: With every cut ■ 1: With the last cut ■ 2: No finishing cut
A	Plunge angle (range: 0° <= A < 90°; default: 0°)
W	End angle—oblique cut at contour end (Range: 0° <= W < 90°)
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Roughing**

Cycle run

1. Calculate the proportioning of cuts (infeed)
2. Approach the workpiece from starting point for first pass
3. Plunge-cut at **plunging angle A** with reduced feed
4. Move at the programmed feed rate to the **contour end point Z2,** or to the oblique surface defined by the **end angle W**
5. The contour is machined depending on the **contour smoothing H**:
6. Return and approach again for next pass
7. Repeat 3 to 6 until **contour end point X2** is reached
8. Return to starting point on paraxial path
9. Move to the tool change point according to the G14 setting

Cut, transverse plunge



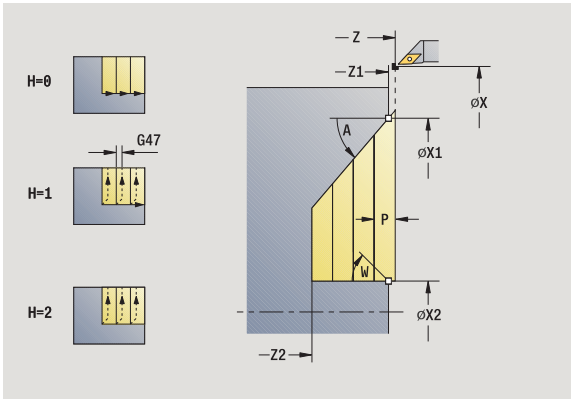
Select **cut, longitudinal/transverse**

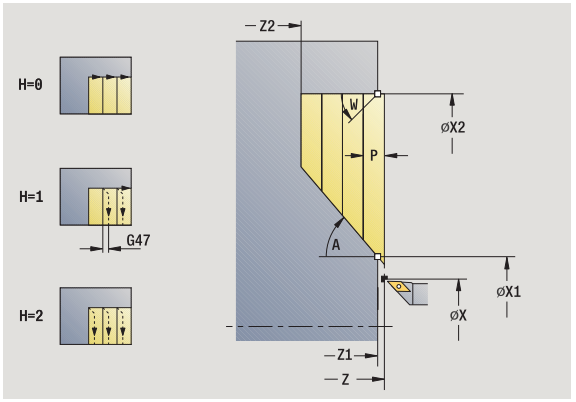


Select **plunge, transverse**

The cycle roughs the area described by the **contour starting point**, **contour end point** and **plunge angle**.

- The tool plunges with the maximum possible angle, leaving material remaining.
- The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50 %).





Cycle parameters

X, Z	Starting point
X1, Z1	Contour starting point
X2, Z2	Contour end point
P	Infeed depth: Maximum infeed depth
H	Contour smoothing ■ 0: With every cut ■ 1: With the last cut ■ 2: No finishing cut
A	Plunge angle (range: 0° <= A < 90°; default: 0°)
W	End angle—oblique cut at contour end (Range: 0° <= W < 90°)
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Roughing**

Cycle run

1. Calculate the proportioning of cuts (infeed)
2. Approach the workpiece from starting point for first pass
3. Plunge-cut at **plunging angle A** with reduced feed
4. Move at the programmed feed rate to the **contour end point X2** or to the oblique surface defined by the **end angle W**
5. The contour is machined depending on the **contour smoothing H**:
6. Return and approach again for next pass
7. Repeat 3 to 6 until **contour end point X2** is reached
8. Return to starting point on paraxial path
9. Move to the tool change point according to the G14 setting

Cut, longitudinal plunging—expanded



Select **cut, longitudinal/transverse**



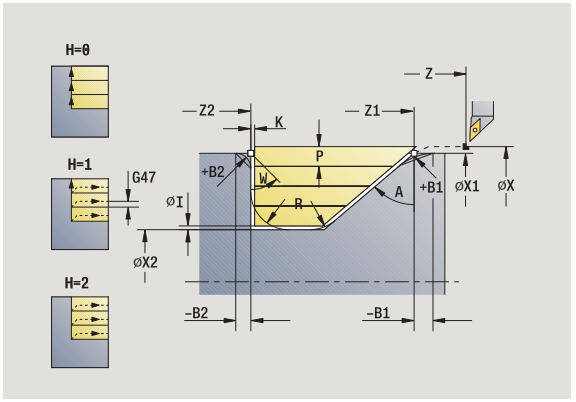
Select **plunge, longitudinal**

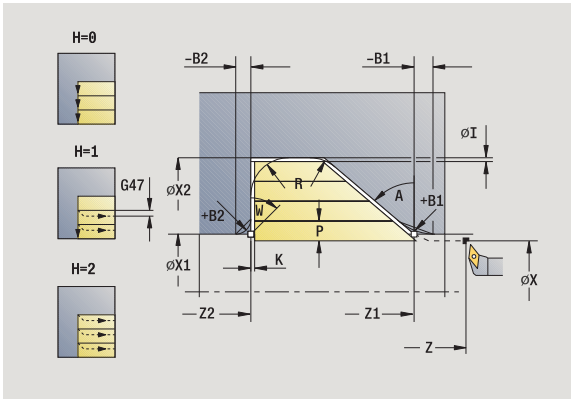
Expanded

Press the **Expanded** soft key

The cycle roughs the area described by the **contour starting point, contour end point** and **plunge angle,** taking the oversizes into consideration.

- The tool plunges with the maximum possible angle, leaving material remaining.
- The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50 %).





Cycle parameters

X, Z	Starting point
X1, Z1	Contour starting point
X2, Z2	Contour end point
P	Infeed depth: Maximum infeed depth
H	Contour smoothing ■ 0: With every cut ■ 1: With the last cut ■ 2: No finishing cut
I, K	Oversize X, Z
R	Rounding arc
A	Plunge angle (range: 0° <= A < 90°; default: 0°)
W	End angle—oblique cut at contour end (Range: 0° <= W < 90°)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
BP	Break duration: Time span for interruption of the feed. The chip is broken by the (intermittent) interruption of the feed.
BF	Feed duration: Time interval until the next break. The chip is broken by the (intermittent) interruption of the feed.
G47	Safety clearance (see page 128)
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Roughing**

By setting the following **optional parameters,** you can define additional contour elements:

- W:Oblique cut at contour end
- R:Rounding arc (in both corners of the contour valley)
- B1:Chamfer/Rounding at contour start
- B2:Chamfer/Rounding at contour end
- BP:Pause duration
- BF:Feed duration

Cycle run

1. Calculate the proportioning of cuts (infeed)
2. Approach the workpiece from starting point for first pass
3. Plunge-cut at **`plunging angle A`** with reduced feed
4. Move at the programmed feed rate to the **`contour end point Z2,`** or if defined, to one of the optional contour elements
5. The contour is machined depending on the **`contour smoothing H`**:
6. Return and approach for next pass
7. Repeat 3 to 6 until **`contour end point X2`** is reached
8. Return to starting point on paraxial path
9. Move to the tool change point according to the G14 setting

Cut, transverse plunging—expanded



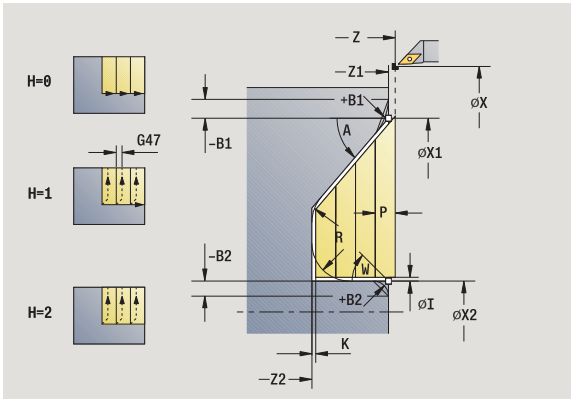
Select **cut, longitudinal/transverse**

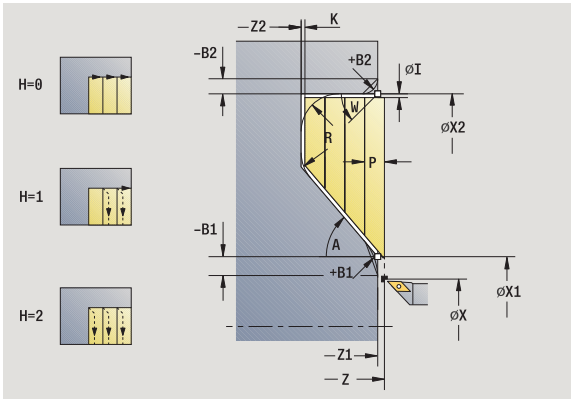


Select **plunge, transverse**

Expanded

Press the **Expanded** soft key





The cycle roughs the area described by the **contour starting point**, **contour end point** and **plunge angle**, taking the oversizes into consideration.

- The tool plunges with the maximum possible angle, leaving material remaining.
- The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50 %).

Cycle parameters

X, Z	Starting point
X1, Z1	Contour starting point
X2, Z2	Contour end point
P	Infeed depth: Maximum infeed depth
H	Contour smoothing ■ 0: With every cut ■ 1: With the last cut ■ 2: No finishing cut
I, K	Oversize X, Z
R	Rounding arc
A	Plunge angle (range: 0° <= A < 90°; default: 0°)
W	End angle—oblique cut at contour end (Range: 0° <= W < 90°)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
BP	Break duration: Time span for interruption of the feed. The chip is broken by the (intermittent) interruption of the feed.
BF	Feed duration: Time interval until the next break. The chip is broken by the (intermittent) interruption of the feed.
G47	Safety clearance (see page 128)
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Roughing**

By setting the following **optional parameters,** you can define additional contour elements:

- W:Oblique cut at contour end
- R:Rounding arc (in both corners of the contour valley)
- B1:Chamfer/Rounding at contour start
- B2:Chamfer/Rounding at contour end
- BP:Pause duration
- BF:Feed duration

Cycle run

1. Calculate the proportioning of cuts (infeed)
2. Approach the workpiece from starting point for first pass
3. Plunge-cut at **`plunging angle A`** with reduced feed
4. Move at the programmed feed rate to the **`contour end point X2,`** or if defined, to one of the optional contour elements
5. The contour is machined depending on the **`contour smoothing H`**:
6. Return and approach for next pass
7. Repeat 3 to 6 until **`contour end point Z2`** is reached
8. Return to starting point on paraxial path
9. Move to the tool change point according to the G14 setting

Cut, longitudinal finishing plunge



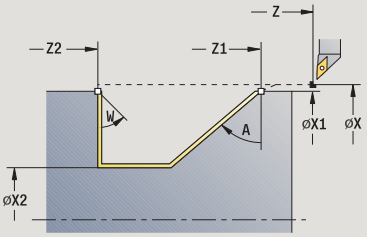
Select **cut, longitudinal/transverse**



Select **plunge, longitudinal**

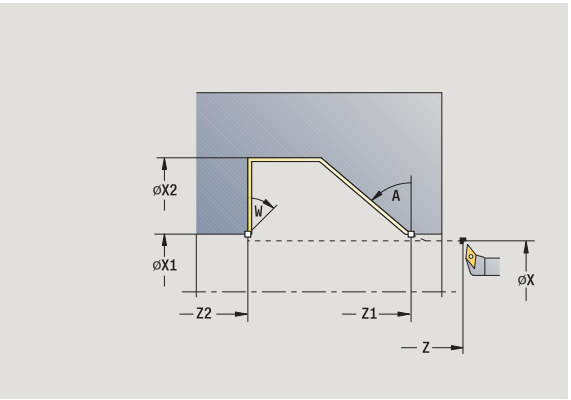
Finishing run

Press the **Finishing run** soft key



The cycle finishes the contour area from the **contour starting point** to the **contour end point**. At the end of the cycle, the tool returns to the starting point.

- The tool plunges with the maximum possible angle, leaving material remaining.
- The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50 %).



Cycle parameters

X, Z	Starting point
X1, Z1	Contour starting point
X2, Z2	Contour end point
A	Plunge angle (range: 0° <= A < 90°; default: 0°)
W	End angle—oblique cut at contour end (Range: 0° <= W < 90°)
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Finishing**

Cycle run

1. Move in transverse direction from the starting point to the **contour starting point X1, Z1**
2. Finish defined contour area
3. Return to starting point on paraxial path
4. Move to the tool change point according to the G14 setting

Cut, transverse finishing plunge



Select **cut, longitudinal/transverse**



Select **plunge, transverse**

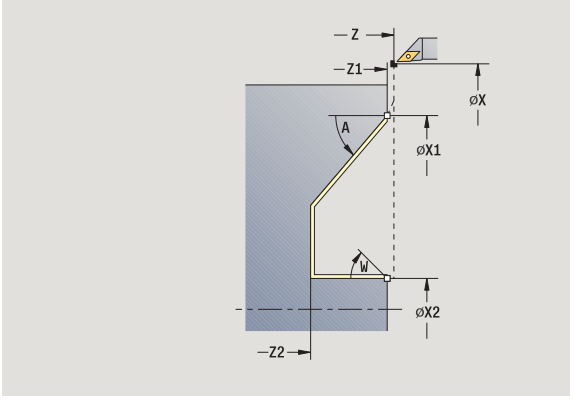
Finishing run

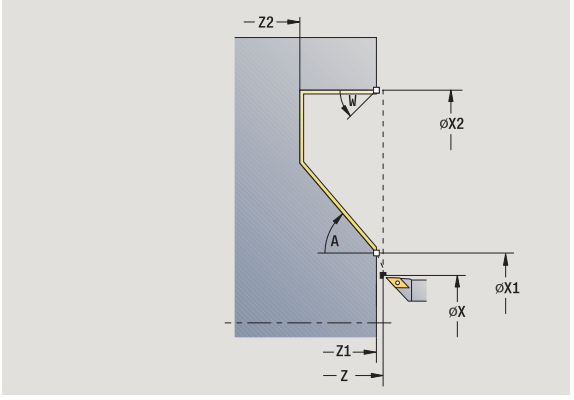
Press the **Finishing run** soft key

The cycle finishes the contour area from the **contour starting point** to the **contour end point**. At the end of the cycle, the tool returns to the starting point.



- The tool plunges with the maximum possible angle, leaving material remaining.
- The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50 %).





Cycle parameters

X, Z Starting point

X1, Z1 Contour starting point

X2, Z2 Contour end point

A Plunge angle (range: 0° <= A < 90°; default: 0°)

W End angle—oblique cut at contour end (Range: 0° <= W < 90°)

G47 Safety clearance (see page 128)

G14 Tool change point (see page 128)

T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent) ■ Main drive ■ Opposing spindle for rear-face machining

Type of machining for technology database access: **Finishing**

Cycle run

1. Move in transverse direction from the starting point to the **`contour starting point X1, Z1`**
2. Finish defined contour area
3. Return to starting point on paraxial path
4. Move to the tool change point according to the G14 setting

Cut, longitudinal finishing plunge—expanded



Select **cut, longitudinal/transverse**



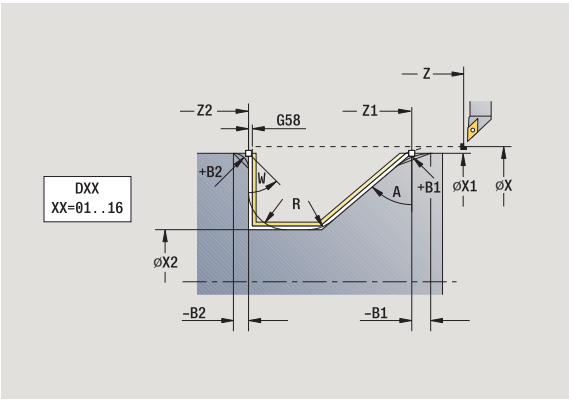
Select **plunge, longitudinal**

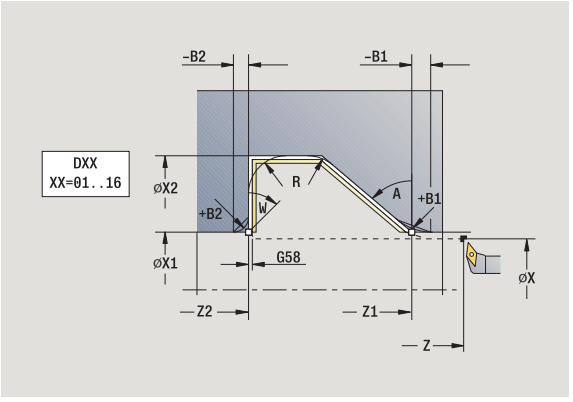
Expanded

Press the **Expanded** soft key

Finishing run

Press the **Finishing run** soft key





The cycle finishes the contour area from the **contour starting point** to the **contour end point**. When the cycle is completed, the tool remains at the cycle end position.

- The tool plunges with the maximum possible angle, leaving material remaining.
- The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50 %).

Cycle parameters

X, Z	Starting point
X1, Z1	Contour starting point
X2, Z2	Contour end point
DXX	Additive compensation number 1 –16 (see page 128)
G58	Contour-parallel oversize
A	Plunge angle (range: 0° <= A < 90°; default: 0°)
W	End angle—oblique cut at contour end (Range: 0° <= W < 90°)
R	Rounding arc
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
B1, B2	Chamfer/rounding arc (B1 contour start; B2 contour end) ■ B>0: Rounding radius ■ B<0: Chamfer width
G47	Safety clearance (see page 128)
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Finishing**

By setting the following **optional parameters,** you can define additional contour elements:

- W:Oblique cut at contour end
- R:Rounding arc (in both corners of the contour valley)
- B1:Chamfer/Rounding at contour start
- B2:Chamfer/Rounding at contour end

Cycle run

1. Move paraxially from the starting point to the **`contour starting point X1, Z1`**
2. Finish the defined contour area, taking optional contour elements into account
3. Move to the tool change point according to the G14 setting

Cut, transverse finishing plunge—expanded



Select **cut, longitudinal/transverse**



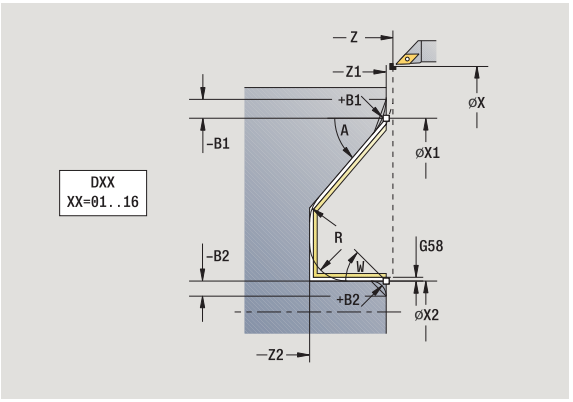
Select **plunge, transverse**

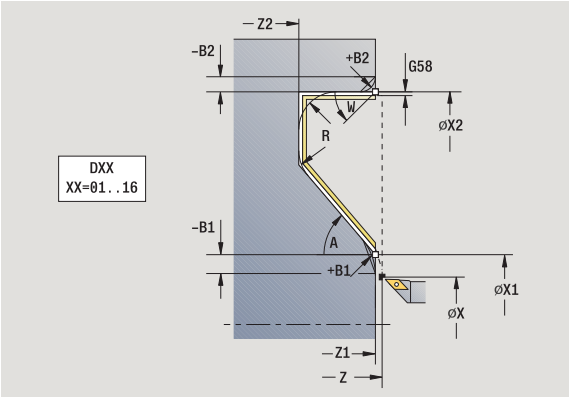
Expanded

Press the **Expanded** soft key

Finishing run

Press the **Finishing run** soft key





The cycle finishes the contour area from the **contour starting point** to the **contour end point**. When the cycle is completed, the tool remains at the cycle end position.



- The tool plunges with the maximum possible angle, leaving material remaining.
- The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50 %).

Cycle parameters

X, Z	Starting point
X1, Z1	Contour starting point
X2, Z2	Contour end point
DXX	Additive compensation number 1 –16 (see page 128)
G58	Contour-parallel oversize
A	Plunge angle (range: 0° <= A < 90°; default: 0°)
W	End angle—oblique cut at contour end (Range: 0° <= W < 90°)
R	Rounding arc
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
B1, B2	Chamfer/rounding arc (B1 contour start; B2 contour end) ■ B>0: Rounding radius ■ B<0: Chamfer width
G47	Safety clearance (see page 128)
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Finishing**

By setting the following **optional parameters,** you can define additional contour elements:

- W:Oblique cut at contour end
- R:Rounding arc (in both corners of the contour valley)
- B1:Chamfer/Rounding at contour start
- B2:Chamfer/Rounding at contour end

Cycle run

1. Move paraxially from the starting point to the **`contour starting point X1, Z1`**
2. Finish the defined contour area, taking optional contour elements into account
3. Move to the tool change point according to the G14 setting

Cut, ICP contour-parallel, longitudinal



Select **cut, longitudinal/transverse**



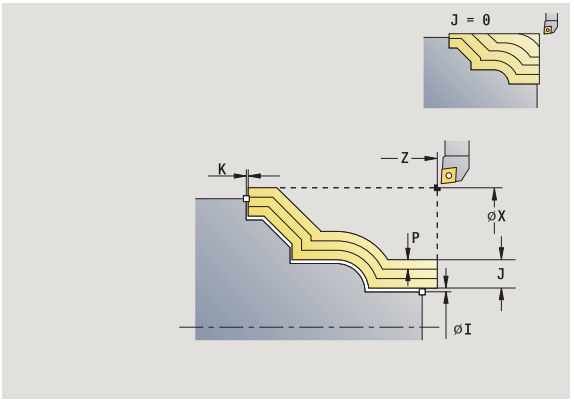
Select **ICP contour-parallel, longitudinal**

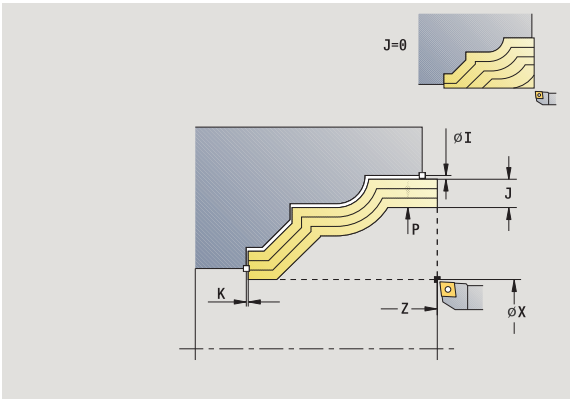
The cycle roughs the defined area on contour-parallel paths.

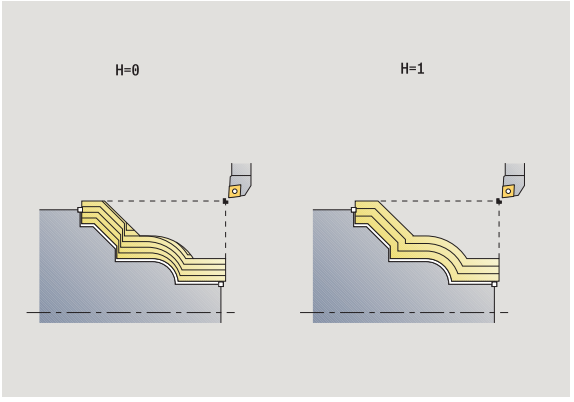
- The cycle roughs contour parallel depending on the **workpiece blank oversize J** and the **type of cutting lines H**:
  - J=0: the area defined by X, Z and the ICP contour, taking the oversizes into account.
  - J>0: The area defined by the ICP contour (plus oversizes) and the **workpiece blank oversize J**.
- The tool plunges with the maximum possible angle, leaving material remaining.

Danger of collision!

For **workpiece blank oversize J>0**: Set the **infeed depth P** to the smaller infeed, if the maximum infeed differs for the longitudinal and transverse directions due to the cutting geometry.



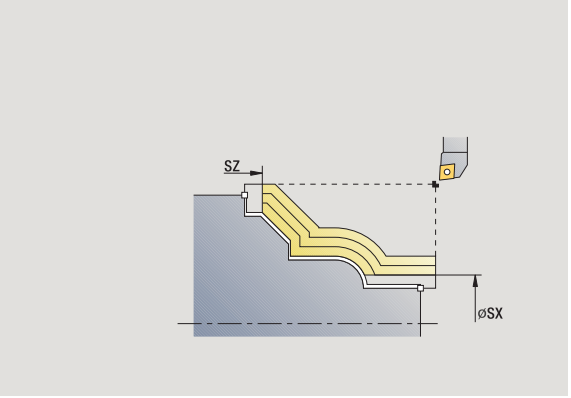




Cycle parameters

X, Z	Starting point
FK	ICP finished part: Name of the contour to be machined
P	Infeed depth—is determined taking J into account ■ J=0: P is the maximum infeed depth. The cycle reduces the infeed depth if the programmed infeed is not possible in the transverse or longitudinal direction due to the cutting geometry. ■ J>0: P is the infeed depth. This infeed is used in the longitudinal and transverse directions.
H	Type of cutting lines—the cycle machines ■ 0: with constant cutting depth ■ 1: with equidistant cutting lines
I, K	Oversize X, Z
J	Workpiece blank oversize—the cycle machines ■ J=0: From the current tool position ■ J>0: The area defined by the workpiece blank oversize
HR	Specify primary machining direction
SX, SZ	Cutting limits (see page 128)

G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
BP	Break duration: Time span for interruption of the feed. The chip is broken by the (intermittent) interruption of the feed.
BF	Feed duration: Time interval until the next break. The chip is broken by the (intermittent) interruption of the feed.
A	Approach angle (reference: Z-axis)—(default: parallel to Z axis)
W	Departure angle (reference: Z axis)—(default: orthogonal to Z axis)
XA, ZA	Starting point of blank (only effective if no blank was programmed): ■ XA, ZA not programmed: The workpiece blank contour is calculated from the tool position and the ICP contour. ■ XA, ZA programmed: Definition of the corner point of the workpiece blank
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent) ■ Main drive ■ Opposing spindle for rear-face machining



Type of machining for technology database access: **Roughing**

Cycle run

1. Calculate the proportioning of cuts (infeed), taking the **workpiece blank oversize J** and the **type of cutting lines H** into account
   - J=0: The cutting geometry is taken into account. This may result in the use of different infeeds for the longitudinal and transverse directions.
   - J>0: The same infeed is used for both the longitudinal and the transverse direction.
2. Approach the workpiece from starting point for first pass
3. Machine the workpiece according to the calculated proportioning of cuts
4. Return and approach for next pass
5. Repeat 3 to 4 until the defined area has been machined
6. Return to starting point on paraxial path
7. Move to the tool change point according to the G14 setting

Cut, ICP contour-parallel, transverse



Select **cut, longitudinal/transverse**



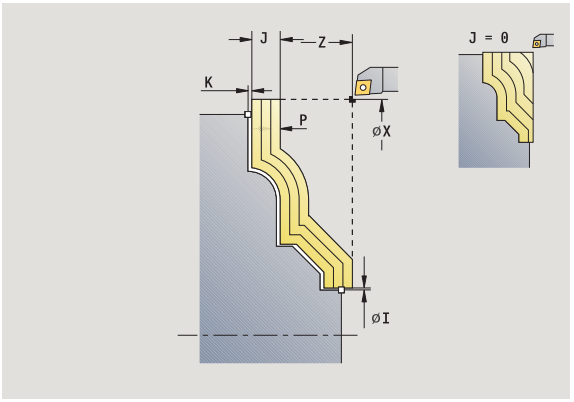
Select **ICP contour-parallel, transverse**

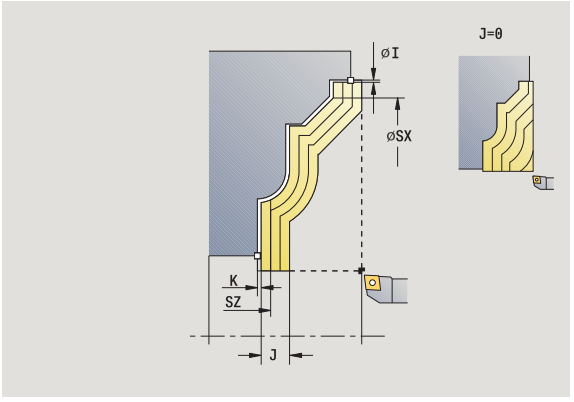
The cycle roughs the defined area on contour-parallel paths.

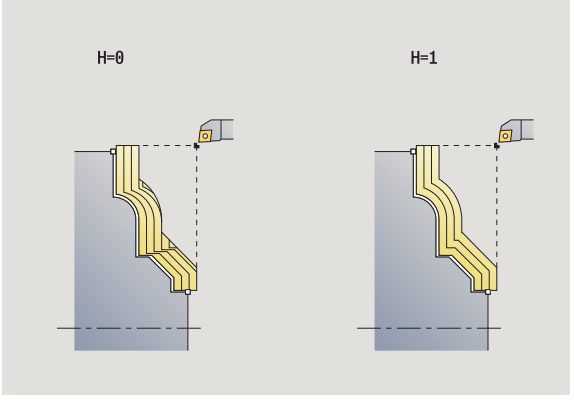
- The cycle roughs **contour parallel** depending on the **workpiece blank oversize J** and the **type of cutting lines H:**
  - J=0: the area defined by X, Z and the ICP contour, taking the oversizes into account.
  - J>0: The area defined by the ICP contour (plus oversizes) and the **workpiece blank oversize J**.
- The tool plunges with the maximum possible angle, leaving material remaining.

Danger of collision!

For **workpiece blank oversize J>0**: Set the **infeed depth P** to the smaller infeed, if the maximum infeed differs for the longitudinal and transverse directions due to the cutting geometry.

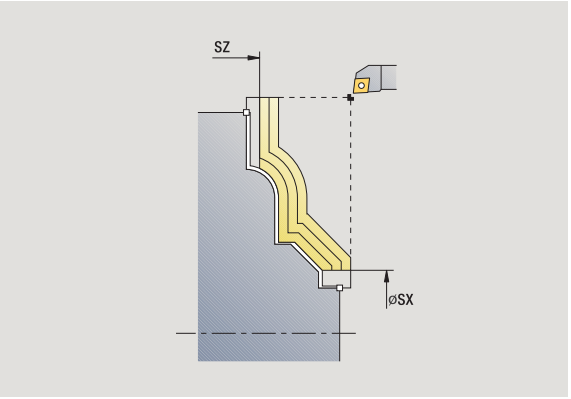






Cycle parameters

X, Z	Starting point
FK	ICP finished part: Name of the contour to be machined
P	Infeed depth—is determined taking J into account ■ J=0: P is the maximum infeed depth. The cycle reduces the infeed depth if the programmed infeed is not possible in the transverse or longitudinal direction due to the cutting geometry. ■ J>0: P is the infeed depth. This infeed is used in the longitudinal and transverse directions.
H	Type of cutting lines—the cycle machines ■ 0: with constant cutting depth ■ 1: with equidistant cutting lines
I, K	Oversize X, Z
J	Workpiece blank oversize—the cycle machines ■ J=0: From the current tool position ■ J>0: The area defined by the workpiece blank oversize
HR	Specify primary machining direction
SX, SZ	Cutting limits (see page 128)



G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
BP	Break duration: Time span for interruption of the feed. The chip is broken by the (intermittent) interruption of the feed.
BF	Feed duration: Time interval until the next break. The chip is broken by the (intermittent) interruption of the feed.
XA, ZA	Starting point of blank (only effective if no blank was programmed): ■ XA, ZA not programmed: The workpiece blank contour is calculated from the tool position and the ICP contour. ■ XA, ZA programmed: Definition of the corner point of the workpiece blank.
A	Departure angle (reference: Z axis)—(default: orthogonal to Z axis)
W	Departure angle (reference: Z-axis)—(default: parallel to Z axis)
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent) ■ Main drive ■ Opposing spindle for rear-face machining

Type of machining for technology database access: **Roughing**

Cycle run

1 Calculate the proportioning of cuts (infeed), taking the **workpiece blank oversize J** into account
- J=0: The cutting geometry is taken into account. This may result in the use of different infeeds for the longitudinal and transverse directions.
- J>0: The same infeed is used for both the longitudinal and the transverse direction.

2 Approach the workpiece from starting point for first pass

3 Machine the workpiece according to the calculated proportioning of cuts

4 Return and approach for next pass

5 Repeat 3 to 4 until the defined area has been machined

6 Return to starting point on paraxial path

7 Move to the tool change point according to the G14 setting

Cut, ICP contour-parallel, longitudinal finishing



Select **cut, longitudinal/transverse**



Select **ICP contour-parallel, longitudinal**

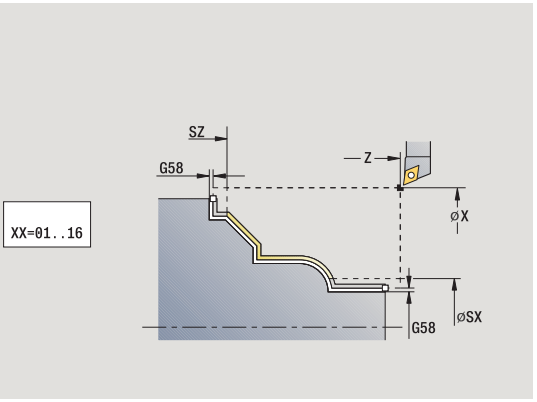
Finishing run

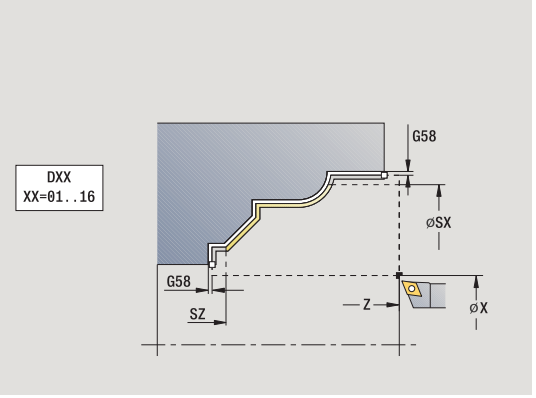
Press the **Finishing run** soft key

The cycle finishes the contour area defined by the ICP contour. When the cycle is completed, the tool remains at the cycle end position.



The tool plunges with the maximum possible angle, leaving material remaining.





Cycle parameters

X, Z	Starting point
FK	ICP finished part: Name of the contour to be machined
DXX	Additive compensation number 1 –16 (see page 128)
G58	Contour-parallel oversize
DI	Axis-parallel oversize X
DK	Axis-parallel oversize Z
SX, SZ	Cutting limits (see page 128)
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Finishing**

Cycle run

1. Move paraxially from the starting point to the ICP contour starting point
2. Finish defined contour area
3. Move to the tool change point according to the G14 setting

Cut, ICP contour-parallel, transverse finishing



Select **cut, longitudinal/transverse**

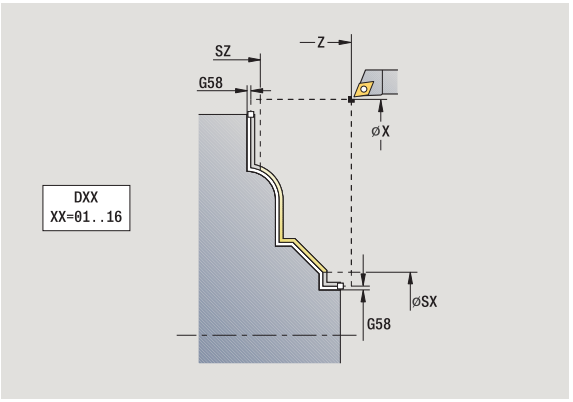


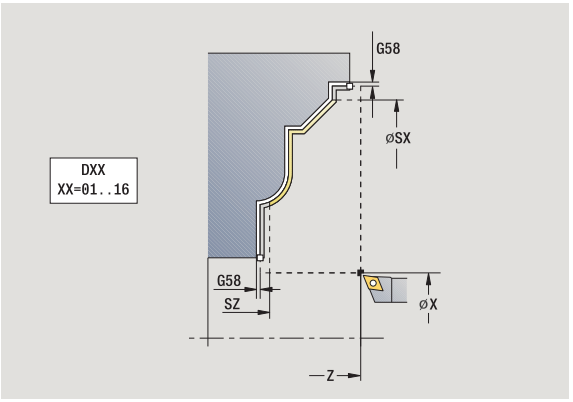
Select **ICP contour-parallel, transverse**

Finishing run

Press the **Finishing run** soft key

The cycle finishes the contour area defined by the ICP contour. When the cycle is completed, the tool remains at the cycle end position.







The tool plunges with the maximum possible angle, leaving material remaining.

Cycle parameters

X, Z	Starting point
FK	ICP finished part: Name of the contour to be machined
DXX	Additive compensation number 1 –16 (see page 128)
G58	Contour-parallel oversize
DI	Axis-parallel oversize X
DK	Axis-parallel oversize Z
SX, SZ	Cutting limits (see page 128)
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Finishing**

Cycle run

1 Move paraxially from the starting point to the ICP contour starting point

2 Finish defined contour area

3 Move to the tool change point according to the G14 setting

ICP cutting, longitudinal



Select **cut, longitudinal/transverse**

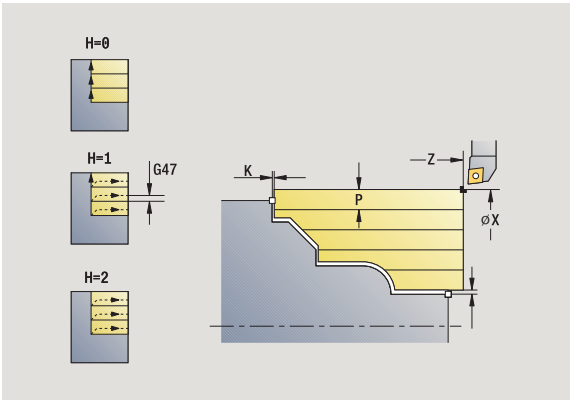


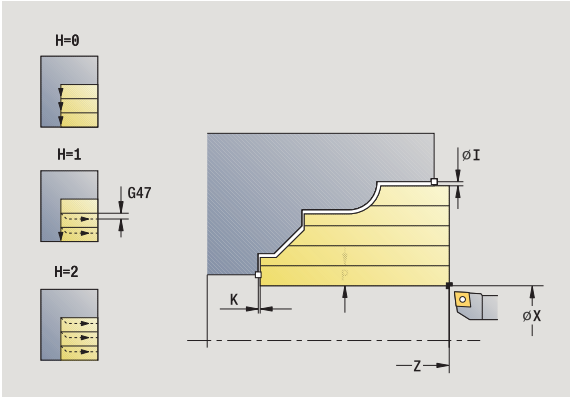
Select **ICP cutting, longitudinal**

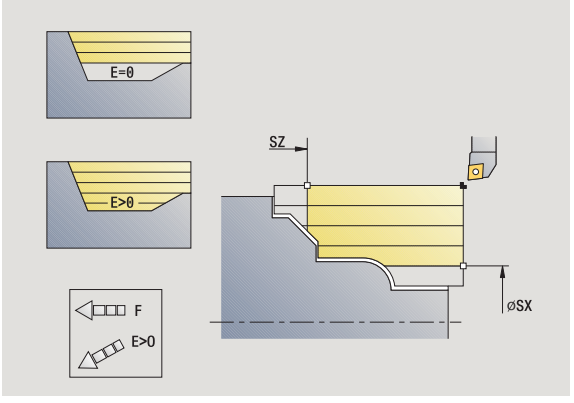
The cycle machines the area defined by the starting point and the ICP contour, taking the oversizes into account.



- The tool plunges with the maximum possible angle, leaving material remaining.
- The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50 %).







Cycle parameters

X, Z	Starting point
FK	ICP finished part: Name of the contour to be machined
P	Infeed depth: Maximum infeed depth
H	Contour smoothing ■ 0: With every cut ■ 1: With the last cut ■ 2: No finishing cut
I, K	Oversize X, Z
E	Plunging behavior: ■ No input: Automatic feed-rate reduction ■ E=0: No plunging ■ E>0: Plunging feed rate in use
SX, SZ	Cutting limits (see page 128)
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
BP	Break duration: Time span for interruption of the feed. The chip is broken by the (intermittent) interruption of the feed.
BF	Feed duration: Time interval until the next break. The chip is broken by the (intermittent) interruption of the feed.
A	Approach angle (reference: Z-axis)—(default: parallel to Z axis)
W	Departure angle (reference: Z axis)—(default: orthogonal to Z axis)

XA, ZA Starting point of blank (only effective if no blank was programmed):

- XA, ZA not programmed: The workpiece blank contour is calculated from the tool position and the ICP contour.
- XA, ZA programmed: Definition of the corner point of the workpiece blank.

MT M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Roughing**

Cycle run

1. Calculate the proportioning of cuts (infeed)
2. Approach the workpiece from starting point for first pass
3. For sloping contours, plunge into the material at reduced feed rate
4. Machine the workpiece according to the calculated proportioning of cuts
5. The contour is machined depending on the **`contour smoothing H`**:
6. Return and approach for next pass
7. Repeat 3 to 6 until the defined area has been machined
8. Return to starting point on paraxial path
9. Move to the tool change point according to the G14 setting

ICP cutting, transverse



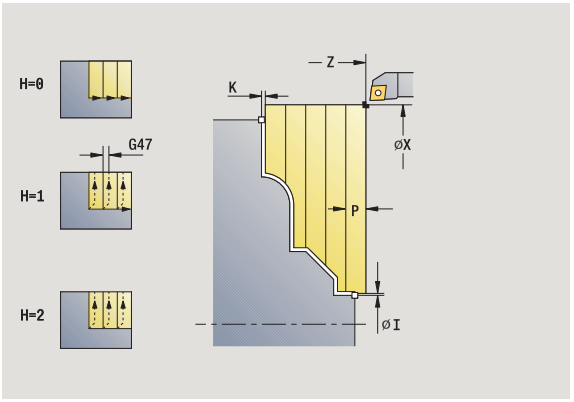
Select **cut, longitudinal/transverse**

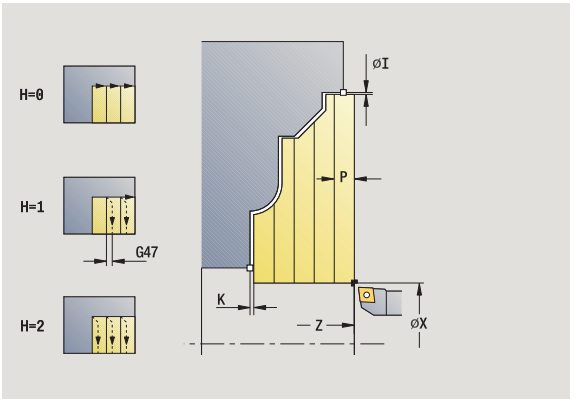


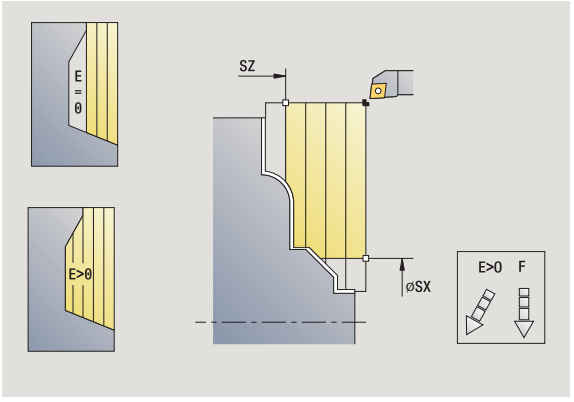
Select **ICP cutting, transverse**

The cycle machines the area defined by the starting point and the ICP contour, taking the oversizes into account.

- The tool plunges with the maximum possible angle, leaving material remaining.
- The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50 %).







Cycle parameters

X, Z	Starting point
FK	ICP finished part: Name of the contour to be machined
P	Infeed depth: Maximum infeed depth
H	Contour smoothing ■ 0: With every cut ■ 1: With the last cut ■ 2: No finishing cut
I, K	Oversize X, Z
E	Plunging behavior: ■ No input: Automatic feed-rate reduction ■ E=0: No plunging ■ E>0: Plunging feed rate in use
SX, SZ	Cutting limits (see page 128)
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
BP	Break duration: Time span for interruption of the feed. The chip is broken by the (intermittent) interruption of the feed.
BF	Feed duration: Time interval until the next break. The chip is broken by the (intermittent) interruption of the feed.
XA, ZA	Starting point of blank (only effective if no blank was programmed): ■ XA, ZA not programmed: The workpiece blank contour is calculated from the tool position and the ICP contour. ■ XA, ZA programmed: Definition of the corner point of the workpiece blank.

A	Departure angle (reference: Z axis)—(default: orthogonal to Z axis)
W	Departure angle (reference: Z-axis)—(default: parallel to Z axis)
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent) ■ Main drive ■ Opposing spindle for rear-face machining

Type of machining for technology database access: **Roughing**

Cycle run

1 Calculate the proportioning of cuts (infeed)
2 Approach the workpiece from starting point for first pass
3 For sloping contours, plunge into the material at reduced feed rate
4 Machine the workpiece according to the calculated proportioning of cuts
5 The contour is machined depending on the **`contour smoothing H`**:
6 Return and approach for next pass
7 Repeat 3 to 6 until the defined area has been machined
8 Return to starting point on paraxial path
9 Move to the tool change point according to the G14 setting

ICP longitudinal finishing cut



Select **cut, longitudinal/transverse**

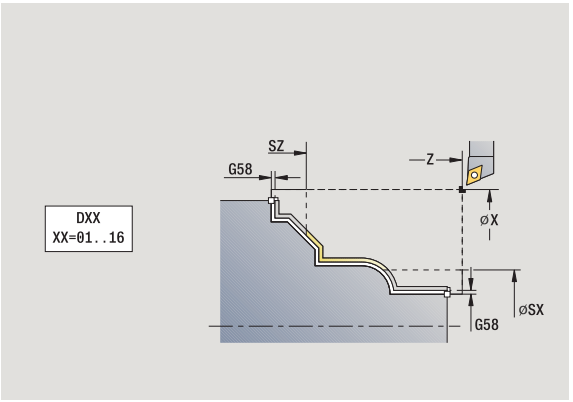


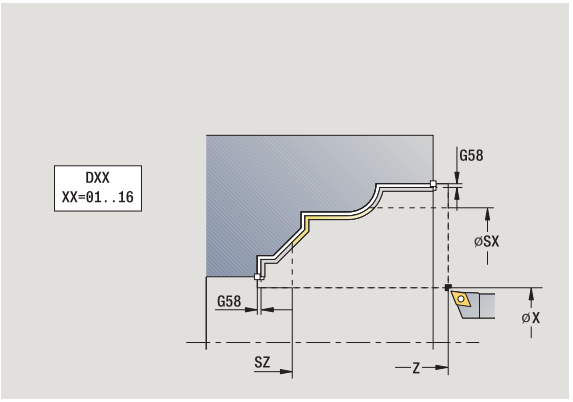
Select **ICP cutting, longitudinal**

Finishing run

Press the **Finishing run** soft key

The cycle finishes the contour area defined by the ICP contour. When the cycle is completed, the tool remains at the cycle end position.







The tool plunges with the maximum possible angle, leaving material remaining.

Cycle parameters

X, Z	Starting point
FK	ICP finished part: Name of the contour to be machined
DXX	Additive compensation number 1 –16 (see page 128)
G58	Contour-parallel oversize
DI	Axis-parallel oversize X
DK	Axis-parallel oversize Z
SX, SZ	Cutting limits (see page 128)
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Finishing**

Cycle run

1 Move paraxially from the starting point to the ICP contour starting point

2 Finish defined contour area

3 Move to the tool change point according to the G14 setting

ICP transverse finishing cut



Select **cut, longitudinal/transverse**

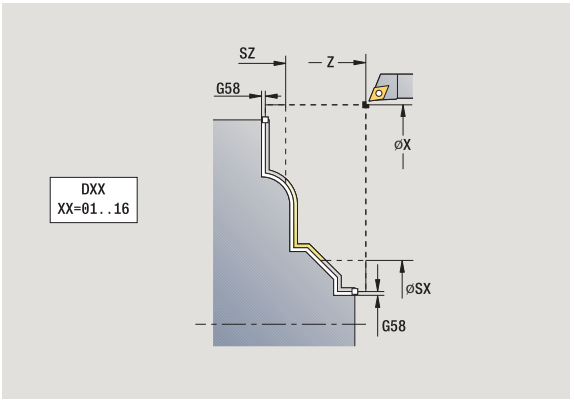


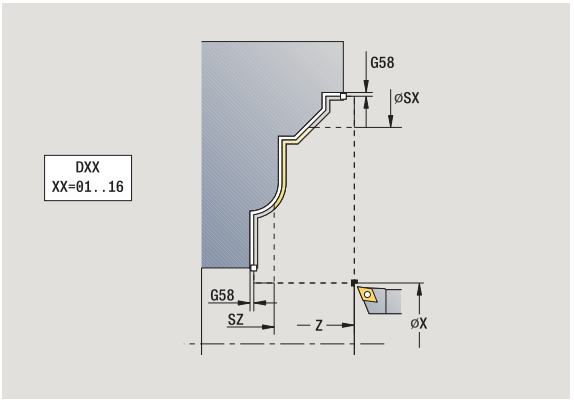
Select **ICP cutting, transverse**

Finishing run

Press the **Finishing run** soft key

The cycle finishes the contour area defined by the ICP contour. When the cycle is completed, the tool remains at the cycle end position.





The tool plunges with the maximum possible angle, leaving material remaining.

Cycle parameters

X, Z	Starting point
FK	ICP finished part: Name of the contour to be machined
DXX	Additive compensation number 1 –16 (see page 128)
G58	Contour-parallel oversize
DI	Axis-parallel oversize X
DK	Axis-parallel oversize Z
SX, SZ	Cutting limits (see page 128)
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Finishing**

Cycle run

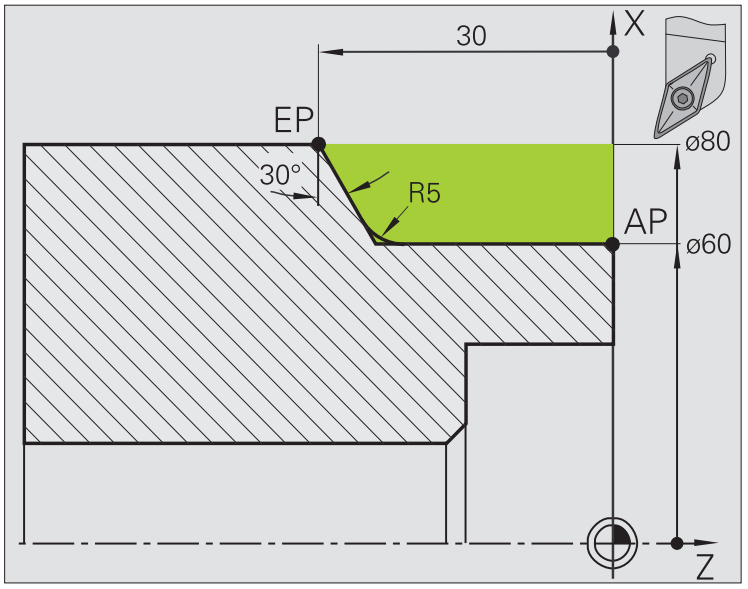
1 Move paraxially from the starting point to the ICP contour starting point

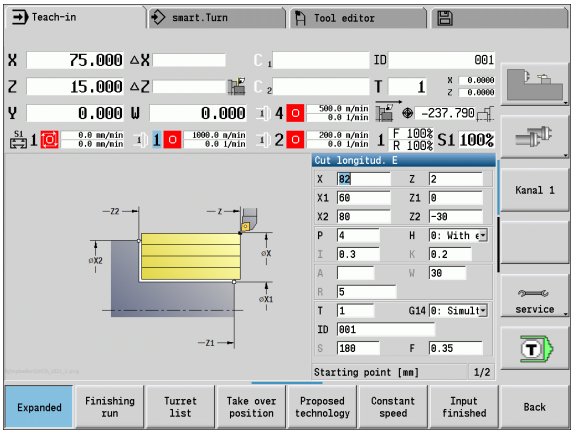
2 Finish defined contour area

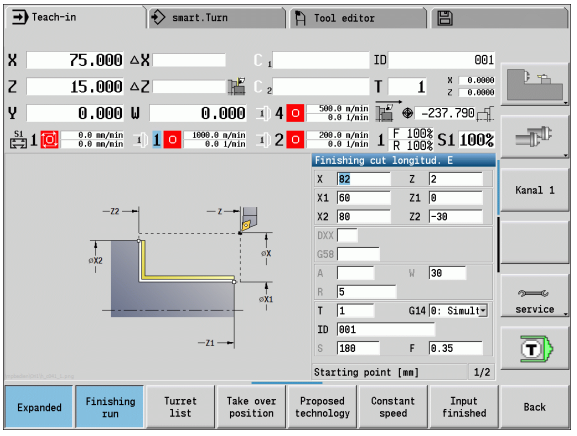
3 Move to the tool change point according to the G14 setting

Examples of turning cycles

Roughing and finishing an outside contour







The shaded area from **AP** (starting point of contour) to **EP** (contour end point) is rough-machined with the cycle **Plunge longitudinal–expanded,** taking oversizes into account. This contour area is to be finished subsequently with the cycle **Finishing cut longitudinal–expanded.**

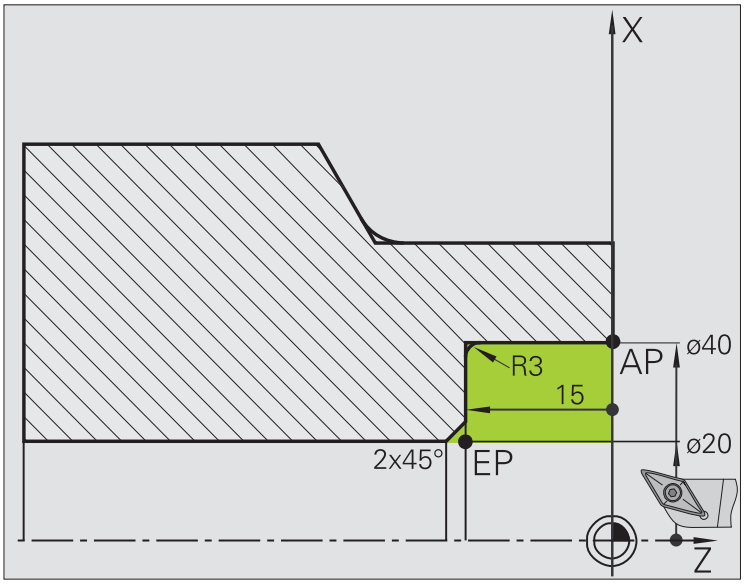
The rounding arc and the oblique cut at the contour end are also machined in "expanded mode."

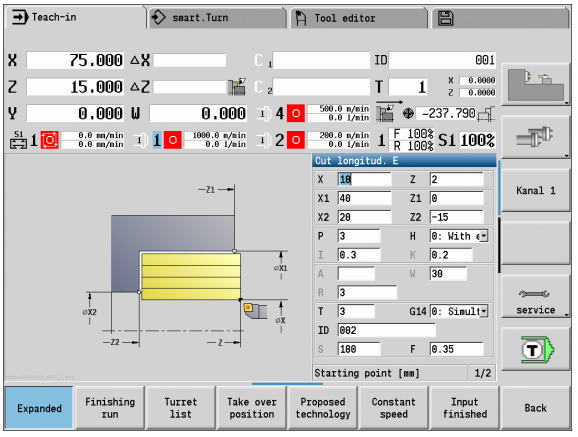
The parameters for **contour starting point X1, Z1** and **contour end point X2, Z2** determine the cutting and infeed directions—in this example, external machining and infeed in negative X-axis direction.

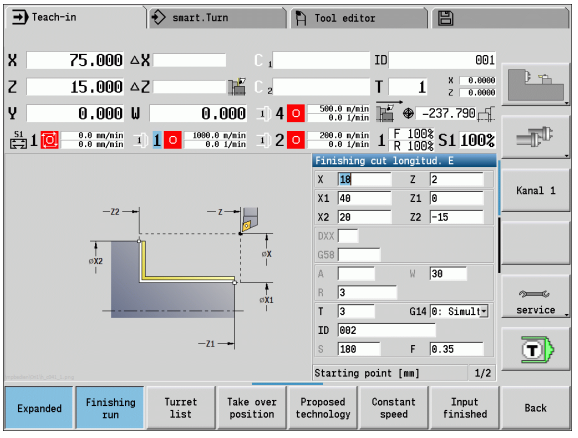
Tool data

- Turning tool (for external machining)
- WO = 1 (tool orientation)
- A = 93° (tool angle)
- B = 55° (point angle)

Roughing and finishing an inside contour







The shaded area from **AP** (starting point of contour) to **EP** (contour end point) is rough-machined with the cycle **Plunge longitudinal–expanded,** taking oversizes into account. This contour area is to be finished subsequently with the cycle **Finishing cut longitudinal–expanded.**

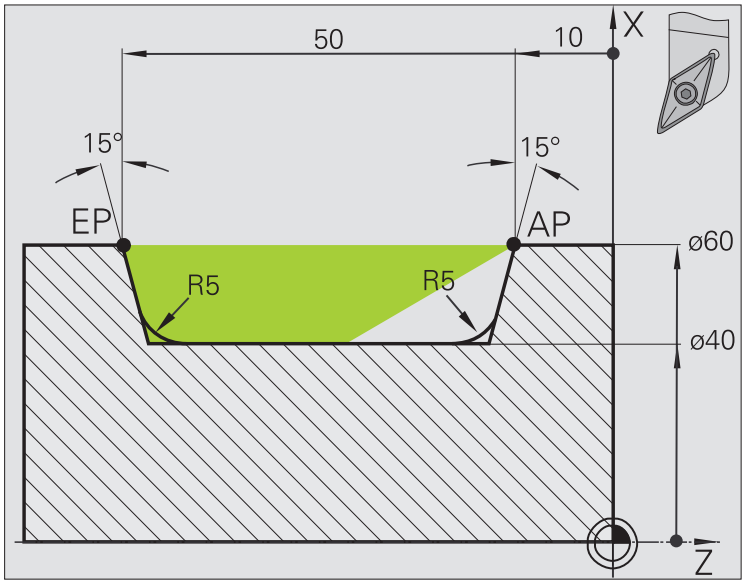
The rounding arc and the chamfer at the contour end are also machined in "expanded mode."

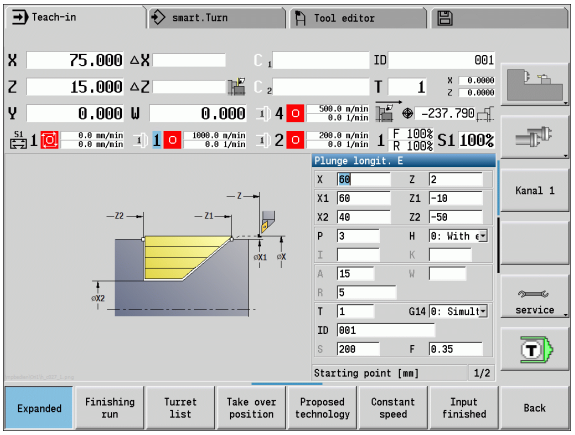
The parameters for **contour starting point X1, Z1** and **contour end point X2, Z2** determine the cutting and infeed directions—in this example, internal machining and infeed in negative X-axis direction.

Tool data

- Turning tool (for internal machining)
- WO = 7 (tool orientation)
- A = 93° (tool angle)
- B = 55° (point angle)

Roughing (recess clearance) with plunge cycle





The tool to be used cannot plunge at the required angle of 15°. The roughing process for the area therefore requires two steps.

First step:

The shaded area from **AP** (starting point of contour) to **EP** (contour end point) is rough-machined with the cycle **`Plunge longitudinal–expanded,`** taking oversizes into account.

The **`starting angle A`** is defined with 15°, as specified in the workpiece drawing. From the tool parameters, the CNC PILOT automatically calculates the maximum plunging angle that can be achieved with the programmed tool. The resulting contour will not be complete and will be reworked in the second step.

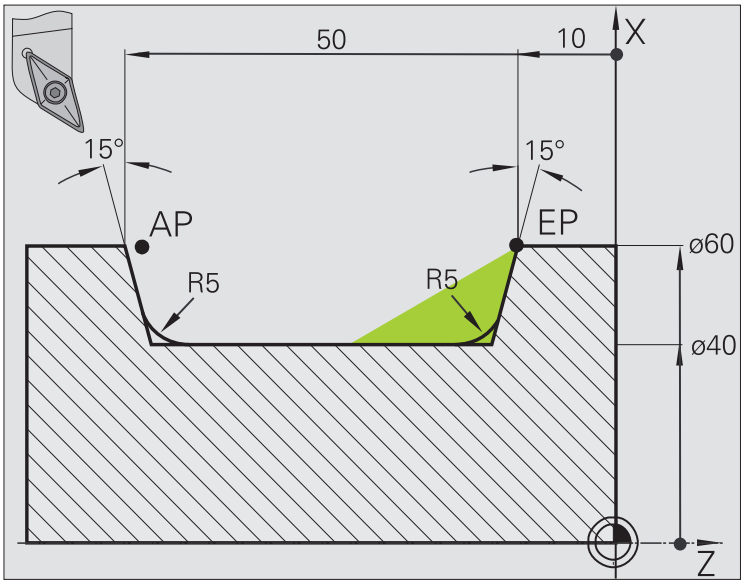
The rounding arcs in the contour valley are also machined in "expanded mode."

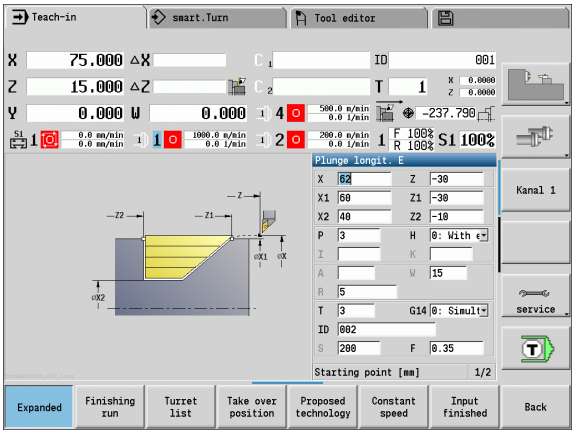
Be sure to enter the correct values for the parameters **`contour starting point X1, Z1`** and **`contour end point X2, Z2.`** These parameters determine the cutting and infeed directions—in this example, external machining and infeed in negative X-axis direction.

Tool data

- Turning tool (for external machining)
- WO = 1 (tool orientation)
- A = 93° (tool angle)
- B = 55° (point angle)

Second step:





The area that was left out in the first step (shaded area in the figure) is machined with the cycle **Plunge, longitudinal–expanded.** Before executing this step, you must change tools.

The rounding arcs in the contour valley are also machined in "expanded mode."

The parameters for **contour starting point X1, Z1** and **contour end point X2, Z2** determine the cutting and infeed directions—in this example, external machining and infeed in negative X-axis direction.

The parameter for the **contour starting point Z1** was determined during simulation of the first step.

Tool data

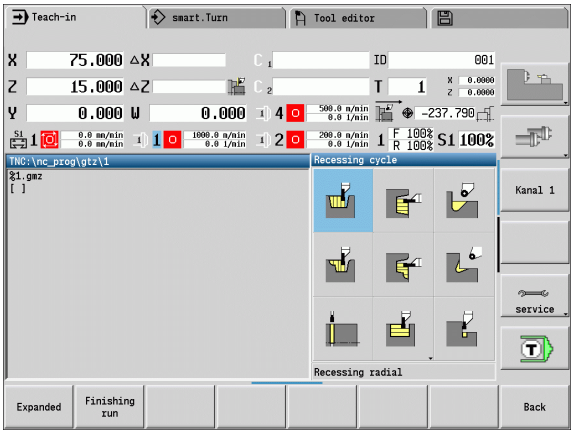
- Turning tool (for external machining)
- WO = 3 (tool orientation)
- A = 93° (tool angle)
- B = 55° (point angle)

4.5 Recessing cycles



The recessing cycle group comprises recessing, recess turning, undercut and parting cycles. Simple contours are machined in **basic mode,** complex contours in **expanded mode.** ICP recessing cycles machine contours defined with **ICP** (see "ICP contours" on page 360).

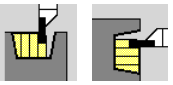
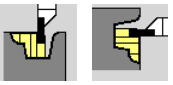
- **Proportioning of cuts:** The CNC PILOT calculates a constant recessing width that is <=**infeed depth P.**
- **Oversizes** are considered in "expanded" mode
- **Cutter radius compensation** is conducted (exception: Undercut type K).



Cutting and infeed directions for recessing cycles

The CNC PILOT automatically determines the cutting and infeed directions from the cycle parameters. The decisive ones are:

- **Basic mode:** The parameters for starting point X, Z (Manual mode: current tool position) and contour starting point X1 / contour end point Z2
- **Expanded mode:** The parameters for contour starting point X1, Z1 and contour end point X2, Z2
- **ICP cycles:** The parameters for starting point X, Z (Manual mode: current tool position) and starting point of the ICP contour

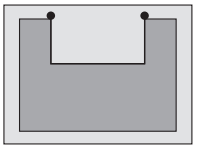
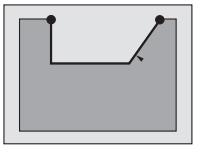
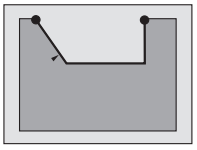
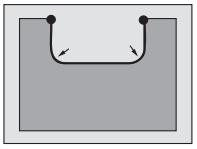
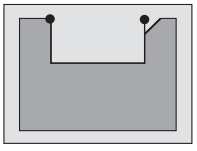
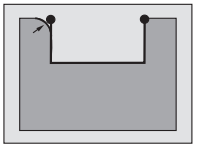
Recessing cycles	Symbol
Recessing radial/axial Recessing and finishing cycles for simple contours	
ICP recessing, radial/axial Recessing and finishing cycles for any contour	
Recess turning, radial/axial Recess-turning and finishing cycles for simple contours and any type of contour	
Undercut H Undercut type H	
Undercut K Undercut type K	
Undercut U Undercut type U	
Parting Cycle for parting the workpiece	

Undercut position

The CNC PILOT determines the position of an undercut from the cycle parameters for **`starting point X, Z`** (current tool position in Manual mode) and **`corner point of contour X1, Z1.`**

Undercuts can only be executed in orthogonal, paraxial contour corners along the longitudinal axis.

Contour forms

Contour elements in recessing cycles	
Basic mode Machining a rectangular area	
Expanded mode Oblique cut at contour start	
Expanded mode Oblique cut at contour end	
Expanded mode Rounding arc in both corners of the contour valley	
Expanded mode Chamfer (or rounding arc) at contour start	
Expanded mode Chamfer (or rounding arc) at contour end	

Recessing, radial

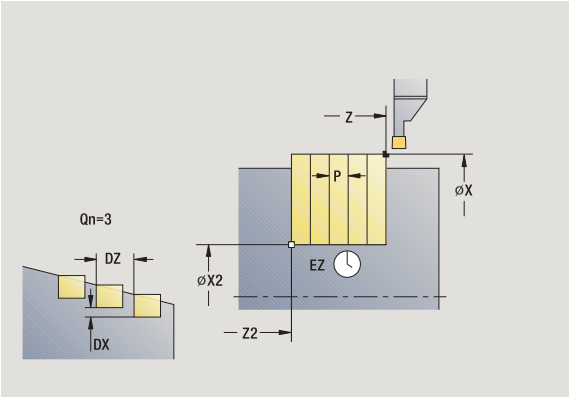


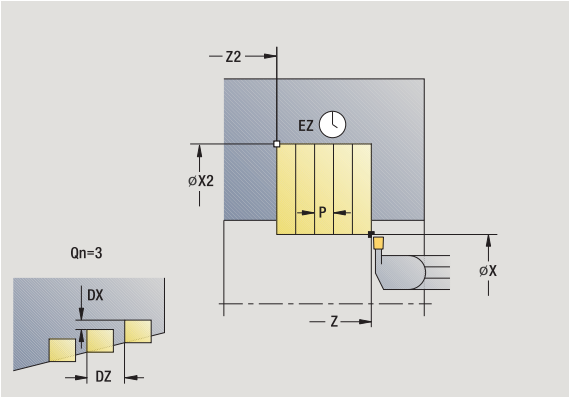
Call the **recessing cycles**



Select **recessing, radial**

The cycle machines the number of recesses defined in **number Qn.** The parameters for **starting point** and **end point of contour** define the first recess (position, recess depth and recess width).





Cycle parameters

X, Z	Starting point
X2, Z2	Contour end point
P	Recessing width: infeeds <= P (no input: P = 0.8 * cutting width of the tool)
EZ	Dwell time for chip breaking (default: length of time for two revolutions)
Qn	Number of recess cycles (default: 1)
DX, DZ	Distance to subsequent recess with respect to the preceding recess
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **contour recessing**

Cycle run

1. Calculate the recess positions and the proportioning of cuts
2. Approach the workpiece for next recess from starting point or from last recess on paraxial path
3. Move at the programmed feed rate to the **contour end point X2**
4. Remain at this position for **dwell time EZ**
5. Retract and approach for next pass
6. Repeat 3 to 5 until the complete recess has been machined
7. Repeat 2 to 6 until all recesses have been machined
8. Return to starting point on paraxial path
9. Move to the tool change point according to the G14 setting

Recessing, axial

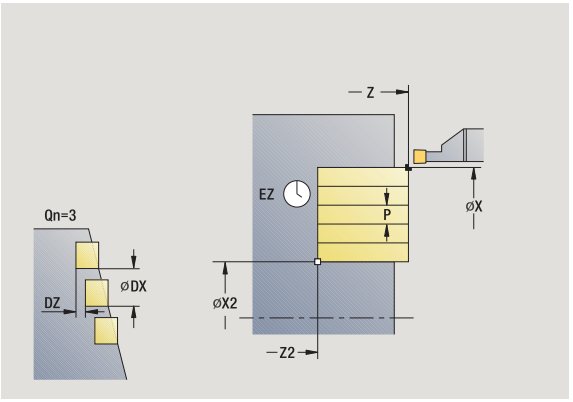


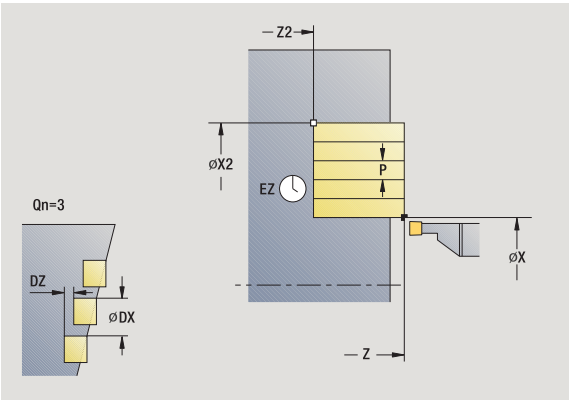
Call the **recessing cycles**



Select **axial recessing**

The cycle machines the number of recesses defined in **number Qn.** The parameters for **starting point** and **end point of contour** define the first recess (position, recess depth and recess width).





Cycle parameters

X, Z	Starting point
X2, Z2	Contour end point
P	Recessing width: infeeds <= P (no input: P = 0.8 * cutting width of the tool)
EZ	Dwell time for chip breaking (default: length of time for two revolutions)
Qn	Number of recess cycles (default: 1)
DX, DZ	Distance to subsequent recess with respect to the preceding recess
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **contour recessing**

Cycle run

1. Calculate the recess positions and the proportioning of cuts
2. Approach the workpiece for next recess from starting point or from last recess on paraxial path
3. Move at the programmed feed rate to the **contour end point Z2**
4. Remain at this position for **dwell time EZ**
5. Retract and approach for next pass
6. Repeat 3 to 5 until the complete recess has been machined
7. Repeat 2 to 6 until all recesses have been machined
8. Return to starting point on paraxial path
9. Move to the tool change point according to the G14 setting

Recessing, radial—expanded



Call the **recessing cycles**

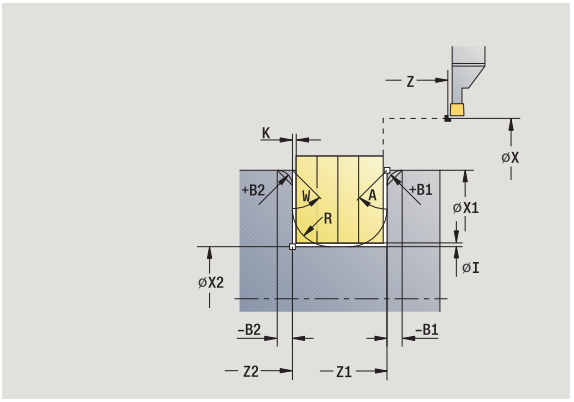


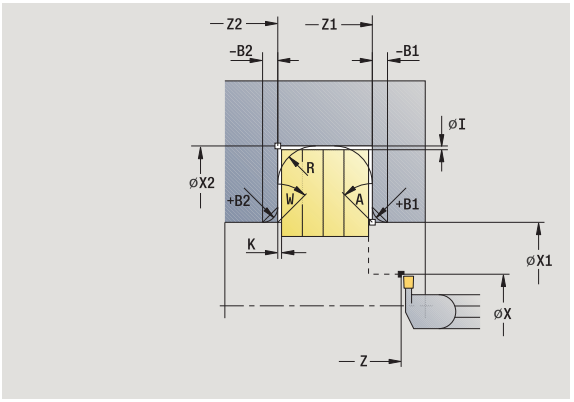
Select **recessing, radial**

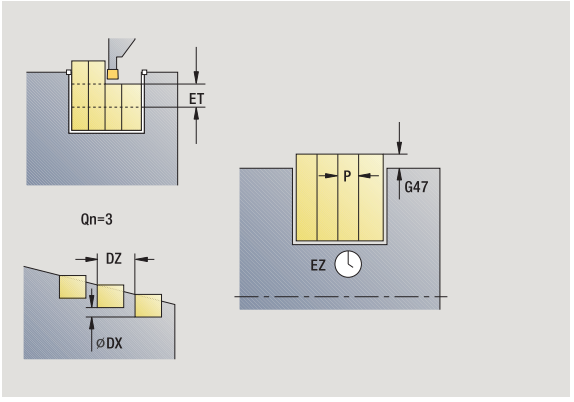
Expanded

Press the **Expanded** soft key

The cycle machines the number of recesses defined in **number Qn.** The parameters for the **contour starting point** and **end point of contour** define the first recess (position, recess depth and recess width).







Cycle parameters

X, Z	Starting point
X1, Z1	Contour starting point
X2, Z2	Contour end point
B1, B2	Chamfer/rounding arc (B1 contour start; B2 contour end) ■ B>0: Rounding radius ■ B<0: Chamfer width
A	Starting angle (range: 0° <= A < 90°)
W	End angle (range: 0° <= W < 90°)
R	Rounding arc
I, K	Oversize X, Z
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
P	Recessing width: infeeds <= P (no input: P = 0.8 * cutting width of the tool)
ET	Recessing depth by which one cut is fed
EZ	Dwell time for chip breaking (default: length of time for two revolutions)
Qn	Number of recess cycles (default: 1)
DX, DZ	Distance to subsequent recess with respect to the preceding recess
G47	Safety clearance (see page 128)

MT M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

Type of machining for technology database access: **contour recessing**

By setting the following **optional parameters,** you can define additional contour elements:

- A:Oblique cut at contour start
- W:Oblique cut at contour end
- R:Rounding arc (in both corners of the contour valley)
- B1:Chamfer/Rounding at contour start
- B2:Chamfer/Rounding at contour end

Cycle run

1 Calculate the recess positions and the proportioning of cuts

2 Approach the workpiece for next recess from starting point or from last recess on paraxial path

3 Move at the programmed feed rate to the **`contour end point X2,`** or if defined, to one of the optional contour elements

4 Remain at this position for a dwell time of two revolutions

5 Retract and approach for next pass

6 Repeat 3 to 5 until the complete recess has been machined

7 Repeat 2 to 6 until all recesses have been machined

8 Return to starting point on paraxial path

9 Move to the tool change point according to the G14 setting

Recessing, axial—expanded



Call the **recessing cycles**

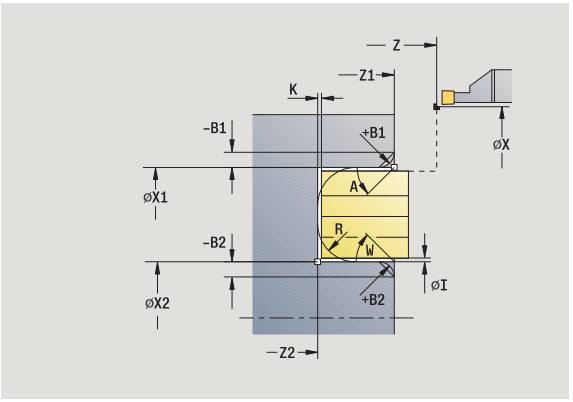


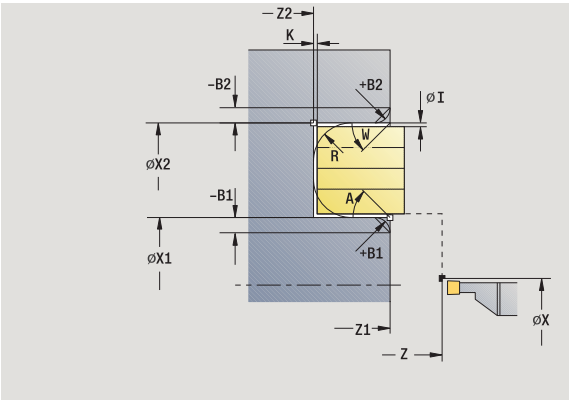
Select **axial recessing**

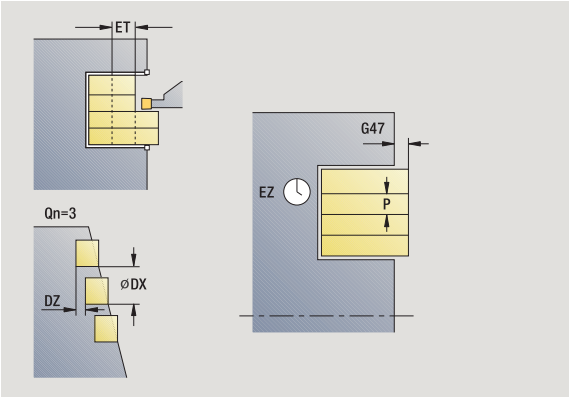
Expanded

Press the **Expanded** soft key

The cycle machines the number of recesses defined in **number Qn.** The parameters for the **contour starting point** and **end point of contour** define the first recess (position, recess depth and recess width).







Cycle parameters

X, Z	Starting point
X1, Z1	Contour starting point
X2, Z2	Contour end point
B1, B2	Chamfer/rounding arc (B1 contour start; B2 contour end) ■ B>0: Rounding radius ■ B<0: Chamfer width
A	Starting angle (range: 0° <= A < 90°)
W	End angle (range: 0° <= W < 90°)
R	Rounding arc
I, K	Oversize X, Z
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
P	Recessing width: infeeds <= P (no input: P = 0.8 * cutting width of the tool)
ET	Recessing depth by which one cut is fed
EZ	Dwell time for chip breaking (default: length of time for two revolutions)
Qn	Number of recess cycles (default: 1)
DX, DZ	Distance to subsequent recess with respect to the preceding recess
G47	Safety clearance (see page 128)
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **contour recessing**

By setting the following **optional parameters,** you can define additional contour elements:

- A:Oblique cut at contour start
- W:Oblique cut at contour end
- R:Rounding arc (in both corners of the contour valley)
- B1:Chamfer/Rounding at contour start
- B2:Chamfer/Rounding at contour end

Cycle run

1. Calculate the recess positions and the proportioning of cuts
2. Approach the workpiece for next recess from starting point or from last recess on paraxial path
3. Move at the programmed feed rate to the **`contour end point Z2,`** or if defined, to one of the optional contour elements
4. Remain at this position for a dwell time of two revolutions
5. Retract and approach for next pass
6. Repeat 3 to 5 until the complete recess has been machined
7. Repeat 2 to 6 until all recesses have been machined
8. Return to starting point on paraxial path
9. Move to the tool change point according to the G14 setting

Recessing radial, finishing



Call the **recessing cycles**

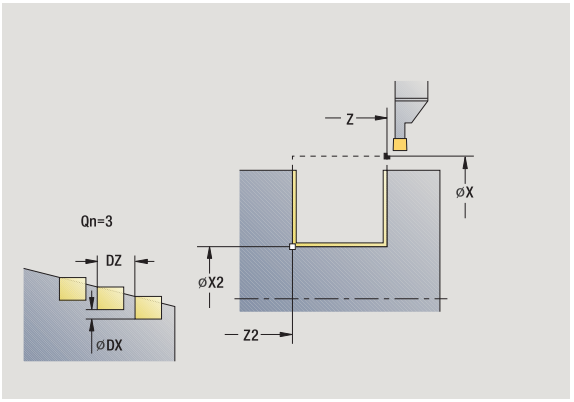


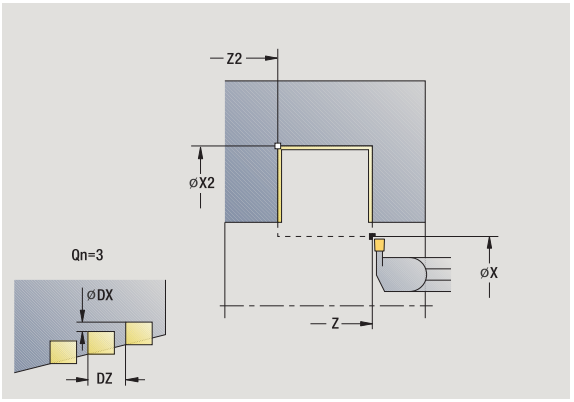
Select **recessing, radial**



Press the **Finishing run** soft key

The cycle finishes the number of recesses defined in **number Qn.** The parameters for **starting point** and **end point of contour** define the first recess (position, recess depth and recess width).





Cycle parameters

X, Z	Starting point
X2, Z2	Contour end point
Qn	Number of recess cycles (default: 1)
DX, DZ	Distance to subsequent recess with respect to the preceding recess
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **contour recessing**

Cycle run

1 Calculate the recess positions

2 Approach the workpiece for next recess from starting point or from last recess on paraxial path

3 Finish first side and the contour valley up to position just before recess end point

4 Approach workpiece for finishing the second side on paraxial path

5 Finish the second side and the remainder of the contour valley

6 Repeat 2 to 5 until all recesses have been machined

7 Return to starting point on paraxial path

8 Move to the tool change point according to the G14 setting

Recessing axial, finishing



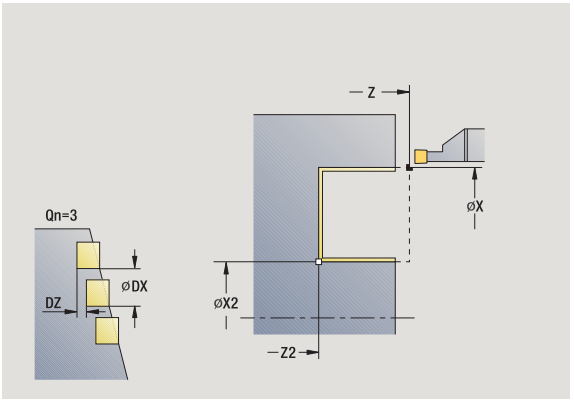
Call the **recessing cycles**

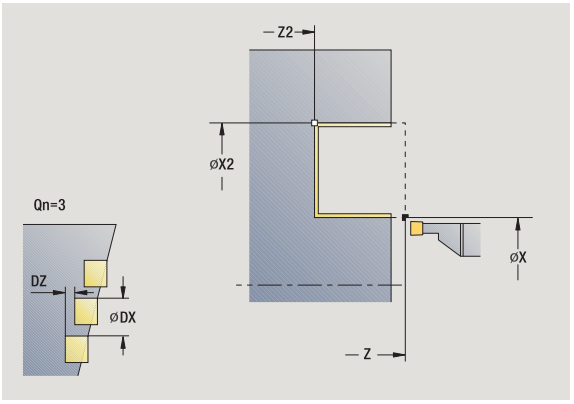


Select **axial recessing**



Press the **Finishing run** soft key





The cycle finishes the number of recesses defined in **number Qn.** The parameters for **starting point** and **end point of contour** define the first recess (position, recess depth and recess width).

Cycle parameters

X, Z	Starting point
X2, Z2	Contour end point
Qn	Number of recess cycles (default: 1)
DX, DZ	Distance to subsequent recess with respect to the preceding recess
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **contour recessing**

Cycle run

1 Calculate the recess positions

2 Approach the workpiece for next recess from starting point or from last recess on paraxial path

3 Finish first side and the contour valley up to position just before recess end point

4 Approach workpiece for finishing the second side on paraxial path

5 Finish the second side and the remainder of the contour valley

6 Repeat 2 to 5 until all recesses have been machined

7 Return to starting point on paraxial path

8 Move to the tool change point according to the G14 setting

Recessing radial, finishing—expanded



Call the **recessing cycles**



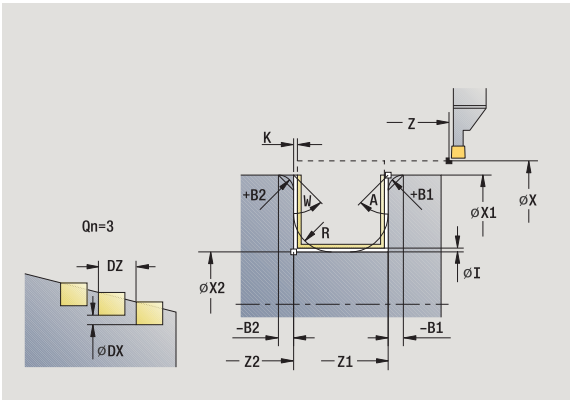
Select **recessing, radial**

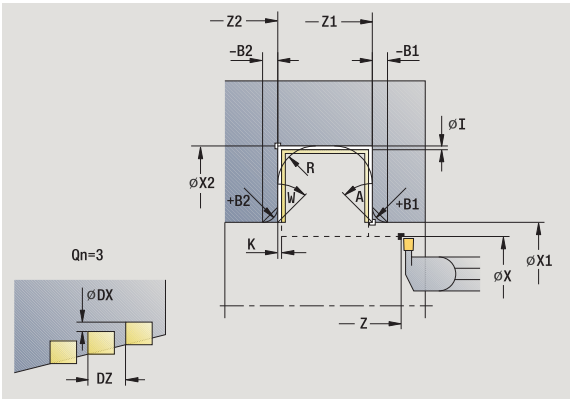
Expanded

Press the **Expanded** soft key

Finishing run

Press the **Finishing run** soft key





The cycle machines the number of recesses defined in **number Qn.** The parameters for the **contour starting point** and **end point of contour** define the first recess (position, recess depth and recess width).

Cycle parameters

X, Z	Starting point
X1, Z1	Contour starting point
X2, Z2	Contour end point
B1, B2	Chamfer/rounding arc (B1 contour start; B2 contour end) ■ B>0: Rounding radius ■ B<0: Chamfer width
A	Starting angle (range: 0° <= A < 90°)
W	End angle (range: 0° <= W < 90°)
R	Rounding arc
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
Qn	Number of recess cycles (default: 1)
DX, DZ	Distance to subsequent recess with respect to the preceding recess
G47	Safety clearance (see page 128)
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **contour recessing**

By setting the following **optional parameters,** you can define additional contour elements:

- A:Oblique cut at contour start
- W:Oblique cut at contour end
- R:Rounding arc (in both corners of the contour valley)
- B1:Chamfer/Rounding at contour start
- B2:Chamfer/Rounding at contour end

Cycle run

1. Calculate the recess positions
2. Approach the workpiece for next recess from starting point or from last recess on paraxial path
3. Finish first side (taking the optional contour elements into account) and the contour valley up to position just before recess end point
4. Approach workpiece for finishing the second side on paraxial path
5. Finish second side, taking optional contour elements into account; then finish remainder of contour valley
6. Repeat 2 to 5 until all recesses have been finished
7. Return to starting point on paraxial path
8. Move to the tool change point according to the G14 setting

Recessing axial, finishing—expanded



Call the **recessing cycles**



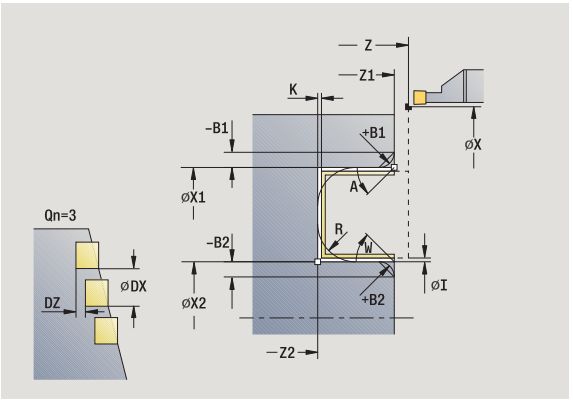
Select **axial recessing**

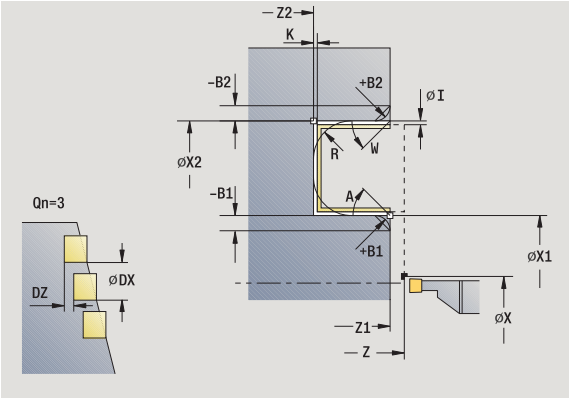
Expanded

Press the **Expanded** soft key

Finishing run

Press the **Finishing run** soft key





The cycle machines the number of recesses defined in **number Qn.** The parameters for the **contour starting point** and **end point of contour** define the first recess (position, recess depth and recess width).

Cycle parameters

X, Z	Starting point
X1, Z1	Contour starting point
X2, Z2	Contour end point
B1, B2	Chamfer/rounding arc (B1 contour start; B2 contour end)
	■ B>0: Rounding radius
	■ B<0: Chamfer width
A	Starting angle (range: 0° <= A < 90°)
W	End angle (range: 0° <= W < 90°)
R	Rounding arc
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
Qn	Number of recess cycles (default: 1)
DX, DZ	Distance to subsequent recess with respect to the preceding recess
G47	Safety clearance (see page 128)
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **contour recessing**

By setting the following **optional parameters,** you can define additional contour elements:

- A:Oblique cut at contour start
- W:Oblique cut at contour end
- R:Rounding arc (in both corners of the contour valley)
- B1:Chamfer/Rounding at contour start
- B2:Chamfer/Rounding at contour end

Cycle run

1. Calculate the recess positions
2. Approach the workpiece for next recess from starting point or from last recess on paraxial path
3. Finish first side (taking the optional contour elements into account) and the contour valley up to position just before recess end point
4. Approach workpiece for finishing the second side on paraxial path
5. Finish second side, taking optional contour elements into account; then finish remainder of contour valley
6. Repeat 2 to 5 until all recesses have been finished
7. Return to starting point on paraxial path
8. Move to the tool change point according to the G14 setting

ICP recessing cycles, radial



Call the **recessing cycles**

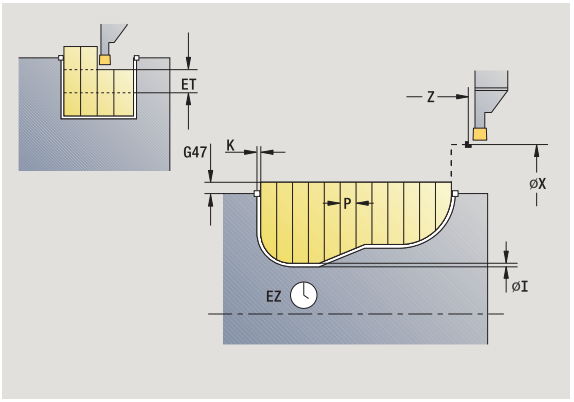


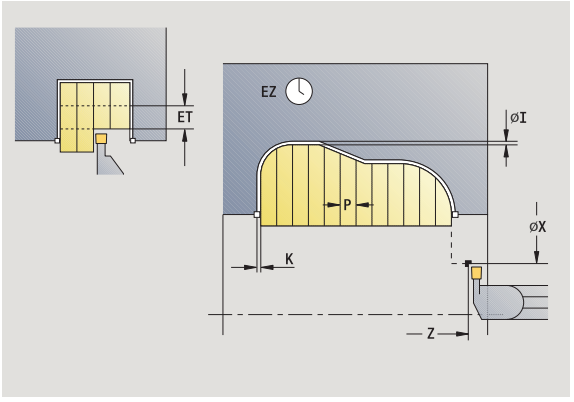
Select **recessing, radial ICP**.

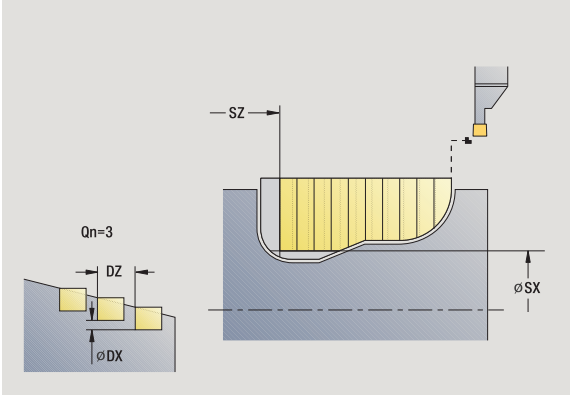
The cycle machines the number of recesses defined in **number Qn** with the ICP recessing contour. The **starting point** defines the position of the first recess.

Cycle parameters

X, Z	Starting point
FK	ICP finished part: Name of the contour to be machined
P	Recessing width: infeeds <= P (no input: P = 0.8 * cutting width of the tool)
ET	Recessing depth by which one cut is fed.
I, K	Oversize X, Z
EZ	Dwell time for chip breaking (default: length of time for two revolutions)
Qn	Number of recess cycles (default: 1)
DX, DZ	Distance to subsequent recess with respect to the preceding recess
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
SX, SZ	Cutting limits (see page 128)
G47	Safety clearance (see page 128)
MT	M after T: M function that is executed after the tool call T







MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **contour recessing**

Cycle run

1. Calculate the recess positions and the proportioning of cuts
2. Approach the workpiece for next recess from starting point or from last recess on paraxial path
3. Machine along the defined contour
4. Return and approach for next pass
5. Repeat 3 to 4 until the complete recess has been machined
6. Repeat 2 to 5 until all recesses have been machined
7. Return to starting point on paraxial path
8. Move to the tool change point according to the G14 setting

ICP recessing cycles, axial



Call the **recessing cycles**

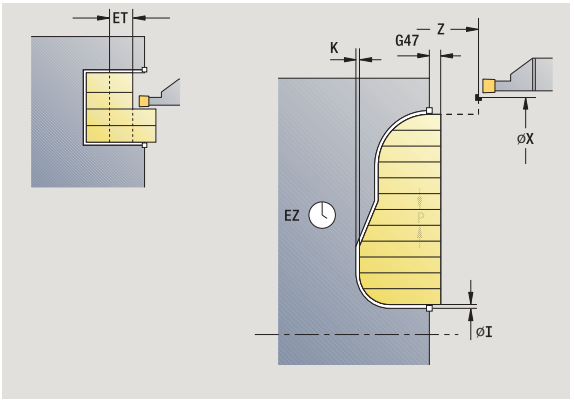


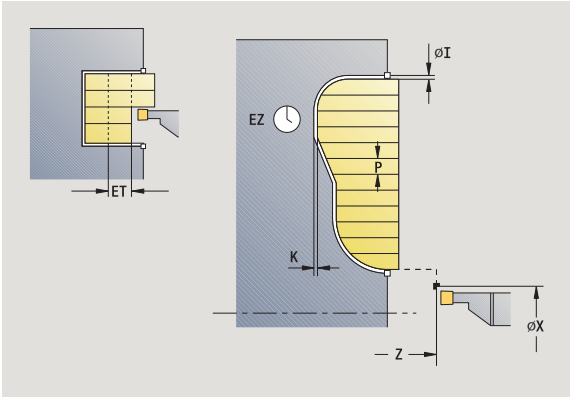
Select **recessing, axial ICP**

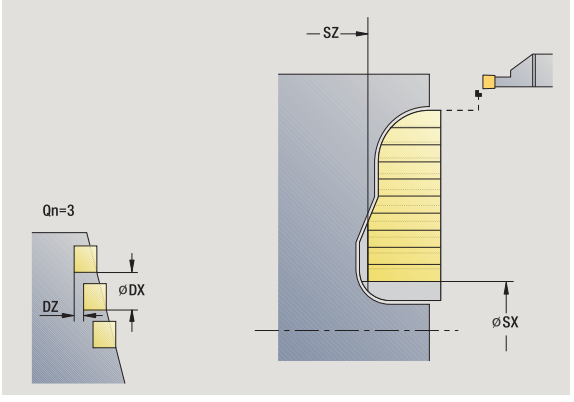
The cycle machines the number of recesses defined in **number Qn** with the ICP recessing contour. The **starting point** defines the position of the first recess.

Cycle parameters

X, Z	Starting point
FK	ICP finished part: Name of the contour to be machined
P	Recessing width: infeeds <= P (no input: P = 0.8 * cutting width of the tool)
ET	Recessing depth by which one cut is fed.
I, K	Oversize X, Z
EZ	Dwell time for chip breaking (default: length of time for two revolutions)
Qn	Number of recess cycles (default: 1)
DX, DZ	Distance to subsequent recess with respect to the preceding recess
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
SX, SZ	Cutting limits (see page 128)
G47	Safety clearance (see page 128)
MT	M after T: M function that is executed after the tool call T







MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **contour recessing**

Cycle run

1. Calculate the recess positions and the proportioning of cuts
2. Approach the workpiece for next recess from starting point or from last recess on paraxial path
3. Machine along the defined contour
4. Return and approach for next pass
5. Repeat 3 to 4 until the complete recess has been machined
6. Repeat 2 to 5 until all recesses have been machined
7. Return to starting point on paraxial path
8. Move to the tool change point according to the G14 setting

ICP recessing, radial finishing



Call the **recessing cycles**



Select **recessing, radial ICP**.

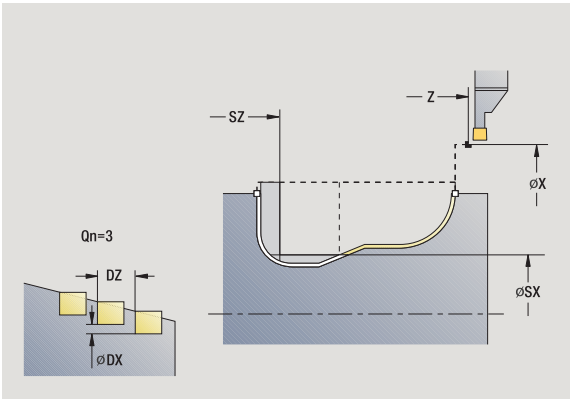
Finishing run

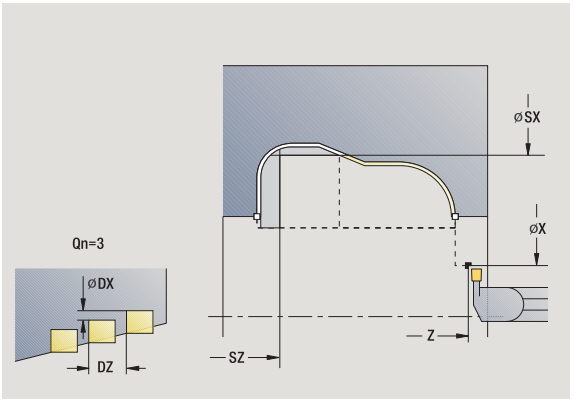
Press the **Finishing run** soft key

The cycle finishes the number of recesses defined in **number Qn** with the ICP recessing contour. The **starting point** defines the position of the first recess.



At the end of the cycle, the tool returns to the starting point.





Cycle parameters

X, Z	Starting point
FK	ICP finished part: Name of the contour to be machined
Qn	Number of recess cycles (default: 1)
DX, DZ	Distance to subsequent recess with respect to the preceding recess
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
SX, SZ	Cutting limits (see page 128)
G47	Safety clearance (see page 128)
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **contour recessing**

Cycle run

1 Calculate the recess positions

2 Approach the workpiece for next recess from starting point or from last recess on paraxial path

3 Finish the recess

4 Repeat 2 to 3 until all recesses have been machined

5 Return to starting point on paraxial path

6 Move to the tool change point according to the G14 setting

ICP recessing, axial finishing



Call the **recessing cycles**



Select **recessing, axial ICP**

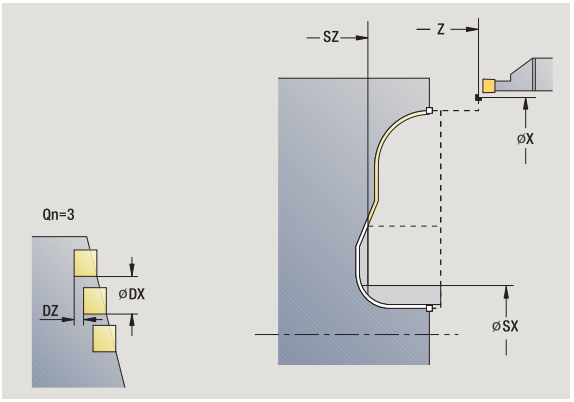


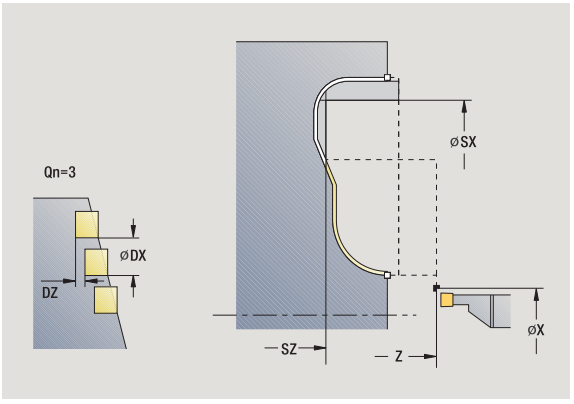
Press the **Finishing run** soft key

The cycle finishes the number of recesses defined in **number Qn** with the ICP recessing contour. The **starting point** defines the position of the first recess.



At the end of the cycle, the tool returns to the starting point.





Cycle parameters

X, Z	Starting point
FK	ICP finished part: Name of the contour to be machined
Qn	Number of recess cycles (default: 1)
DX, DZ	Distance to subsequent recess with respect to the preceding recess
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
SX, SZ	Cutting limits (see page 128)
G47	Safety clearance (see page 128)
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **contour recessing**

Cycle run

1. Calculate the recess positions
2. Approach the workpiece for next recess from starting point or from last recess on paraxial path
3. Finish the recess
4. Repeat 2 to 3 until all recesses have been machined
5. Return to starting point on paraxial path
6. Move to the tool change point according to the G14 setting

Recess turning

The recess turning cycles machine by alternate recessing and roughing movements. The machining process requires a minimum of retraction and infeed movements.

To influence recess-turning operations, use the following parameters:

- **`Recessing feed rate O:`** Feed rate for recessing movement
- **`Turning operation, unidirectional/bidirectional U:`** You can perform a unidirectional or bidirectional turning operation. With **radial** recess-turning cycles, unidirectional turning operations are always performed in the direction of the spindle. With **axial** ICP recess-turning cycles, the machining direction corresponds to the direction of contour definition.
- **`Offset width B:`** After the second infeed movement, during the transition from turning to recessing, the path to be machined is reduced by the offset width. Each time the system switches from turning to recessing on this side, the path is reduced by the offset width—in addition to the previous offset. The total offset is limited to 80 % of the effective cutting width (effective cutting width = cutting width –2*cutting radius). If required, the CNC PILOT reduces the programmed offset width. After precutting, the remaining material is removed with a single cut.
- **`Depth compensation RB:`** Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. The resulting infeed error can be compensated with depth compensation during "finishing, expanded." The depth compensation factor is usually determined empirically.



These cycles require the use of **recess-turning tools.**

Recess turning, radial



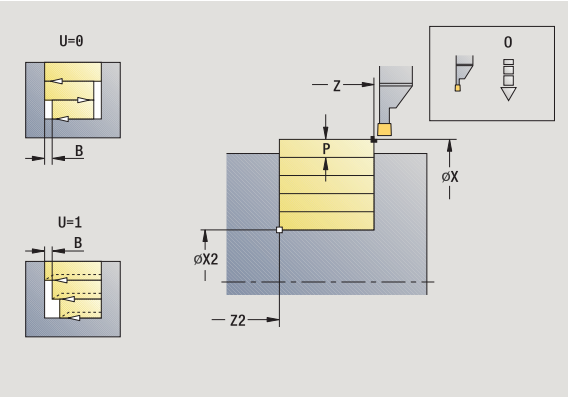
Call the **recessing cycles**



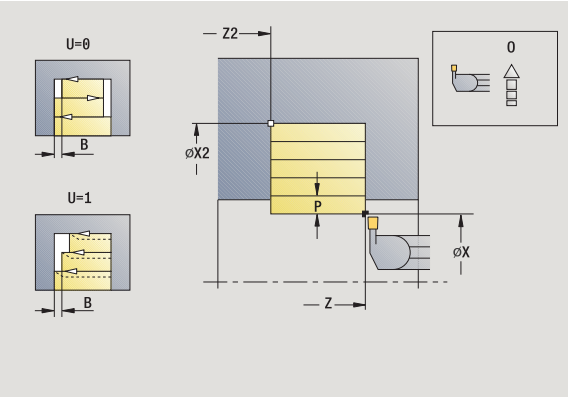
Select **recess turning**



Select **recess turning, radial**



The cycle clears the rectangle described by the **starting point** and **contour end point.**



Cycle parameters

X, Z	Starting point
X2, Z2	Contour end point
P	Infeed depth: Maximum infeed depth
O	Recessing feed rate (default: active feed rate)
B	Offset width (default: 0)
U	Unidirectional turning (default: 0) ■ 0: Bidirectional ■ 1: Unidirectional
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Recess turning**

Cycle run

1. Calculate the proportioning of cuts
2. Approach the workpiece from starting point for first pass
3. Execute the first cut (recessing)
4. Machine perpendicularly to recessing direction (turning)
5. Repeat 3 to 4 until **contour end point X2, Z2** is reached
6. Return to starting point on paraxial path
7. Move to the tool change point according to the G14 setting

Recess turning, axial



Call the **recessing cycles**

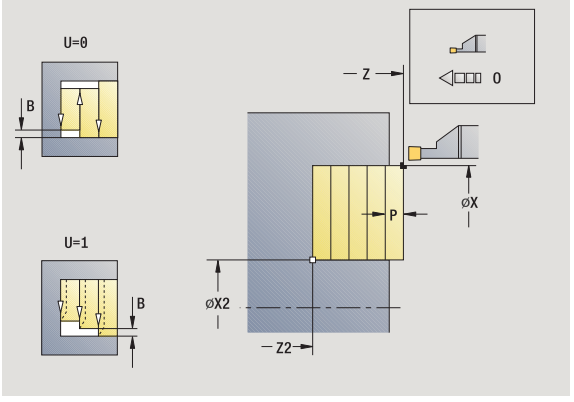


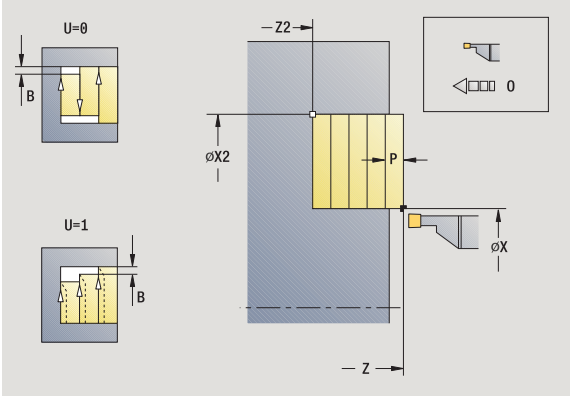
Select **recess turning**



Select **recess turning, axial**

The cycle clears the rectangle described by the **starting point** and **contour end point.**





Cycle parameters

X, Z Starting point

X2, Z2 Contour end point

P Infeed depth: Maximum infeed depth

O Recessing feed rate (default: active feed rate)

B Offset width (default: 0)

U Unidirectional turning (default: 0)

- 0: Bidirectional
- 1: Unidirectional

G47 Safety clearance (see page 128)

G14 Tool change point (see page 128)

T Turret pocket number

ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent) ■ Main drive ■ Opposing spindle for rear-face machining

Type of machining for technology database access: **Recess turning**

Cycle run

1 Calculate the proportioning of cuts
2 Approach the workpiece from starting point for first pass
3 Execute the first cut (recessing)
4 Machine perpendicularly to recessing direction (turning)
5 Repeat 3 to 4 until **`contour end point X2, Z2`** is reached
6 Return to starting point on paraxial path
7 Move to the tool change point according to the G14 setting

Recess turning, radial—expanded



Call the **recessing cycles**



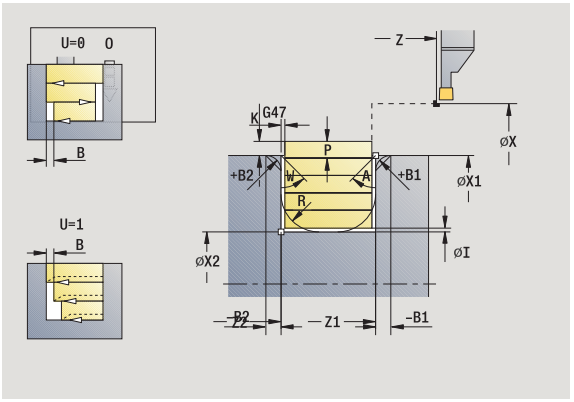
Select **recess turning**

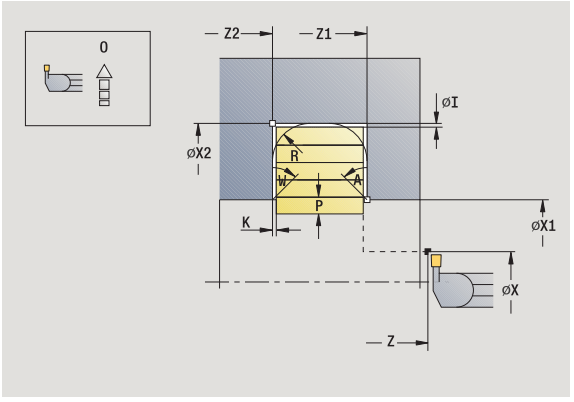


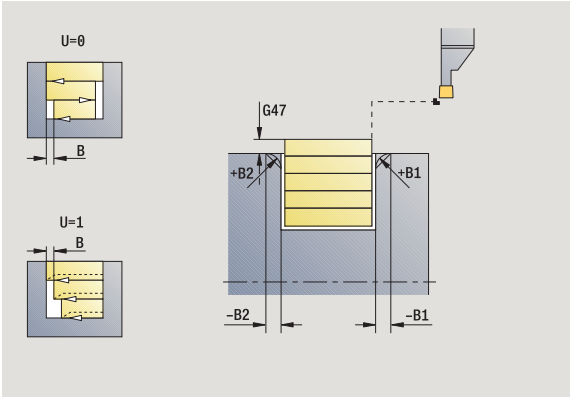
Select **recess turning, radial**

Expanded

Press the **Expanded** soft key







Taking the oversizes into account, the cycle clears the area described by the **starting point X** / **contour starting point Z1** and **contour end point** (see also "Recess turning" on page 224).

Cycle parameters

X, Z	Starting point
X1, Z1	Contour starting point
X2, Z2	Contour end point
P	Infeed depth: Maximum infeed depth
O	Recessing feed rate (default: active feed rate)
I, K	Oversize X, Z
A	Starting angle (range: 0° <= A < 90°)
W	End angle (range: 0° <= W < 90°)
R	Rounding arc
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
B1, B2	Chamfer/rounding arc (B1 contour start; B2 contour end) ■ B>0: Rounding radius ■ B<0: Chamfer width
B	Offset width (default: 0)
U	Unidirectional turning (default: 0) ■ 0: Bidirectional ■ 1: Unidirectional
G47	Safety clearance (see page 128)
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Recess turning**

By setting the following **optional parameters,** you can define additional contour elements:

- A:Oblique cut at contour start
- W:Oblique cut at contour end
- R:Rounding arc (in both corners of the contour valley)
- B1:Chamfer/Rounding at contour start
- B2:Chamfer/Rounding at contour end

Cycle run

1 Calculate the proportioning of cuts
2 Approach the workpiece from starting point for first pass
3 Execute the first cut (recessing)
4 Machine perpendicularly to recessing direction (turning)
5 Repeat 3 to 4 until **`contour end point X2, Z2`** is reached
6 Machine chamfer/rounding at contour start / contour end if defined
7 Return to starting point on paraxial path
8 Move to the tool change point according to the G14 setting

Recessing turning, axial—expanded



Call the **recessing cycles**



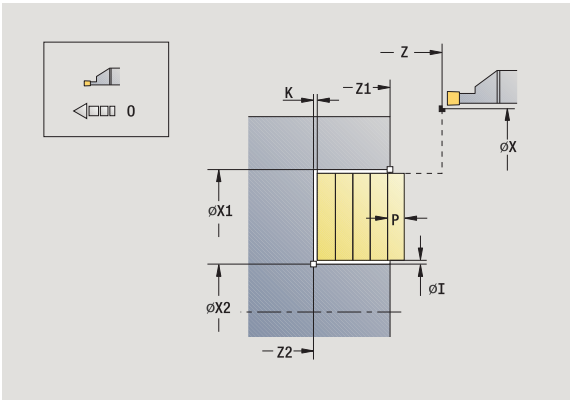
Select **recess turning**

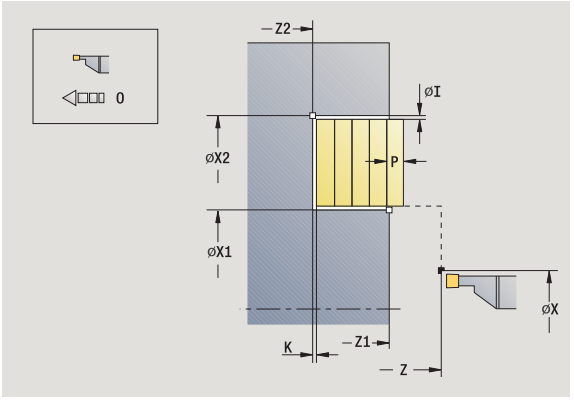


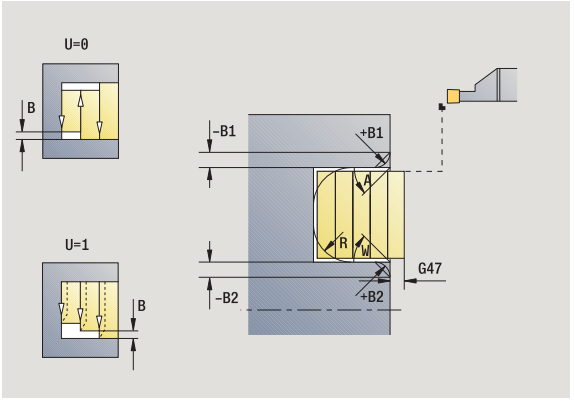
Select **recess turning, axial**

Expanded

Press the **Expanded** soft key







Taking the oversizes into account, the cycle clears the area described by the **contour starting point X1 / starting point Z** and **contour end point** (see also "Recess turning" on page 224).

Cycle parameters

X, Z	Starting point
X1, Z1	Contour starting point
X2, Z2	Contour end point
P	Infeed depth: Maximum infeed depth
O	Recessing feed rate (default: active feed rate)
I, K	Oversize X, Z
A	Starting angle (range: 0° <= A < 90°)
W	End angle [range (0° <= W < 90°)]
R	Rounding arc
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
B1, B2	Chamfer/rounding arc (B1 contour start; B2 contour end)
	■ B>0: Rounding radius
	■ B<0: Chamfer width
B	Offset width (default: 0)
U	Unidirectional turning (default: 0)
	■ 0: Bidirectional
	■ 1: Unidirectional
G47	Safety clearance (see page 128)
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Recess turning**

By setting the following **optional parameters,** you can define additional contour elements:

- A:Oblique cut at contour start
- W:Oblique cut at contour end
- R:Rounding arc (in both corners of the contour valley)
- B1:Chamfer/Rounding at contour start
- B2:Chamfer/Rounding at contour end

Cycle run

1. Calculate the proportioning of cuts
2. Approach the workpiece from starting point for first pass
3. Execute the first cut (recessing)
4. Machine perpendicularly to recessing direction (turning)
5. Repeat 3 to 4 until **`contour end point X2, Z2`** is reached
6. Machine chamfer/rounding at contour start / contour end if defined
7. Return to starting point on paraxial path
8. Move to the **`tool change position`** according to the **`G14`** setting

Recess turning, radial finishing



Call the **recessing cycles**



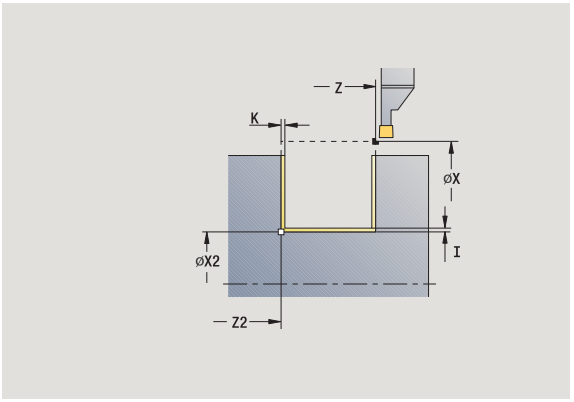
Select **recess turning**

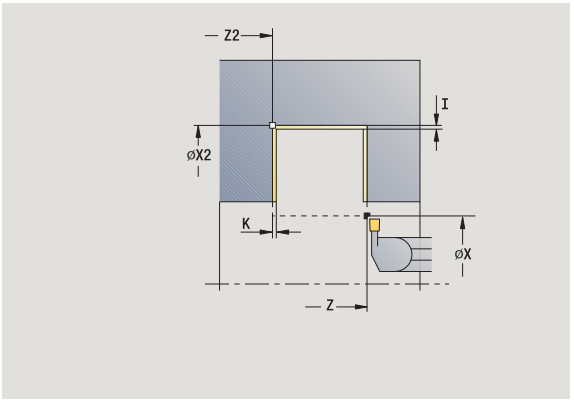


Select **recess turning, radial**

Finishing run

Press the **Finishing run** soft key





The cycle finishes the contour area defined by the **starting point** and **contour end point** (see also "Recess turning" on page 224).



With **oversizes I, K** for the workpiece blank, you define the material to be machined during the finishing cycle. For this purpose, enter the oversizes for recess turning, finishing.

Cycle parameters

X, Z	Starting point
X2, Z2	Contour end point
I, K	Workpiece blank oversize X, Z
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
G47	Safety clearance (see page 128)
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Recess turning**

Cycle run

1. Approach workpiece from starting point
2. Finish the first side, then finish the contour valley up to position just before **contour end point X2, Z2**
3. Move paraxially to the **starting point X/end point Z2**
4. Finish second side, then finish remainder of contour valley
5. Return to starting point on paraxial path
6. Move to the tool change point according to the G14 setting

Recess turning, axial finishing



Call the **recessing cycles**



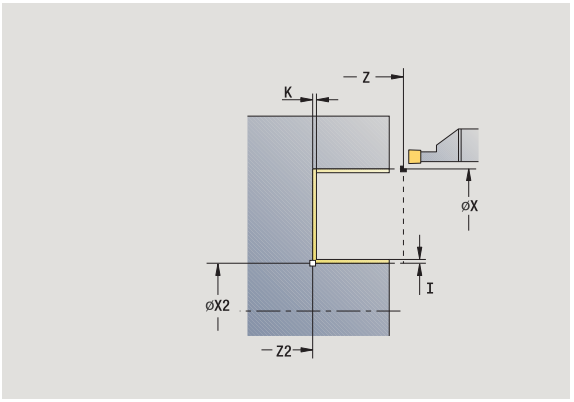
Select **recess turning**

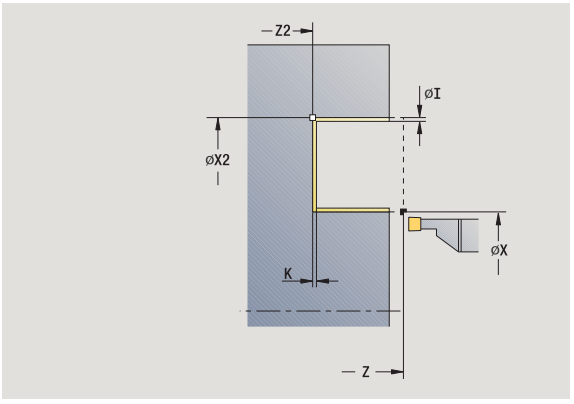


Select **recess turning, axial**

Finishing run

Press the **Finishing run** soft key





The cycle finishes the contour area defined by the **starting point** and **contour end point** (see also "Recess turning" on page 224).



With **oversizes I, K** for the workpiece blank, you define the material to be machined during the finishing cycle. For this purpose, enter the oversizes for recess turning, finishing.

Cycle parameters

X, Z	Starting point
X2, Z2	Contour end point
I, K	Workpiece blank oversize X, Z
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
G47	Safety clearance (see page 128)
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

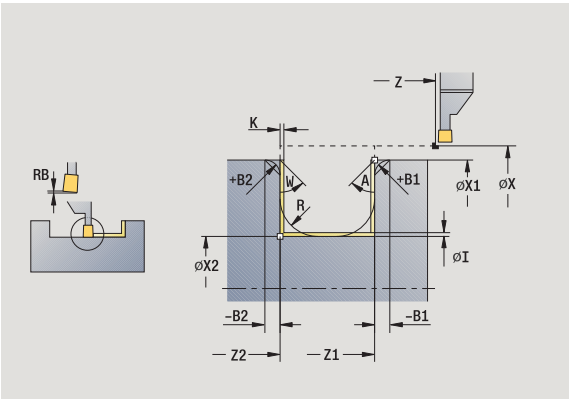
- Main drive
- Opposing spindle for rear-face machining

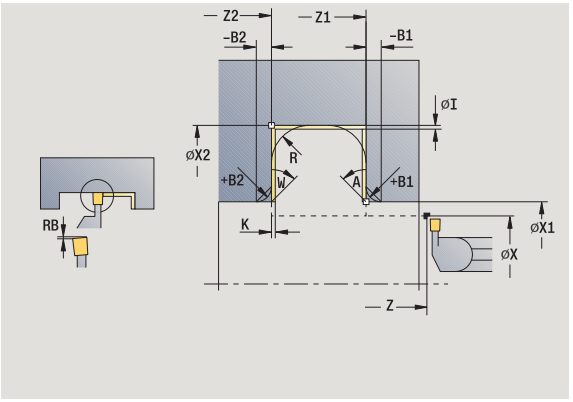
Type of machining for technology database access: **Recess turning**

Cycle run

1. Approach workpiece from starting point
2. Finish the first side, then finish the contour valley up to position just before **contour end point X2, Z2**
3. Move paraxially to the **starting point Z/end point X2**
4. Finish second side, then finish remainder of contour valley
5. Return to starting point on paraxial path
6. Move to the tool change point according to the G14 setting

Recess turning, radial finishing—expanded





Call the **recessing cycles**

Select **recess turning**

Select **recess turning, radial**

Expanded

Press the **Expanded** soft key

Finishing run

Press the **Finishing run** soft key

The cycle finishes the contour area defined by the **contour starting point** and **contour end point** (see also "Recess turning" on page 224).

With **oversizes I, K** for the workpiece blank, you define the material to be machined during the finishing cycle. For this purpose, enter the oversizes for recess turning, finishing.

Cycle parameters

X, Z	Starting point
X1, Z1	Contour starting point
X2, Z2	Contour end point
RB	Depth compensation
I, K	Oversizes in X and Z are taken into account during finishing in subsequent machining operations
A	Starting angle (range: 0° <= A < 90°)
W	End angle (range: 0° <= W < 90°)
R	Rounding arc
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
B1, B2	Chamfer/rounding arc (B1 contour start; B2 contour end) ■ B>0: Rounding radius ■ B<0: Chamfer width
RI, RK	Workpiece blank oversizes in X and Z: Oversize before the finishing operation for calculating the paths for approach/ departure and the finishing area
G47	Safety clearance (see page 128)

MT M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Recess turning**

By setting the following **optional parameters,** you can define additional contour elements:

- A:Oblique cut at contour start
- W:Oblique cut at contour end
- R:Rounding arc (in both corners of the contour valley)
- B1:Chamfer/Rounding at contour start
- B2:Chamfer/Rounding at contour end

Cycle run

1. Approach workpiece from starting point
2. Finish first side, taking optional contour elements into account; then finish contour valley up to position just before **end point X2/ Z2**
3. Approach workpiece for finishing the second side on paraxial path
4. Finish second side, taking optional contour elements into account; then finish remainder of contour valley
5. Finish chamfer/rounding at contour start / contour end, if defined
6. Move to the tool change point according to the G14 setting

Recess turning, axial finishing—expanded



Call the **recessing cycles**



Select **recess turning**



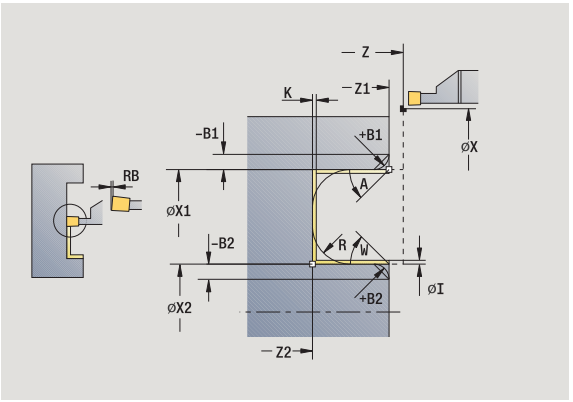
Select **recess turning, axial**

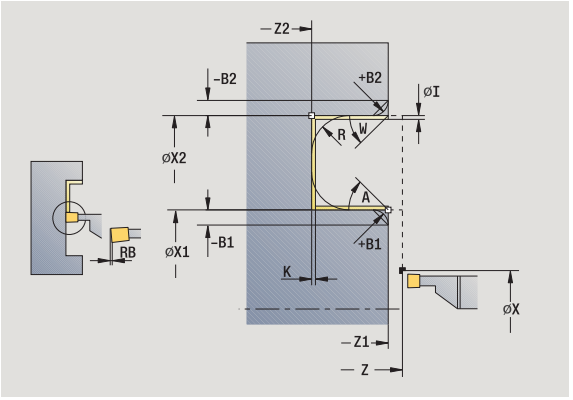
Expanded

Press the **Expanded** soft key

Finishing run

Press the **Finishing run** soft key





The cycle finishes the contour area defined by the **contour starting point** and **contour end point** (see also "Recess turning" on page 224).

With **oversizes I, K**, you define the material to be machined during the finishing cycle. For this purpose, enter the oversizes for recess turning, finishing.

Cycle parameters

X, Z	Starting point
X1, Z1	Contour starting point
X2, Z2	Contour end point
RB	Depth compensation
I, K	Oversizes in X and Z are taken into account during finishing in subsequent machining operations
A	Starting angle (range: 0° <= A < 90°)
W	End angle (range: 0° <= W < 90°)
R	Rounding arc
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
B1, B2	Chamfer/rounding arc (B1 contour start; B2 contour end) ■ B>0: Rounding radius ■ B<0: Chamfer width
RI, RK	Workpiece blank oversizes in X and Z: Oversize before the finishing operation for calculating the paths for approach/ departure and the finishing area
G47	Safety clearance (see page 128)

MT M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Recess turning**

By setting the following **optional parameters,** you can define additional contour elements:

- A:Oblique cut at contour start
- W:Oblique cut at contour end
- R:Rounding arc (in both corners of the contour valley)
- B1:Chamfer/Rounding at contour start
- B2:Chamfer/Rounding at contour end

Cycle run

1. Approach workpiece from starting point
2. Finish first side, taking optional contour elements into account; then finish contour valley up to position just before **end point X2/ Z2**
3. Approach workpiece for finishing the second side on paraxial path
4. Finish second side, taking optional contour elements into account; then finish remainder of contour valley
5. Finish chamfer/rounding at contour start / contour end, if defined
6. Move to the tool change point according to the G14 setting

ICP recess turning, radial



Call the **recessing cycles**



Select **recess turning**



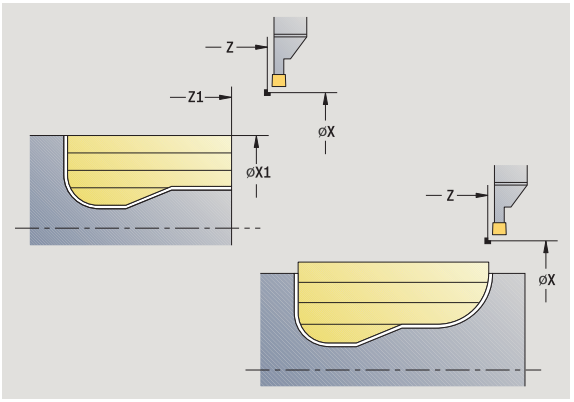
Select **recess turning, radial**

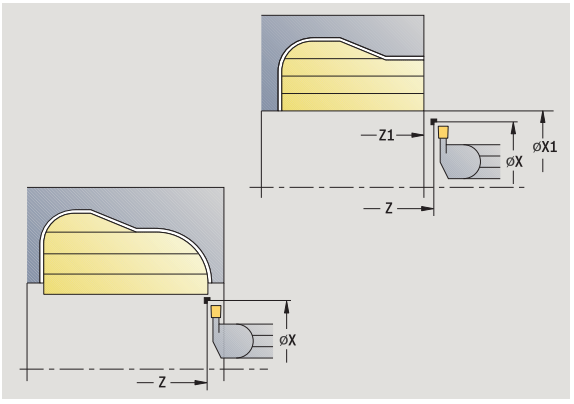
The cycle clears the defined area (see also "Recess turning" on page 224).

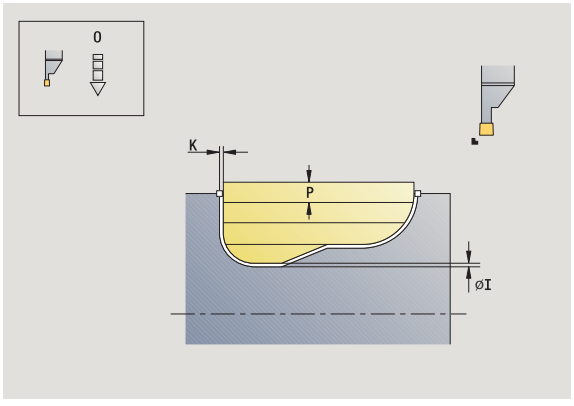


If you are machining

- **descending contours,** define the **starting point**—not the **starting point of the blank**. The cycle clears the area defined by the starting point and the ICP contour, taking the oversizes into account.
- **inclining contours,** define the **starting point** and the **starting point of the blank.** The cycle clears the area defined by the contour starting point and the ICP contour, taking the oversizes into account.







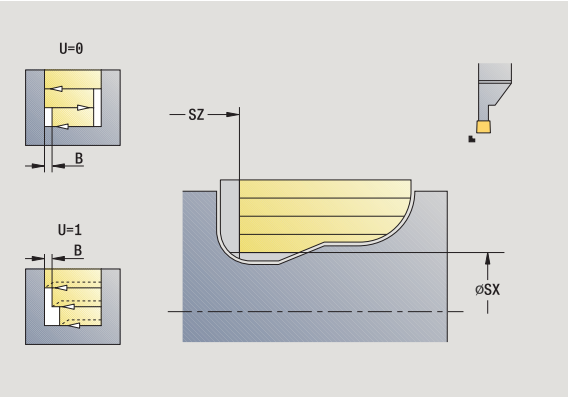
Cycle parameters

X, Z	Starting point
X1, Z1	Starting point of blank
FK	ICP finished part: Name of the contour to be machined
P	Infeed depth: Maximum infeed depth
ET	Recessing depth by which one cut is fed.
O	Recessing feed rate (default: active feed rate)
I, K	Oversizes in X and Z are taken into account during finishing in subsequent machining operations
SX, SZ	Cutting limits (see page 128)
B	Offset width (default: 0)
U	Unidirectional turning (default: 0) ■ 0: Bidirectional ■ 1: Unidirectional (direction: see help graphic)
G14	Tool change point (see page 128)
A	Starting angle defines the machining area at the contour starting point
W	End angle defines the machining area at the contour end point



T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
G47	Safety clearance (see page 128)
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining



Type of machining for technology database access: **Recess turning**

Cycle run

1. Calculate the proportioning of cuts
2. Approach the workpiece from starting point for first pass
3. Execute the first cut (recessing)
4. Machine perpendicularly to recessing direction (turning)
5. Repeat 3 to 4 until the defined area has been machined
6. Return to starting point on paraxial path
7. Move to the tool change point according to the G14 setting

ICP recess turning, axial



Call the **recessing cycles**



Select **recess turning**



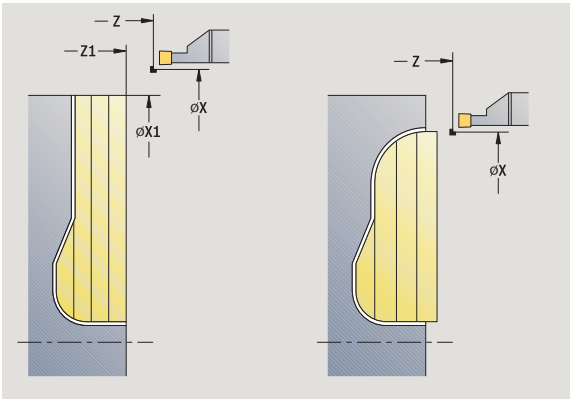
Select **recess turning, axial**

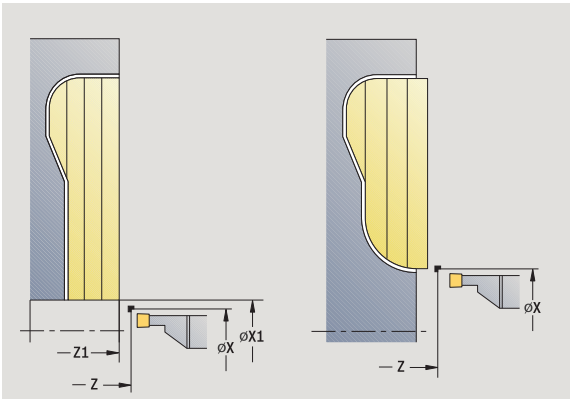
The cycle clears the defined area (see also "Recess turning" on page 224).

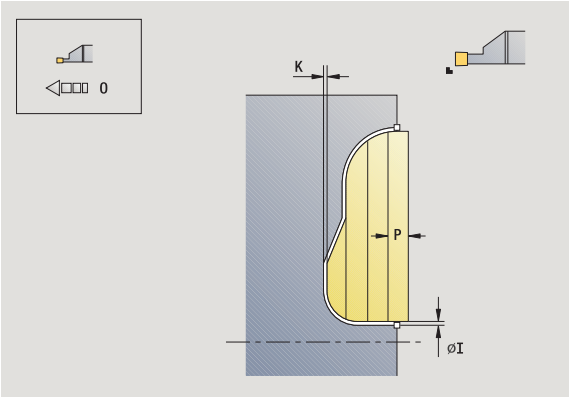


If you are machining

- **descending contours,** define only the **starting point**—not the **contour starting point.** The cycle clears the area defined by the starting point and the ICP contour, taking the oversizes into account.
- **inclining contours,** define the **starting point** and the **contour starting point.** The cycle clears the area defined by the contour starting point and the ICP contour, taking the oversizes into account.







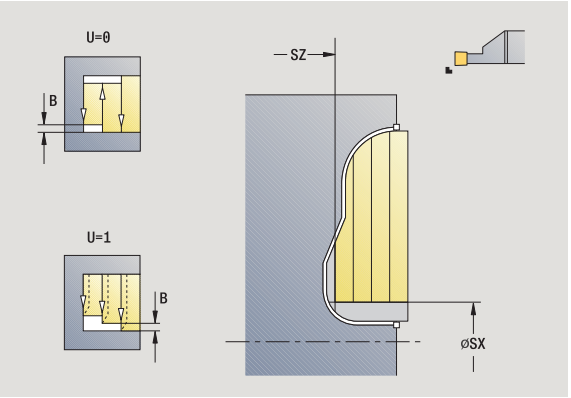
Cycle parameters

X, Z	Starting point
X1, Z1	Starting point of blank
FK	ICP finished part: Name of the contour to be machined
P	Infeed depth: Maximum infeed depth
ET	Recessing depth by which one cut is fed.
O	Recessing feed rate (default: active feed rate)
I, K	Oversize X, Z
SX, SZ	Cutting limits (see page 128)
B	Offset width (default: 0)
U	Unidirectional turning (default: 0) ■ 0: Bidirectional ■ 1: Unidirectional (direction: see help graphic)
G14	Tool change point (see page 128)
A	Starting angle defines the machining area at the contour starting point
W	End angle defines the machining area at the contour end point



T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
G47	Safety clearance (see page 128)
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining



Type of machining for technology database access: **Recess turning**

Cycle run

1. Calculate the proportioning of cuts
2. Approach the workpiece from starting point for first pass
3. Execute the first cut (recessing)
4. Machine perpendicularly to recessing direction (turning)
5. Repeat 3 to 4 until the defined area has been machined
6. Return to starting point on paraxial path
7. Move to the tool change point according to the G14 setting

ICP recess turning, radial finishing



Call the **recessing cycles**



Select **recess turning**



Select **recess turning, radial ICP**

Finishing run

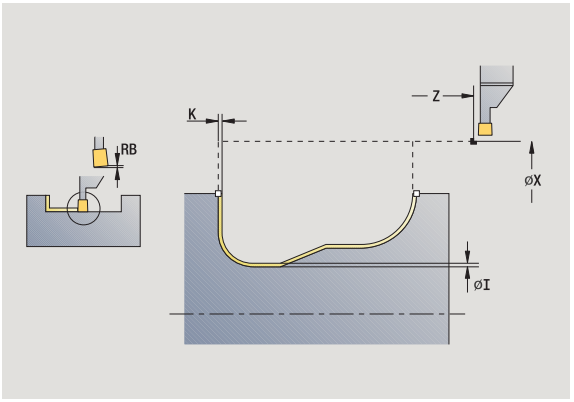
Press the **Finishing run** soft key

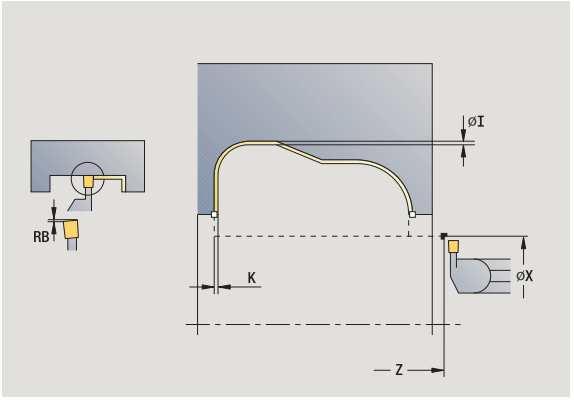
The cycle finishes the contour area defined by the ICP contour (see also "Recess turning" on page 224). At the end of the cycle, the tool returns to the starting point.

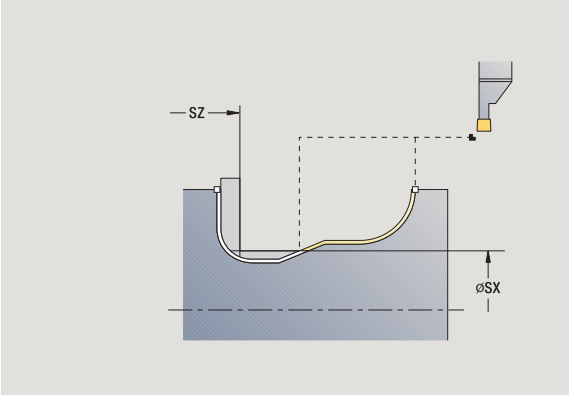
With **oversizes I, K** for the workpiece blank, you define the material to be machined during the finishing cycle. For this purpose, enter the oversizes for recess turning, finishing.

Cycle parameters

X, Z	Starting point
FK	ICP finished part: Name of the contour to be machined
RB	Depth compensation
I, K	Oversize X, Z
SX, SZ	Cutting limits (see page 128)
G14	Tool change point (see page 128)
A	Starting angle defines the machining area at the contour starting point
W	End angle defines the machining area at the contour end point
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
G47	Safety clearance (see page 128)
MT	M after T: M function that is executed after the tool call T







MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Recess turning**

Cycle run

1. Approach workpiece from starting point on paraxial path
2. Finish first side and contour area up to position just before **end point X2, Z2**
3. Approach workpiece for finishing the second side on paraxial path
4. Finish second side, then finish remainder of contour valley
5. Return to starting point on paraxial path
6. Move to the tool change point according to the G14 setting

ICP recess turning, axial finishing



Call the **recessing cycles**



Select **recess turning**



Select **recess turning, axial ICP**

Finishing run

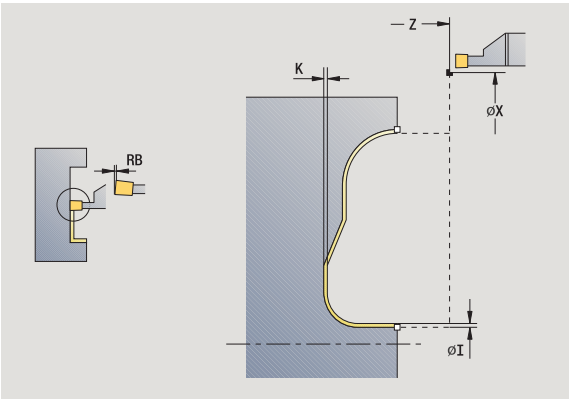
Press the **Finishing run** soft key

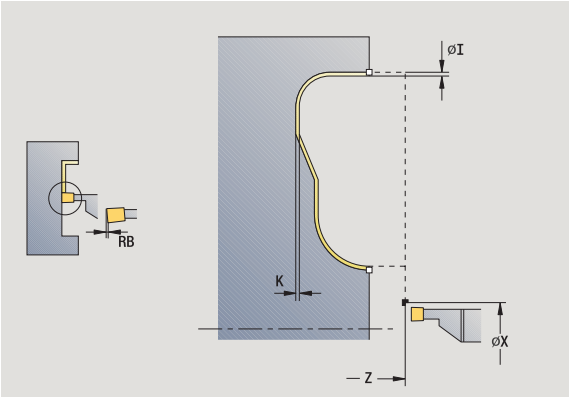
The cycle finishes the contour area defined by the ICP contour (see also "Recess turning" on page 224). At the end of the cycle, the tool returns to the starting point.

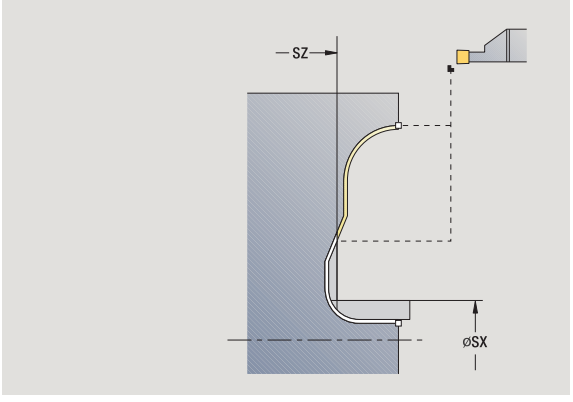
With **oversizes I, K** for the workpiece blank, you define the material to be machined during the finishing cycle. For this purpose, enter the oversizes for recess turning, finishing.

Cycle parameters

X, Z	Starting point
FK	ICP finished part: Name of the contour to be machined
RB	Depth compensation
I, K	Oversize X, Z
SX, SZ	Cutting limits (see page 128)
G14	Tool change point (see page 128)
A	Starting angle defines the machining area at the contour starting point
W	End angle defines the machining area at the contour end point
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
G47	Safety clearance (see page 128)
MT	M after T: M function that is executed after the tool call T







MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Recess turning**

Cycle run

1. Approach workpiece from starting point on paraxial path
2. Finish first side and contour area up to position just before **end point X2, Z2**
3. Approach workpiece for finishing the second side on paraxial path
4. Finish second side, then finish remainder of contour valley
5. Return to starting point on paraxial path
6. Move to the tool change point according to the G14 setting

Undercutting type H



Call the **recessing cycles**

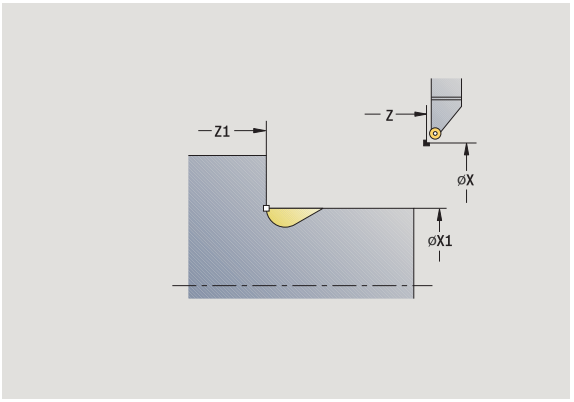


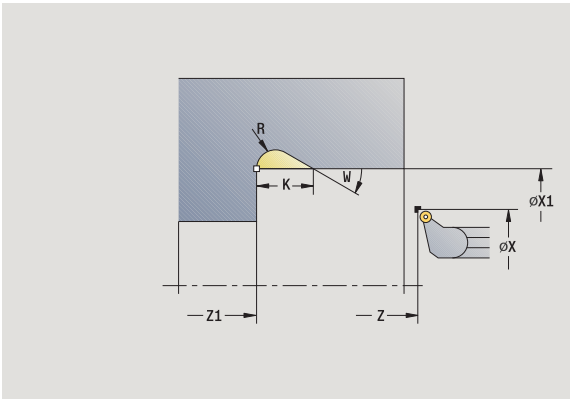
Select **undercutting H**

The contour depends on the parameters defined. If you do not define an **undercut radius R,** the oblique cut will be executed up to **contour corner Z1** (tool radius = undercut radius).

If you do not define the **plunging angle,** it is calculated from the **undercut length** and **undercut radius.** The final point of the undercut is then located at the **contour corner.**

The end point of the undercut is determined from the plunging angle in accordance with **Undercut type H.**





Cycle parameters

X, Z	Starting point
X1, Z1	Corner point of contour
K	Undercut length
R	Undercut radius (default: no circular element)
W	Plunging angle (default: W is calculated)
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Finishing**

Cycle run

1. Approach the workpiece from the starting point to the safety clearance
2. Machine undercut according to cycle parameters.
3. Return to starting point on diagonal path
4. Move to the tool change point according to the G14 setting

Undercutting type K

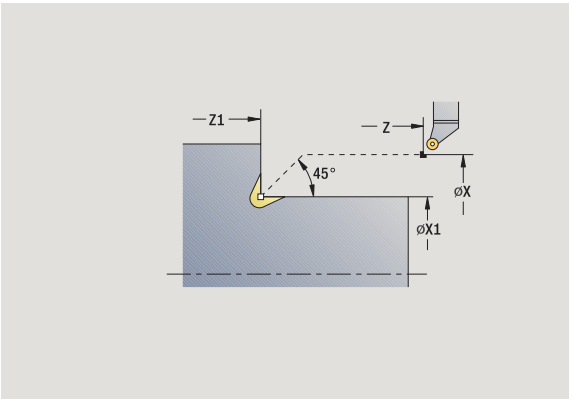


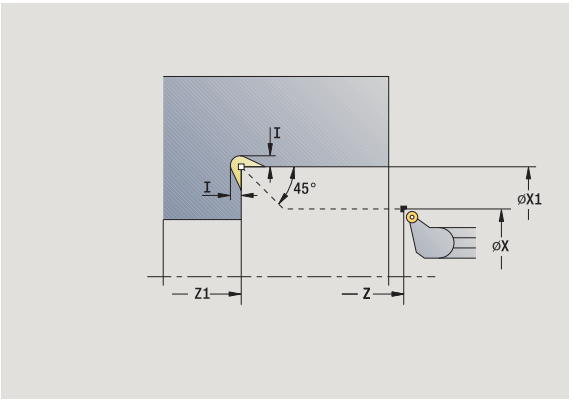
Call the **recessing cycles**



Select **undercut K**

This cycle performs only one cut at an angle of 45°. The resulting contour geometry therefore depends on the tool that is used.





Cycle parameters

X, Z	Starting point
X1, Z1	Corner point of contour
I	Undercut depth
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent) ■ Main drive ■ Opposing spindle for rear-face machining

Type of machining for technology database access: **Finishing**

Cycle run

1. Pre-position at an angle of 45° to safety clearance above the **contour corner point X1, Z1** at rapid traverse
2. Plunge by **undercut depth I**
3. Return the tool to the starting point on same path
4. Move to the tool change point according to the G14 setting

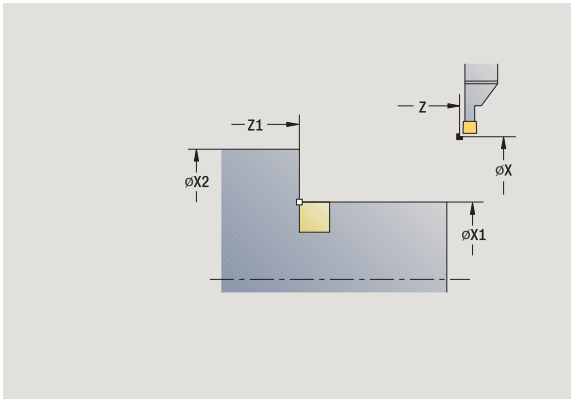
Undercutting type U



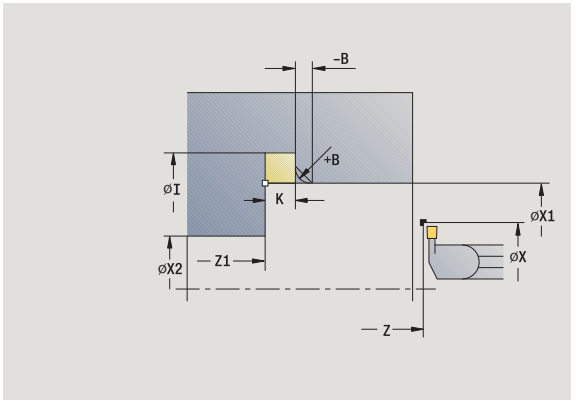
Call the **recessing cycles**



Select **undercutting U**



This cycle machines an **Undercut type U** and, if programmed, finishes the adjoining plane surface. The undercut is executed in several passes if the undercut width is greater than the cutting width of the tool. If the cutting width of the tool is not defined, the control assumes that the tool's cutting width equals the **undercut width.** Either a chamfer or a rounding arc can be machined.



Cycle parameters

X, Z	Starting point
X1, Z1	Corner point of contour
X2	End point on plane surface
I	Undercut diameter
K	Width of undercut
B	Chamfer/Rounding ■ B>0: Rounding radius ■ B<0: Chamfer width
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Finishing**

Cycle run

1. Calculate the proportioning of cuts
2. Approach the workpiece from the starting point to the safety clearance
3. Move at feed rate to **undercut diameter I** and dwell at this position (for two revolutions)
4. Retract and approach for next pass
5. Repeat 3 to 4 until the **corner point Z1** is reached
6. After the last pass, finish the adjoining plane surface, starting from **end point X2,** if defined
7. Machine chamfer/rounding, if defined.
8. Return to starting point on diagonal path
9. Move to the tool change point according to the G14 setting

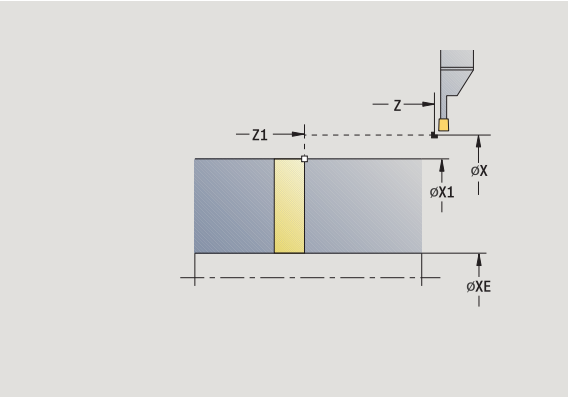
Parting



Call the **recessing cycles**



Select **parting**



The cycle parts the workpiece. If programmed, a chamfer or rounding arc is machined on the outside diameter.

Cycle parameters

X, Z	Starting point
X1, Z1	Corner point of contour
I	Diameter for feed-rate reduction
B	Chamfer/Rounding ■ B>0: Rounding radius ■ B<0: Chamfer width
E	Reduced feed rate
D	Maximum speed
K	Retraction distance after parting: move the tool sideways away from the workpiece surface before retracting
SD	Speed limitation from the diameter I up
U	Diameter from which the part catcher is activated (machine-dependent function)
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Parting**

Cycle run

1. Approach the workpiece from the starting point to the safety clearance
2. Cut to depth of chamfer or rounding and machine the chamfer/ rounding if defined
3. Depending on the cycle parameters, move at feed rate to
   - The turning center, or
   - The **inside diameter (tube) XE**

   If you have programmed a feed rate reduction, the CNC PILOT switches to the **reduced feed E** as soon as the tool reaches the **diameter feed reduction I**.
4. Retract at end face and return to starting point.
5. Move to the tool change point according to the G14 setting

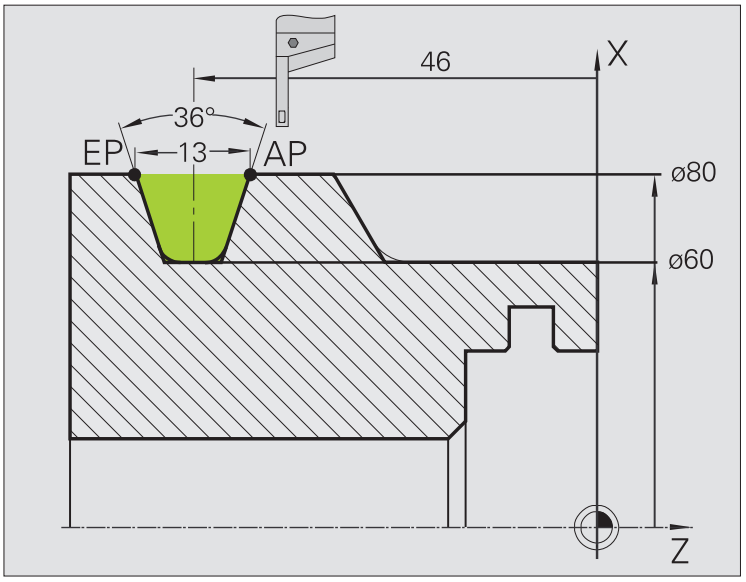


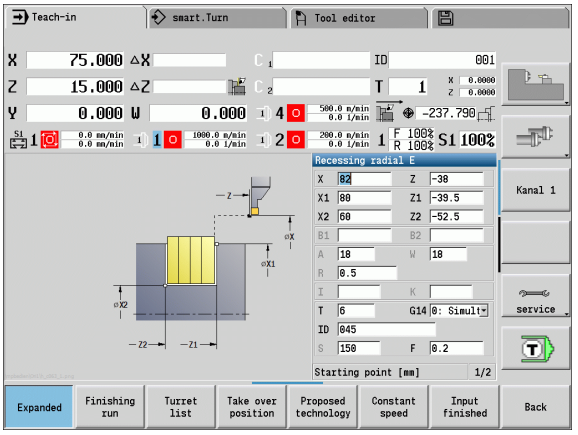
The limit to the maximum speed **"D"** is only effective in the cycle. After the cycle ends, the speed limit before the cycle become effective.

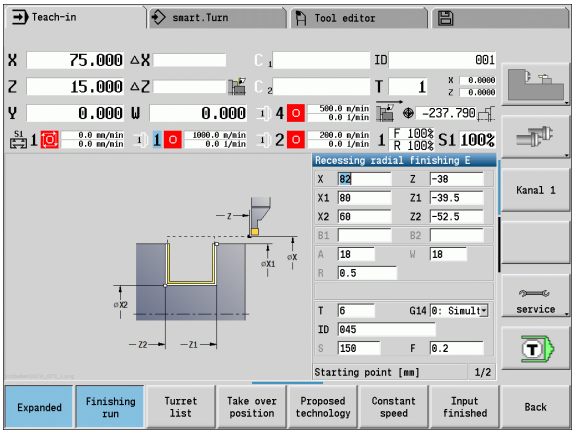


Examples of recessing cycles

Recess outside







The machining operation is to be executed first with the **Recessing, radial–expanded** cycle, taking oversizes into account. This contour area is to be finished subsequently with **Recessing, radial finishing, expanded.**

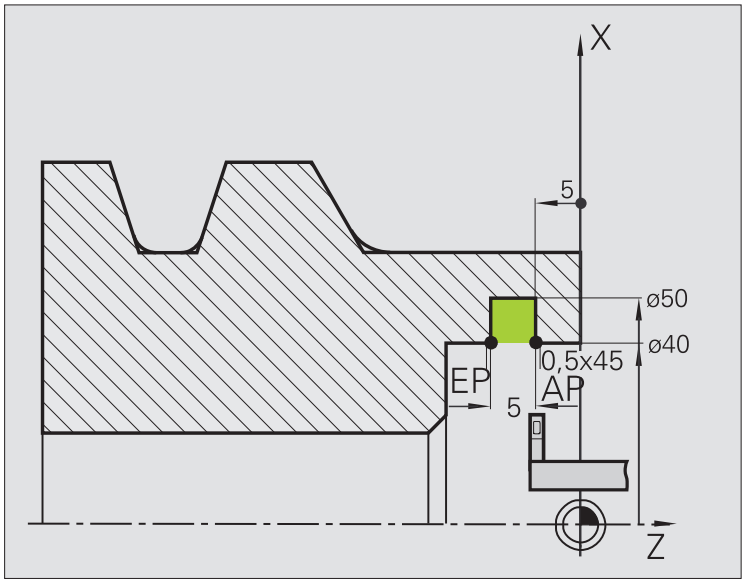
The rounding arcs in the corners of the contour valley and the oblique surfaces at the contour start and end are also machined in "expanded mode."

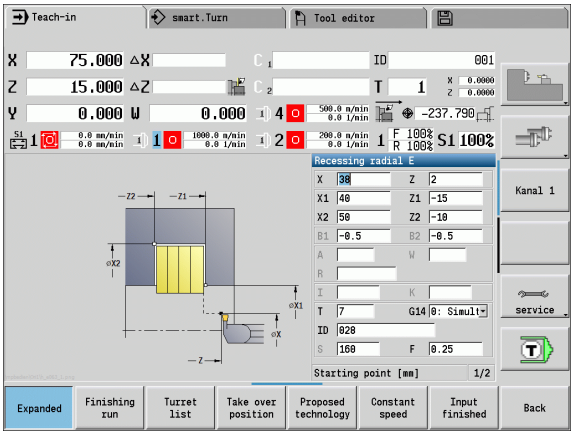
Be sure to enter the correct values for the parameters **contour starting point X1, Z1** and **contour end point X2, Z2.** These parameters determine the cutting and infeed directions—in this example, external machining and infeed in negative Z-axis direction.

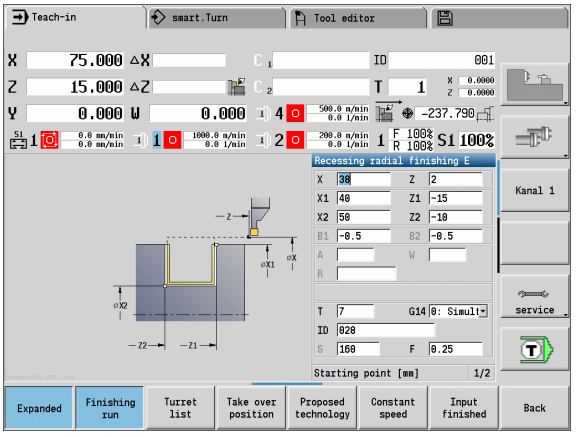
Tool data

- Turning tool (for external machining)
- WO = 1 (tool orientation)
- SB = 4 (cutting width 4 mm)

Recess inside







The machining operation is to be executed first with the **Recessing, radial–expanded** cycle, taking oversizes into account. This contour area is to be finished subsequently with **Recessing, radial finishing, expanded.**

Since the **plunge width P** is not entered, the CNC PILOT plunge-cuts with 80 % of the plunge-width of the tool.

In expanded mode, the chamfers are machined at the start/end of the contour.

Be sure to enter the correct values for the parameters **contour starting point X1, Z1** and **contour end point X2, Z2.** These parameters determine the cutting and infeed directions—in this example, internal machining and infeed in negative Z-axis direction.

Tool data

- Turning tool (for internal machining)
- WO = 7 (tool orientation)
- SB = 2 (cutting width 2 mm)

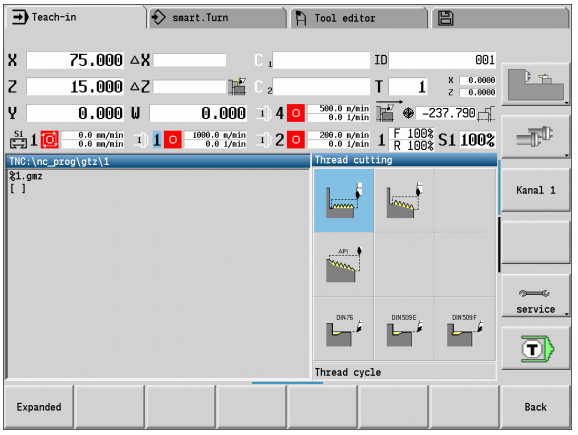
4.6 Thread and undercut cycles



The thread and undercut cycles machine single or multi-start longitudinal and tapered threads, as well as thread undercuts.

In Cycle mode you can

- Repeat the last cut to compensate for tool inaccuracies.
- Use the **Recut** function to rework damaged threads (only in Manual mode).



- Threads are cut with constant rotational speed.
- At a **cycle stop,** the tool retracts with the spindle still rotating. The cycle then has to be restarted.
- Feed rate override is not effective during cycle execution.

Thread position, undercut position

Thread position
The CNC PILOT determines the direction of the thread from the parameters for **`starting point Z`** (or current tool position in Manual mode) and **`end point Z2.`** You select internal or external thread by soft key.

Undercut position
The CNC PILOT determines the position of an undercut from the parameters for **`starting point X, Z`** (current tool position in Manual mode) and **`cylinder starting point X1/end point Z2 on plane surface.`**

An undercut can only be machined in a right-angled paraxial contour corner in the linear axis.

Thread and undercut cycles	Symbol
Thread cycle Longitudinal single or multi-start thread	
Tapered thread Tapered single or multi-start thread	
API thread Single or multi-start API thread (API: American Petroleum Institute)	
Undercut DIN 76 Thread undercut and thread chamfer	
Undercut DIN 509 E Undercut and cylinder chamfer	
Undercut DIN 509 F Undercut and cylinder chamfer	

Handwheel superimposition

If your machine features handwheel superimposition, you can overlap axis movements during thread cutting in a limited area:

- **X direction**: Maximum programmed thread depth depending on the current cutting depth
- **Z direction**: +/- a fourth of the thread pitch

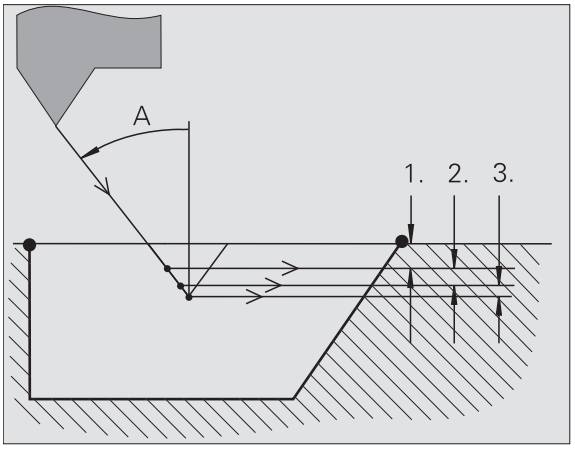
Machine and control must be specially prepared by the machine tool builder for use of this cycle. Refer to your machine tool manual.

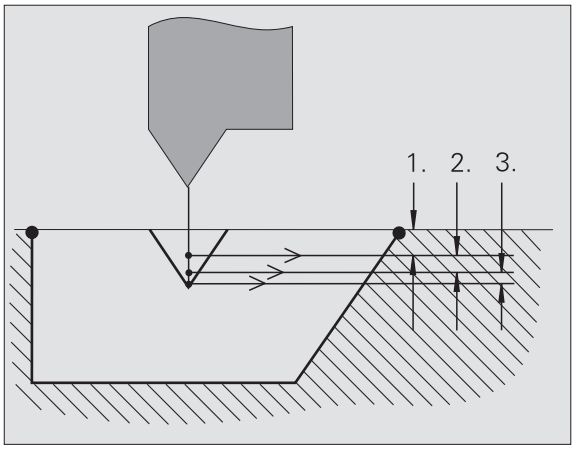
Remember that position changes resulting from handwheel superimposition are no longer effective after the cycle end or the "last cut" function.

Feed angle, thread depth, proportioning of cuts

With some thread cycles, you can indicate the angle of infeed (thread angle). The figures show the operating sequence at an angle of infeed of –30° and an angle of infeed of 0°.

The thread depth is programmed for all thread cycles. The CNC PILOT reduces the cutting depth with each cut (see figures).





Thread run-in / thread run-out

The slide requires a run-in distance to accelerate to the programmed feed rate before starting the actual thread, and a run-out distance at the end of the thread to decelerate again.

If the run-in / run-out length is too short, the thread may not attain the expected quality. In this case, the CNC PILOT displays a warning.

Last cut

After the cycle is finished, the CNC PILOT presents the **Last cut** option. In this way you can enter a tool compensation value and repeat the last thread cut.

SEQUENCE OF THE "LAST CUT" FUNCTION

Initial situation: The thread cut cycle has been run, and the thread depth is not correct.

Perform the tool compensation

Last cut

Press the **Last cut** soft key



Activate **Cycle Start**

Check the thread



The tool compensation and the **last cut** can be repeated as often as necessary until the thread is correct.

Thread cycle (longitudinal)



Call the **thread-cutting** menu



Select **thread cycle**

Inner thread

- **On:** Internal thread
- **Off:** External thread

This cycle cuts a single external or internal thread with a thread angle of 30°. Tool infeed is performed in the X axis only.

Cycle parameters

X, Z Starting point of thread

Z2 End point of thread

F1 Thread pitch (= feed rate)

U Thread depth – No input:

- External thread: U=0.6134*F1
- Internal thread: U=–0.5413*F1

I Maximum infeed

- I<U: First cut with cutting depth I; every further cut: reduction of cutting depth
- I=U: One cut
- No input: I is calculated from U and F1

G47 Safety clearance (see page 128)

G14 Tool change point (see page 128)

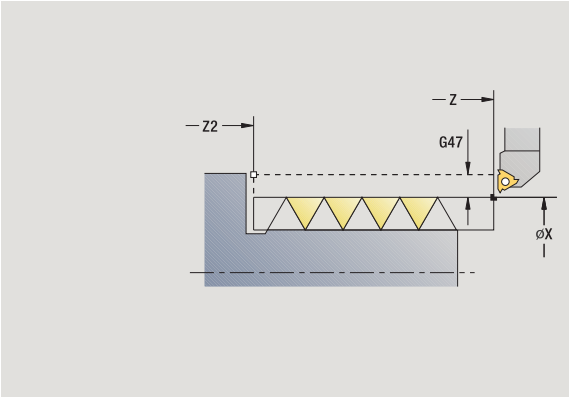
T Turret pocket number

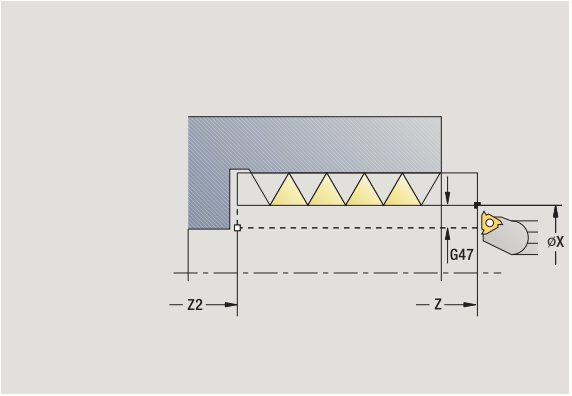
ID Tool ID number

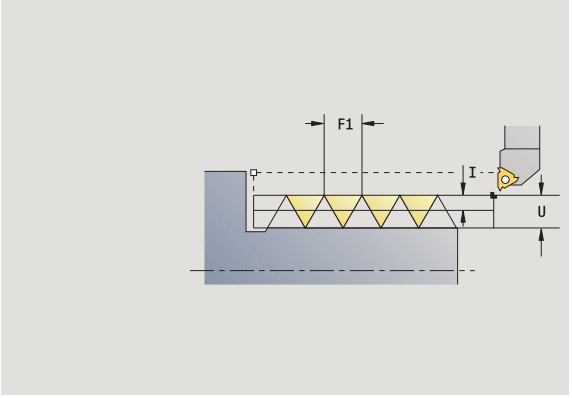
S Spindle speed/cutting speed

GV Type of infeed

- 0: Constant mach. X-section
- 1: Constant infeed
- 2: W/ remaining cutting (with distribution of remaining cuts)
- 3: W/o remaining cutting (without distribution of remaining cuts)
- 4: Same as MANUALplus 4110
- 5: Constant infeed (same as 4290)
- 6: Constant with distribute. (same as 4290)







GH Type of offset

- 0: Without offset
- 1: From left
- 2: From right
- 3: Alternately left/right

A Approach (infeed) angle (range: –60° < A < 60°; default 30°)

- A<0: Infeed on left thread flank
- A>0: Infeed on right thread flank

R Remaining cutting depth only with GV=4 (default 1/100 mm)

IC Number of cuts—the infeed is calculated from IC and U.

Usable with:

- GV=0: Constant chip cross section
- GV=1: Constant infeed

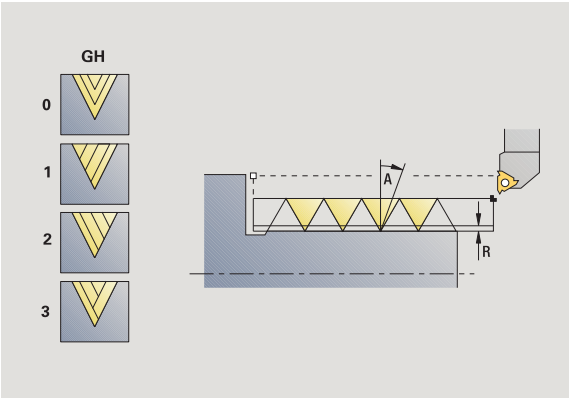
MT M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining



Type of machining for technology database access: **thread cutting**

Cycle run

1. Calculate the proportioning of cuts
2. Start first pass at **`starting point Z`**
3. Move at the programmed feed rate to the **`contour end point Z2`**
4. Return on paraxial path and approach for next pass
5. Repeat 3 to 4 until the **`thread depth U`** is reached
6. Move to the tool change point according to the G14 setting

Thread cycle (longitudinal)—expanded



Call the **thread-cutting** menu



Select **thread cycle**

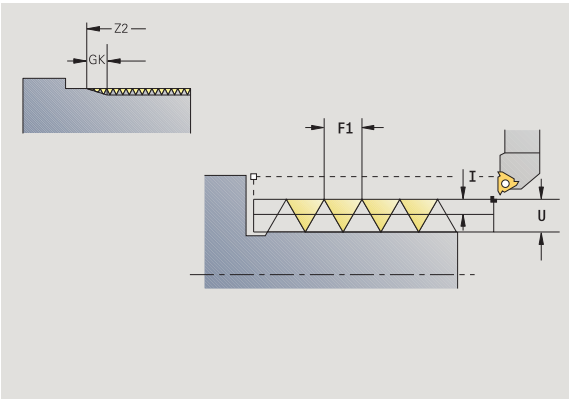
Expanded

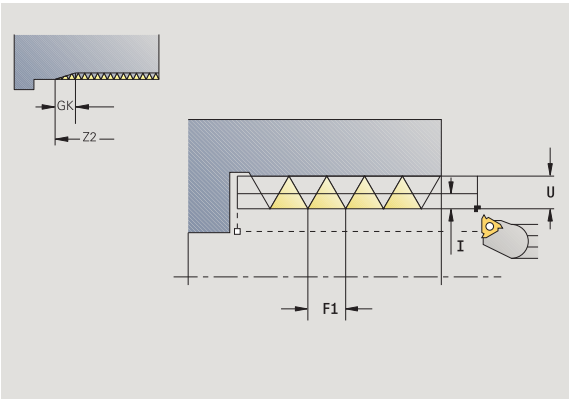
Press the **Expanded** soft key

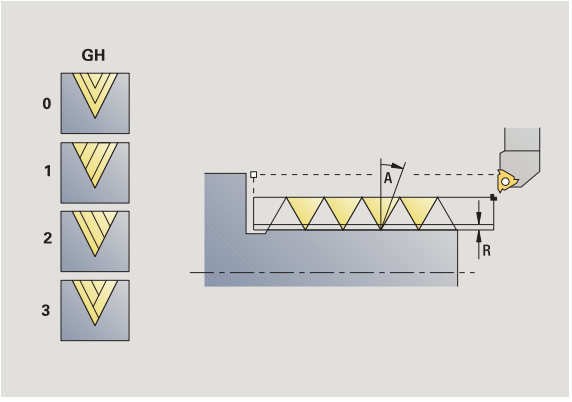
Inner thread

- **On:** Internal thread
- **Off:** External thread

This cycle cuts a single or multi-start external or internal thread. The thread starts at the **starting point** and ends at the **end point of thread** (without a thread run-in or run-out).







Cycle parameters

X, Z	Starting point of thread
Z2	End point of thread
F1	Thread pitch (= feed rate)
D	Number of thread starts (default: 1 single-start thread)
U	Thread depth – No input: ■ External thread: U=0.6134*F1 ■ Internal thread: U=–0.5413*F1
I	Maximum infeed ■ I<U: First cut with cutting depth I; every further cut: reduction of cutting depth ■ I=U: One cut ■ No input: I is calculated from U and F1
GK	Run-out length
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
GH	Type of offset ■ 0: Without offset ■ 1: From left ■ 2: From right ■ 3: Alternately left/right

GV Type of infeed

- 0: Constant mach. X-section
- 1: Constant infeed
- 2: W/ remaining cutting (with distribution of remaining cuts)
- 3: W/o remaining cutting (without distribution of remaining cuts)
- 4: Same as MANUALplus 4110
- 5: Constant infeed (same as 4290)
- 6: Constant with distribute. (same as 4290)

A Approach (infeed) angle (range: –60° < A < 60°; default 30°)

- A<0: Infeed on left thread flank
- A>0: Infeed on right thread flank

R Remaining cutting depth only with GV=4 (default 1/100 mm)

E Variable thread pitch (e.g. for manufacturing spiral conveyors or extrusion shafts)

Q No. no load (number of dry runs)

IC Number of cuts—the infeed is calculated from IC and U.

Usable with:

- GV=0: Constant chip cross section
- GV=1: Constant infeed

MT M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **thread cutting**

Cycle run

1. Calculate the proportioning of cuts
2. Start the first thread groove at **starting point Z**
3. Move at the programmed feed rate to the **contour end point Z2**
4. Return on paraxial path and approach for next thread groove
5. Repeat 3 and 4 for all thread grooves
6. Approach for next pass, taking the **reduced cutting depth** and the **feed angle A** into account
7. Repeat 3 to 6 until **no. threads D** and **depth U** are reached
8. Move to the tool change point according to the G14 setting

Tapered thread



Call the **thread-cutting** menu

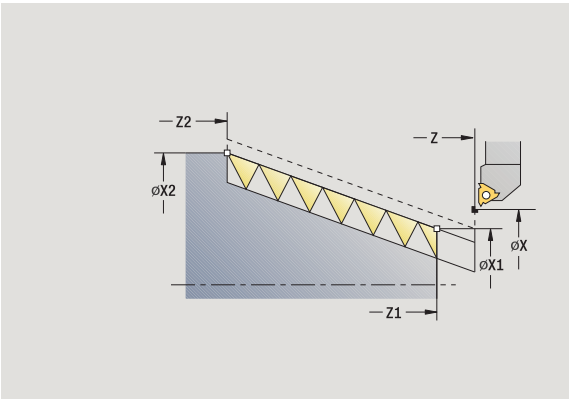


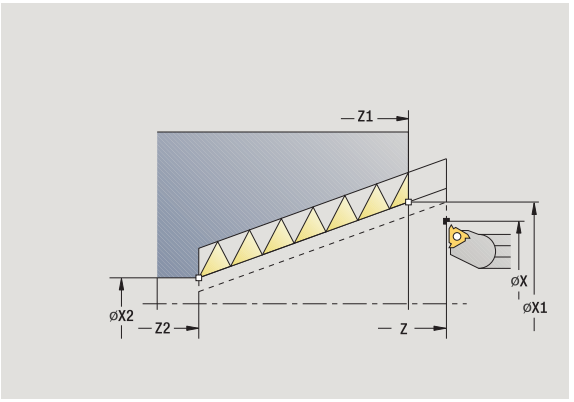
Select **tapered thread**

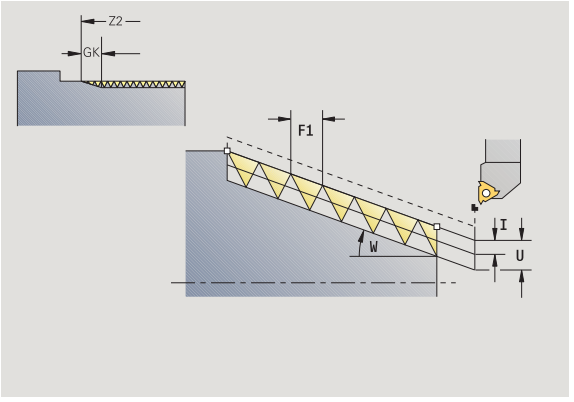
Inner thread

- **On:** Internal thread
- **Off:** External thread

This cycle cuts a single or multi-start tapered external or internal thread.







Cycle parameters

X, Z	Starting point
X1, Z1	Starting point of thread
X2, Z2	End point of thread
F1	Thread pitch (= feed rate)
D	Number of thread starts (default: 1 single-start thread)
U	Thread depth – No input:

- External thread: U=0.6134*F1
- Internal thread: U=–0.5413*F1

I	Maximum infeed

- I<U: First cut with cutting depth I; every further cut: reduction of cutting depth
- I=U: One cut
- No input: I is calculated from U and F1

W	Taper angle (range: –60° < A < 60°)
GK	Run-out length
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
GV	Type of infeed

- 0: Constant mach. X-section
- 1: Constant infeed
- 2: W/ remaining cutting (with distribution of remaining cuts)
- 3: W/o remaining cutting (without distribution of remaining cuts)
- 4: Same as MANUALplus 4110
- 5: Constant infeed (same as 4290)
- 6: Constant with distribute. (same as 4290)

GH Type of offset

- 0: Without offset
- 1: From left
- 2: From right
- 3: Alternately left/right

A Approach (infeed) angle (range: –60° < A < 60°; default 30°)

- A<0: Infeed on left thread flank
- A>0: Infeed on right thread flank

R Remaining cutting depth only with GV=4 (default 1/100 mm)

E Variable thread pitch (e.g. for manufacturing spiral conveyors or extrusion shafts)

Q No. no load (number of dry runs)

IC Number of cuts—the infeed is calculated from IC and U.

Usable with:

- GV=0: Constant chip cross section
- GV=1: Constant infeed

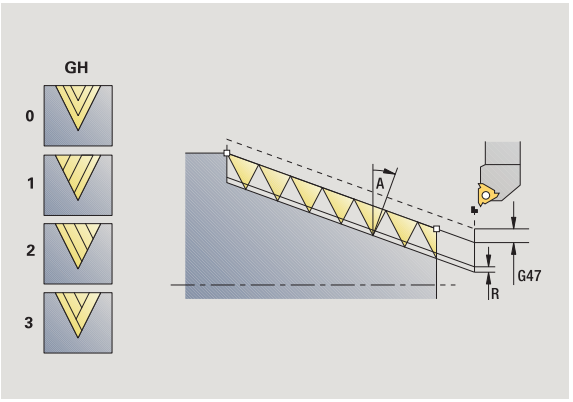
MT M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining



Type of machining for technology database access: **thread cutting**

Parameter combinations for the taper angle:

- X1/Z1, X2/Z2
- X1/Z1, Z2, W
- Z1, X2/Z2, W

Cycle run

1. Calculate the proportioning of cuts
2. Move to **thread starting point X1, Z1.**
3. Move at the programmed feed rate to the **contour end point Z2**
4. Return on paraxial path and approach for next thread groove
5. Repeat 3 and 4 for all thread grooves
6. Approach for next pass, taking the **reduced cutting depth** and the **feed angle A** into account
7. Repeat 3 to 6 until **no. threads D** and **depth U** are reached
8. Move to the tool change point according to the G14 setting

API thread



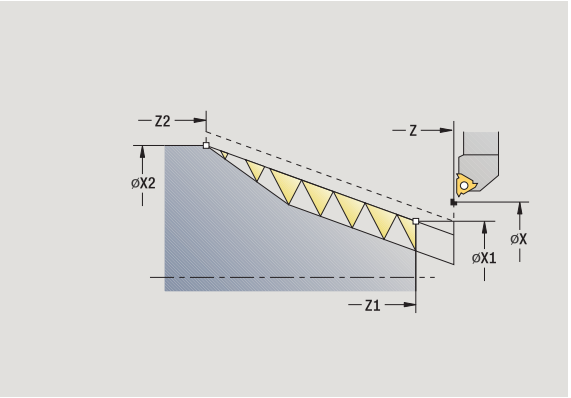
Call the **thread-cutting** menu



Select **API thread**

Inner thread

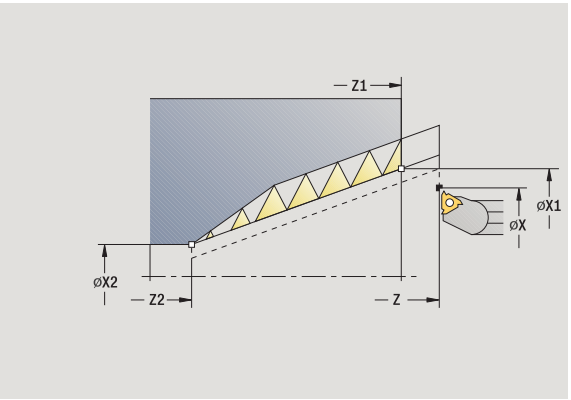
- **On:** Internal thread
- **Off:** External thread

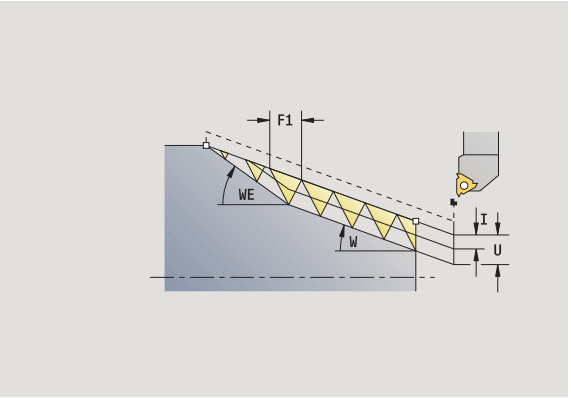


This cycle cuts a single or multi-start API external or internal thread. The depth of thread decreases at the overrun at the end of thread.

Cycle parameters

X, Z	Starting point
X1, Z1	Starting point of thread
X2, Z2	End point of thread
F1	Thread pitch (= feed rate)
D	Number of thread starts (default: 1 single-start thread)
U	Thread depth – No input: ■ External thread: U=0.6134*F1 ■ Internal thread: U=–0.5413*F1
I	First cutting depth ■ I<U: First cut with cutting depth I—further cuts: Reduction of cutting depth down to J. ■ I=U: One cut ■ No input: Calculation from U and F1
WE	Run-out angle (range: 0° < WE < 90°)
W	Taper angle (range: –60° < A < 60°)
G47	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
GV	Type of infeed ■ 0: Constant mach. X-section ■ 1: Constant infeed ■ 2: W/ remaining cutting (with distribution of remaining cuts) ■ 3: W/o remaining cutting (without distribution of remaining cuts) ■ 4: Same as MANUALplus 4110 ■ 5: Constant infeed (same as 4290) ■ 6: Constant with distribute. (same as 4290)





GH Type of offset
- 0: Without offset
- 1: From left
- 2: From right
- 3: Alternately left/right

A Approach (infeed) angle (range: –60° < A < 60°; default 30°)
- A<0: Infeed on left thread flank
- A>0: Infeed on right thread flank

R Remaining cutting depth only with GV=4 (default 1/100 mm)

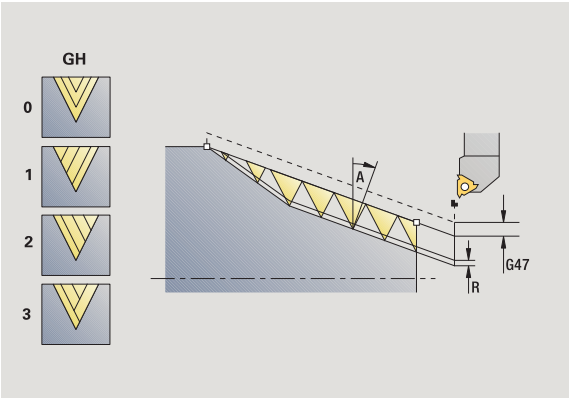
Q No. no load (number of dry runs)

MT M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)
- Main drive
- Opposing spindle for rear-face machining



Type of machining for technology database access: **thread cutting**

Parameter combinations for the taper angle:
- X1/Z1, X2/Z2
- X1/Z1, Z2, W
- Z1, X2/Z2, W

Cycle run

1. Calculate the proportioning of cuts
2. Move to **thread starting point X1, Z1.**
3. Move to **end point Z2** at programmed feed rate, taking the **run-out angle WE** into account
4. Return on paraxial path and approach for next thread groove
5. Repeat 3 and 4 for all thread grooves
6. Approach for next pass, taking the **reduced cutting depth** and the **feed angle A** into account
7. Repeat 3 to 6 until **no. threads D** and **depth U** are reached.
8. Move to the tool change point according to the G14 setting

Recut (longitudinal) thread



Call the **thread-cutting** menu



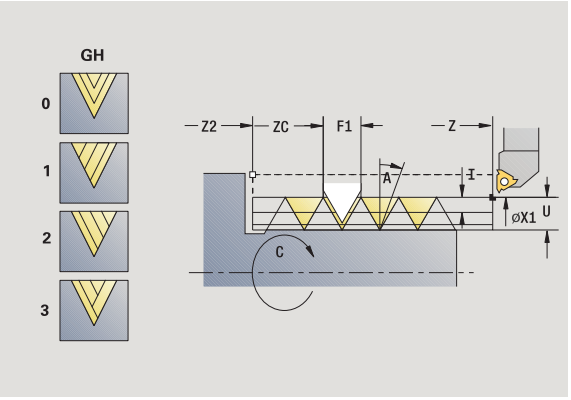
Select **thread cycle**

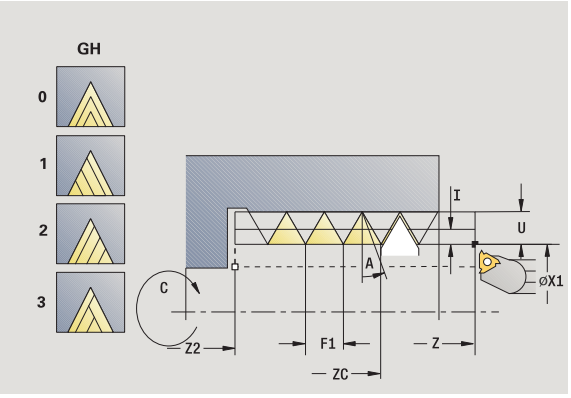
Re-
cut

Press the **Recut** soft key

Inner
thread

- **On:** Internal thread
- **Off:** External thread





This optional cycle reworks a single-start thread. Since you have already unclamped the workpiece, the CNC PILOT needs to know the exact position of the thread. Place the cutting tip of the tap drill in the center of a groove and transfer the positions to the parameters **measured angles** and **measured position** by pressing the **Take over position** soft key. From these values the cycle then calculates the angle of the spindle at the starting point.

This function is available only in manual operation.

Cycle parameters

X1 Starting point of thread

Z2 End point of thread

F1 Thread pitch (= feed rate)

U Thread depth – No input:

- External thread: U=0.6134*F1
- Internal thread: U=–0.5413*F1

I Maximum infeed

- I<U: First cut with cutting depth I—further cuts: Reduction of cutting depth
- I=U: One cut
- No input: Calculation from U and F1

C Measured angle

ZC Measured position

A Approach (infeed) angle (range: –60° < A < 60°; default 30°)

- A<0: Infeed on left thread flank
- A>0: Infeed on right thread flank

R Remaining cutting depth only with GV=4 (default 1/100 mm)

MT M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Cycle run

1. Pre-position threading tool to center of thread groove
2. Use the **Take over position** soft key to transfer the tool position and spindle angle to the parameters **measured positions ZC** and **measured angle C**
3. Move the tool manually out of the thread groove
4. Position the tool to the starting point
5. Start cycle with the **Input finished** soft key, then press **Cycle START**

Recut (longitudinal) thread—expanded



Call the **thread-cutting** menu



Select **thread cycle**

Expanded

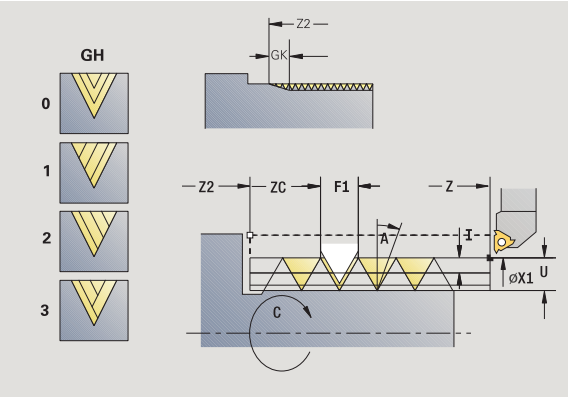
Press the **Expanded** soft key

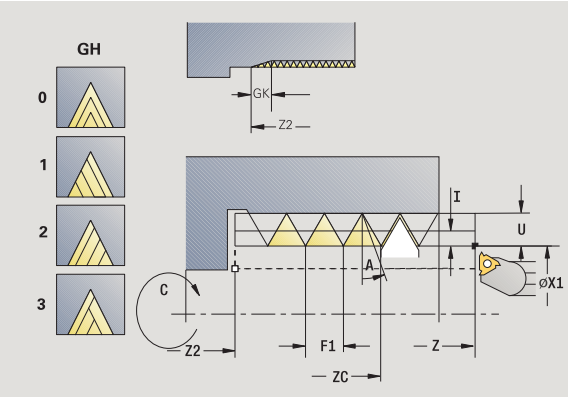
Re-
cut

Press the **Recut** soft key

Inner
thread

- **On:** Internal thread
- **Off:** External thread





This optional cycle recuts a single or multi-start external or internal thread. Since you have already unclamped the workpiece, the CNC PILOT needs to know the exact position of the thread. Place the cutting tip of the tap drill in the center of a groove and transfer the positions to the parameters **measured angles** and **measured position** by pressing the **Take over position** soft key. From these values the cycle then calculates the angle of the spindle at the starting point.

This function is available only in manual operation.

Cycle parameters

X1 Starting point of thread

Z2 End point of thread

F1 Thread pitch (= feed rate)

D No. of gears (threads per unit)

U Thread depth – No input:

- External thread: U=0.6134*F1
- Internal thread: U=–0.5413*F1

I Maximum infeed

- I<U: First cut with cutting depth I—further cuts: Reduction of cutting depth
- I=U: One cut
- No input: Calculation from U and F1

GK Run-out length

C Measured angle

ZC Measured position

A Approach (infeed) angle (range: –60° < A < 60°; default 30°)

- A<0: Infeed on left thread flank
- A>0: Infeed on right thread flank

R	Remaining cutting depth only with GV=4 (default 1/100 mm)
Q	No. no load (number of dry runs)
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent) ■ Main drive ■ Opposing spindle for rear-face machining

Cycle run

1. Pre-position threading tool to center of thread groove
2. Use the **Take over position** soft key to transfer the tool position and spindle angle to the parameters **measured positions ZC** and **measured angle C**
3. Move the tool manually out of the thread groove
4. Position the tool to the starting point
5. Start cycle with the **Input finished** soft key, then press **Cycle START**

Recut tapered thread



Call the **thread-cutting** menu



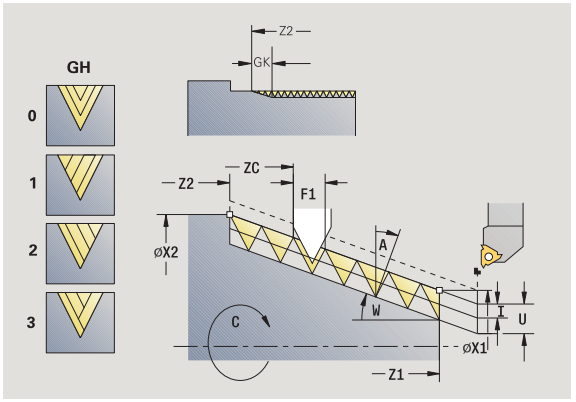
Select tapered thread

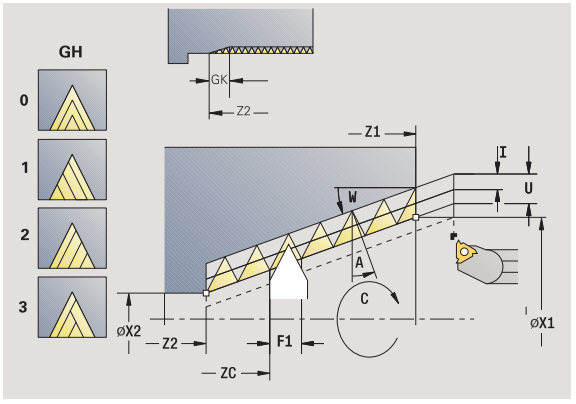


Press the **Recut** soft key



- **On:** Internal thread
- **Off:** External thread





This optional cycle recuts a single or multi-start external or internal taper thread. Since you have already unclamped the workpiece, the CNC PILOT needs to know the exact position of the thread. Place the cutting tip of the tap drill in the center of a groove and transfer the positions to the parameters **measured angles** and **measured position** by pressing the **Take over position** soft key. From these values the cycle then calculates the angle of the spindle at the starting point.

This function is available only in manual operation.

Cycle parameters

X1, Z1 Starting point of thread

X2, Z2 End point of thread

F1 Thread pitch (= feed rate)

D No. of gears (threads per unit)

U Thread depth – No input:

- External thread: U=0.6134*F1
- Internal thread: U=–0.5413*F1

I Maximum infeed

- I<U: First cut with cutting depth I—further cuts: Reduction of cutting depth
- I=U: One cut
- No input: Calculation from U and F1

W Taper angle (range: –60° < A < 60°)

GK Run-out length

C Measured angle

ZC Measured position

A Approach (infeed) angle (range: –60° < A < 60°; default 30°)

- A<0: Infeed on left thread flank
- A>0: Infeed on right thread flank

R	Remaining cutting depth only with GV=4 (default 1/100 mm)
Q	No. no load (number of dry runs)
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent) ■ Main drive ■ Opposing spindle for rear-face machining

Cycle run

1. Pre-position threading tool to center of thread groove
2. Use the **Take over position** soft key to transfer the tool position and spindle angle to the parameters **measured positions ZC** and **measured angle C**
3. Move the tool manually out of the thread groove
4. Position the tool **in front of** the workpiece
5. Start cycle with the **Input finished** soft key, then press **Cycle START**

Recut API thread



Call the **thread-cutting** menu



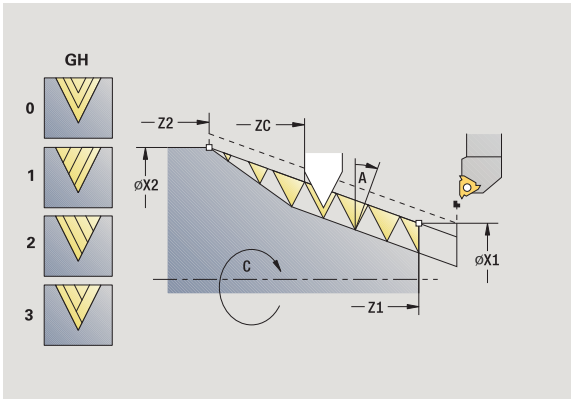
Select API thread

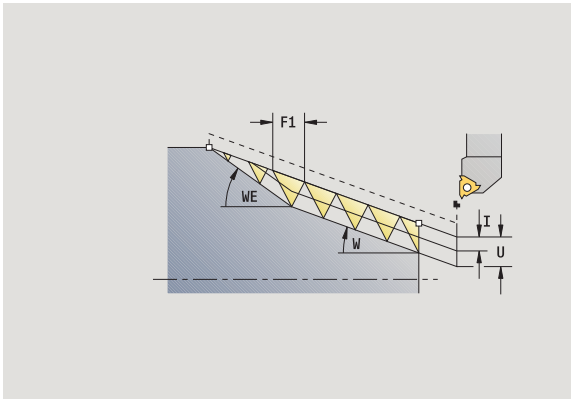


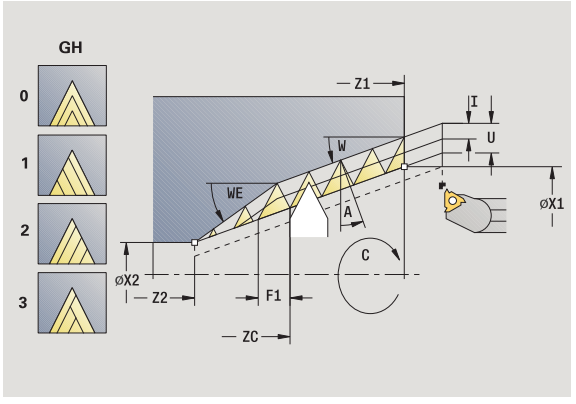
Press the **Recut** soft key



- **On:** Internal thread
- **Off:** External thread







This optional cycle recuts a single or multi-start external or internal API thread. Since you have already unclamped the workpiece, the CNC PILOT needs to know the exact position of the thread. Place the cutting tip of the tap drill in the center of a groove and transfer the positions to the parameters **measured angles** and **measured position** by pressing the **Take over position** soft key. From these values the cycle then calculates the angle of the spindle at the starting point.

This function is available only in manual operation.

Cycle parameters

X1, Z1 Starting point of thread

X2, Z2 End point of thread

F1 Thread pitch (= feed rate)

D No. of gears (threads per unit)

U Thread depth – No input:

- External thread: U=0.6134*F1
- Internal thread: U=–0.5413*F1

I Maximum infeed

- I<U: First cut with cutting depth I—further cuts: Reduction of cutting depth
- I=U: One cut
- No input: Calculation from U and F1

WE Run-out angle (range: 0° < WE < 90°)

W Taper angle (range: –60° < A < 60°)

C Measured angle

ZC Measured position

A Approach (infeed) angle (range: –60° < A < 60°; default 30°)

- A<0: Infeed on left thread flank
- A>0: Infeed on right thread flank

R	Remaining cutting depth only with GV=4 (default 1/100 mm)
Q	No. no load (number of dry runs)
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent) ■ Main drive ■ Opposing spindle for rear-face machining

Cycle run

1. Pre-position threading tool to center of thread groove
2. Use the **Take over position** soft key to transfer the tool position and spindle angle to the parameters **measured positions ZC** and **measured angle C**
3. Move the tool manually out of the thread groove
4. Position the tool **in front of** the workpiece
5. Start cycle with the **Input finished** soft key, then press **Cycle START**

Undercut DIN 76



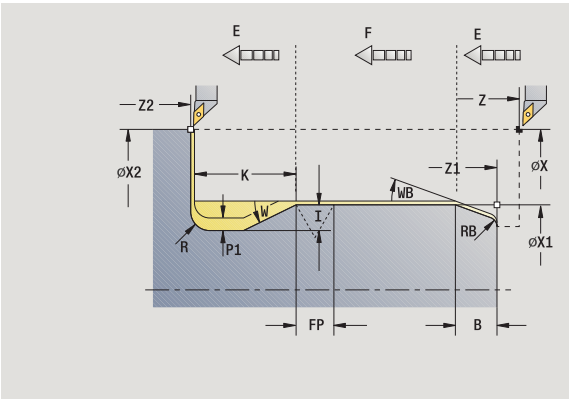
Call the **thread-cutting** menu

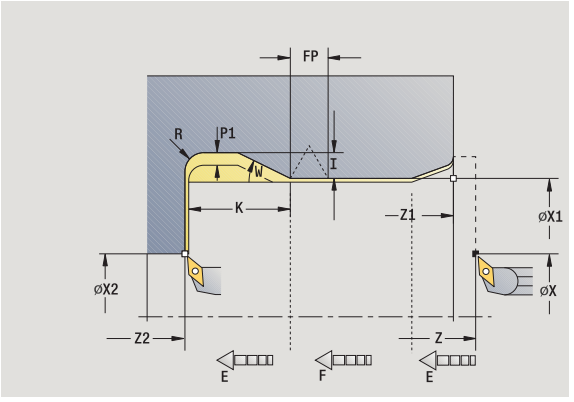


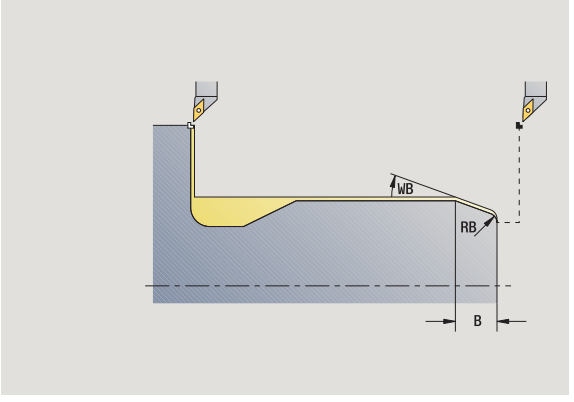
Select Undercut DIN 76

With return

- **Off:** When the cycle is completed, the tool remains at the cycle end position
- **On:** Tool returns to the starting point







The cycle machines a thread undercut according to DIN 76, a thread chamfer, then the cylinder, and finishes with the plane surface. The thread chamfer is executed when you enter at least one of the parameters **cylinder 1st cut length** or **1st cut radius.**

Cycle parameters

X, Z	Starting point
X1, Z1	Starting point of cylinder
X2, Z2	End point on plane surface
FP	Thread pitch (default: value from standard table)
E	Reduced feed rate for the plunge cut and the thread chamfer (default: feed rate F)
I	Undercut depth (default: value from standard table)
K	Undercut length (default: value from standard table)
W	Undercut angle (default: value from standard table)
R	Undercut radius on both sides of the undercut (default: value from standard table)
P1	Undercut oversize ■ No input: Machining in one cut ■ P>0: Division into pre-turning and finish-turning. P is longitudinal oversize; the transverse oversize is preset to 0.1 mm
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution

B	Cylinder start chamfer (default: no start chamfer)
WB	First-cut angle (default: 45°)
RB	First-cut radius (default: no input = no element): Positive value: First-cut radius, negative value = chamfer
G47	Safety clearance (see page 128)—evaluated only if "Omit paths of rapid traverse" is active
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent) ■ Main drive ■ Opposing spindle for rear-face machining

Type of machining for technology database access: **Finishing**

All parameters that you enter will be accounted for—even if the standard table prescribes other values. If the parameters I, K, W and R are not defined, the CNC PILOT determines these parameters from "FP" in the standard table (see "DIN 76—undercut parameters" on page 579).

Cycle run

1. Approach workpiece from starting point
   - to the **`cylinder starting point X1,`** or
   - for the **`thread chamfer`**
2. Machine thread chamfer, if defined
3. Finish cylinder up to beginning of undercut
4. Pre-machine undercut, if defined
5. Machine undercut
6. Finish to **`end point X2 on plane surface`**
7. Return
   - **Without return:** Tool remains at the **`end point on plane surface`**
   - **With return:** Retract and return diagonally to starting point
8. Move to the tool change point according to the G14 setting

Undercut DIN 509 E



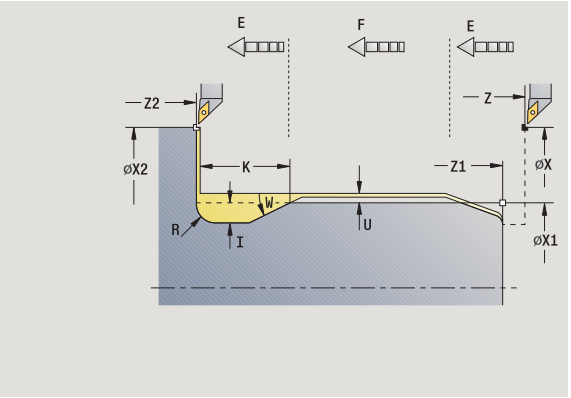
Call the **thread-cutting** menu

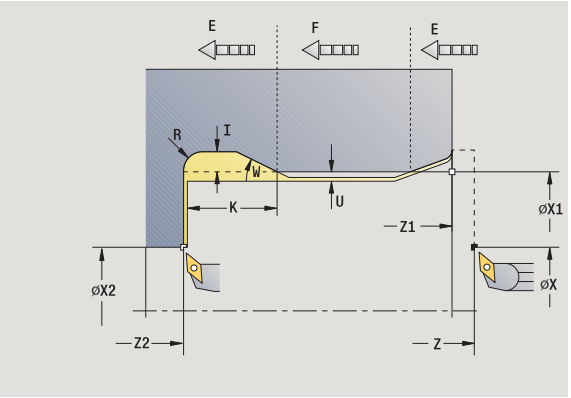


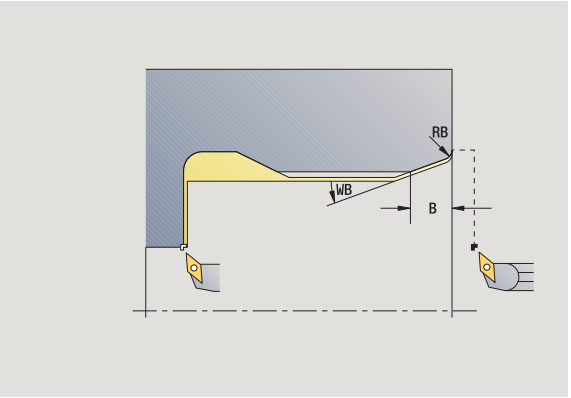
Select undercut DIN 509 E

With return

- **Off:** When the cycle is completed, the tool remains at the cycle end position
- **On:** Tool returns to the starting point







The cycle machines a thread undercut according to DIN 509 type E, a cylinder start chamfer, then the adjoining cylinder, and finishes with the plane surface. You can define a finishing oversize for the area of the cylinder. The cylinder chamfer is executed when you enter at least one of the parameters **cylinder 1st cut length** or **1st cut radius.**

Cycle parameters

X, Z	Starting point
X1, Z1	Starting point of cylinder
X2, Z2	End point on plane surface
U	Grinding oversize for the area of the cylinder (default: 0)
E	Reduced feed rate for the plunge cut and the cylinder chamfer (default: feed rate F)
I	Undercut depth (default: value from standard table)
K	Undercut length (default: value from standard table)
W	Undercut angle (default: value from standard table)
R	Undercut radius on both sides of the undercut (default: value from standard table)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
B	Cylinder start chamfer (default: no start chamfer)
WB	1st cut angle (default: 45°)
RB	First-cut radius (default: no input = no element): Positive value: First cut radius, negative value = chamfer
G47	Safety clearance (see page 128)—evaluated only if "Omit paths of rapid traverse" is active
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Finishing**

All parameters that you enter will be accounted for—even if the standard table prescribes other values. If the parameters I, K, W and R are not defined, the CNC PILOT determines these parameters from the cylinder diameter in the standard table (see "DIN 509 E – undercut parameters" on page 581).

Cycle run

1. Approach workpiece from starting point
   - to the **`cylinder starting point X1,`** or
   - for the **`thread chamfer`**
2. Machine thread chamfer, if defined
3. Finish cylinder up to beginning of undercut
4. Machine undercut
5. Finish to **`end point X2 on plane surface`**
6. Return
   - **Without return:** Tool remains at the **`end point on plane surface`**
   - **With return:** Retract and return diagonally to starting point
7. Move to the tool change point according to the G14 setting

Undercut DIN 509 F



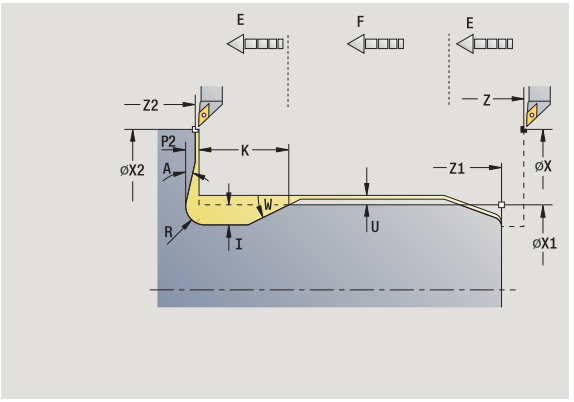
Call the **thread-cutting** menu

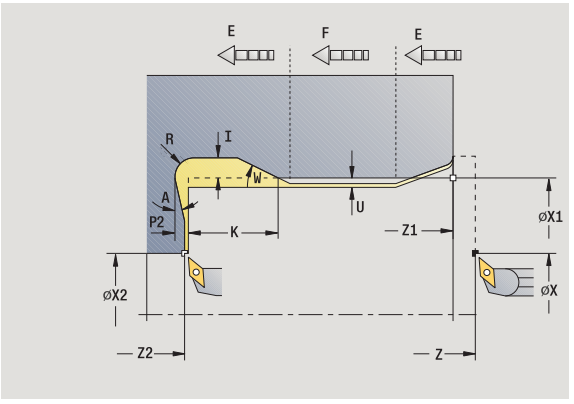


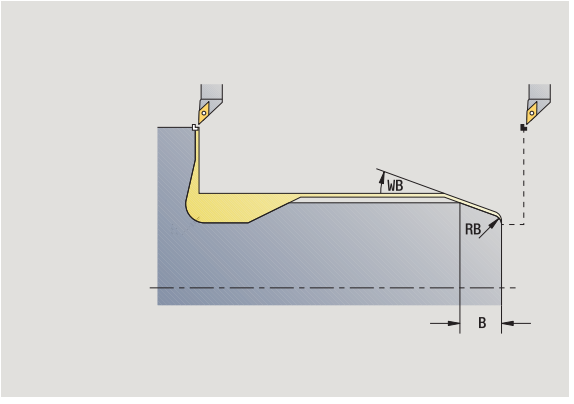
Select undercut DIN 509 F

With return

- **Off:** When the cycle is completed, the tool remains at the cycle end position
- **On:** Tool returns to the starting point







The cycle machines a thread undercut according to DIN 509 type F, a cylinder start chamfer, then the adjoining cylinder, and finishes with the plane surface. You can define a finishing oversize for the area of the cylinder. The cylinder chamfer is executed when you enter at least one of the parameters **cylinder 1st cut length** or **1st cut radius.**

Cycle parameters

X, Z	Starting point
X1, Z1	Starting point of cylinder
X2, Z2	End point on plane surface
U	Grinding oversize for the area of the cylinder (default: 0)
E	Reduced feed rate for the plunge cut and the cylinder chamfer (default: feed rate F)
I	Undercut depth (default: value from standard table)
K	Undercut length (default: value from standard table)
W	Undercut angle (default: value from standard table)
R	Undercut radius on both sides of the undercut (default: value from standard table)
P2	Face depth (default: value from standard table)
A	Face angle (default: value from standard table)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
B	Cylinder start chamfer (default: no start chamfer)
WB	1st cut angle (default: 45°)
RB	First-cut radius (default: no input = no element): Positive value: First cut radius, negative value = chamfer
G47	Safety clearance (see page 128)—evaluated only if "Omit paths of rapid traverse" is active

MT M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Finishing**

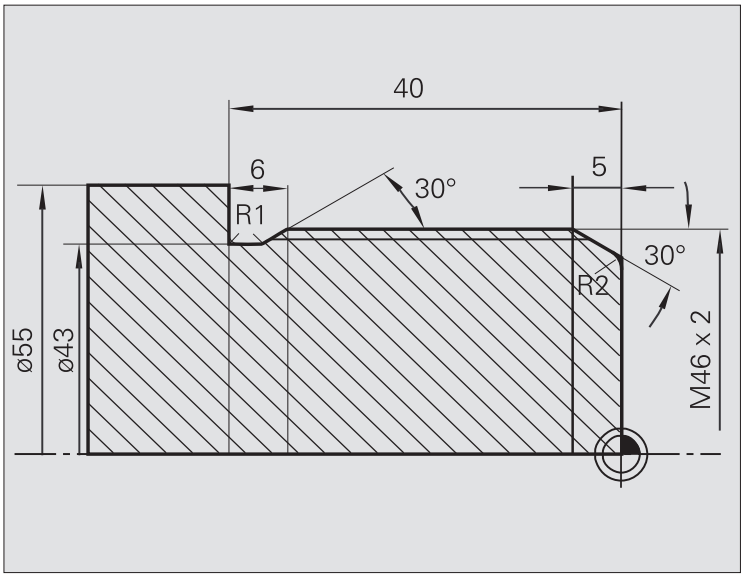
All parameters that you enter will be accounted for—even if the standard table prescribes other values. If the parameters I, K, W, R, P and A are not defined, the CNC PILOT determines these parameters from the cylinder diameter in the standard table (see "DIN 509 F – undercut parameters" on page 581).

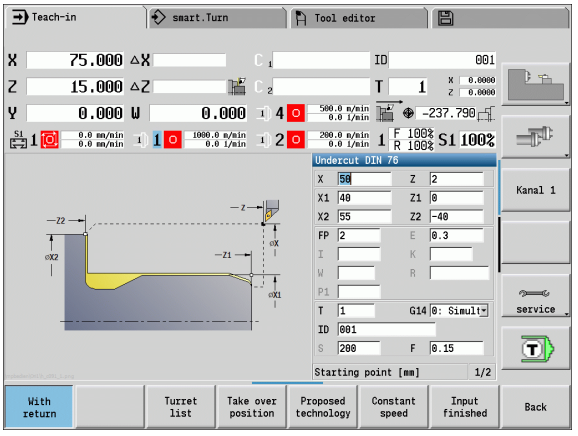
Cycle run

1. Approach workpiece from starting point
   - to the **`cylinder starting point X1,`** or
   - for the **`thread chamfer`**
2. Machine thread chamfer, if defined
3. Finish cylinder up to beginning of undercut
4. Machine undercut
5. Finish to **`end point X2 on plane surface`**
6. Return
   - **Without return:** Tool remains at the **`end point on plane surface`**
   - **With return:** Retract and return diagonally to starting point

Examples of thread and undercut cycles

External thread and thread undercut





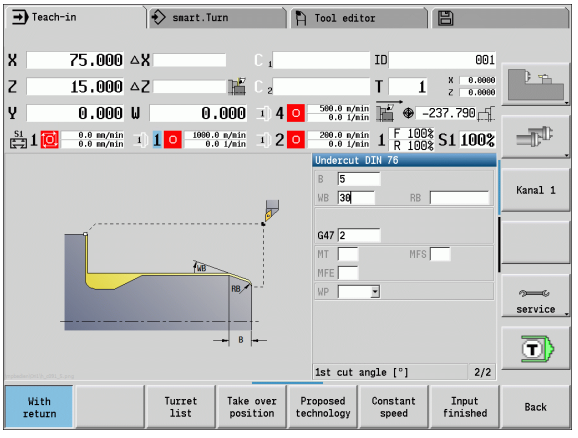
The machining operation is to be performed in two steps. The **thread undercut DIN 76** produces the undercut and thread chamfer. In the second step, the **thread cycle** cuts the thread.

First step

The parameters for the undercut and thread chamfer are programmed in two superimposed input windows.

Tool data

- Turning tool (for external machining)
- WO = 1 (tool orientation)
- A = 93° (tool angle)
- B = 55° (point angle)

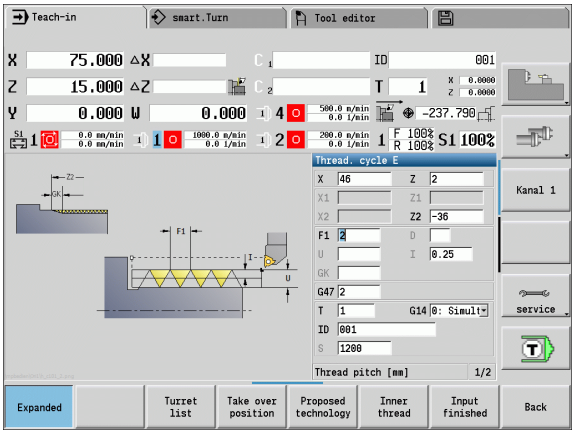


Second step

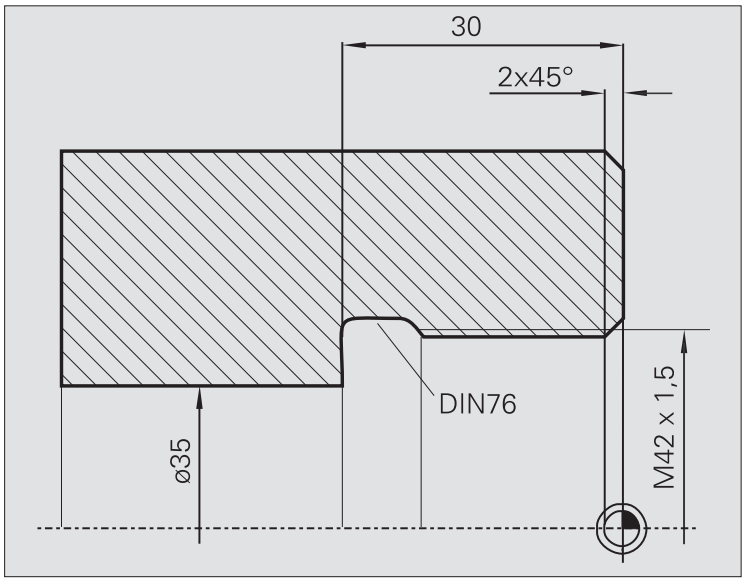
The **Thread cycle (longitudinal)–expanded** cuts the thread. The cycle parameters define the thread depth and the proportioning of cuts.

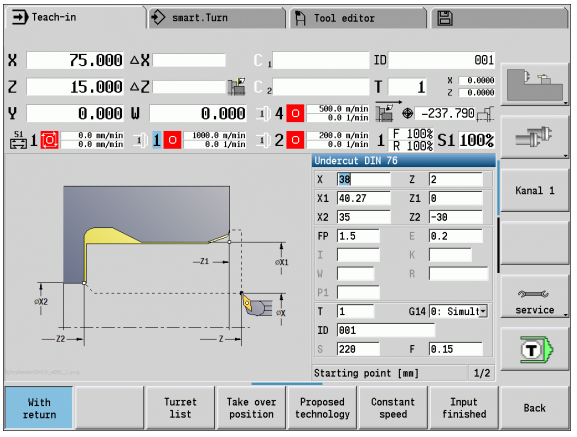
Tool data

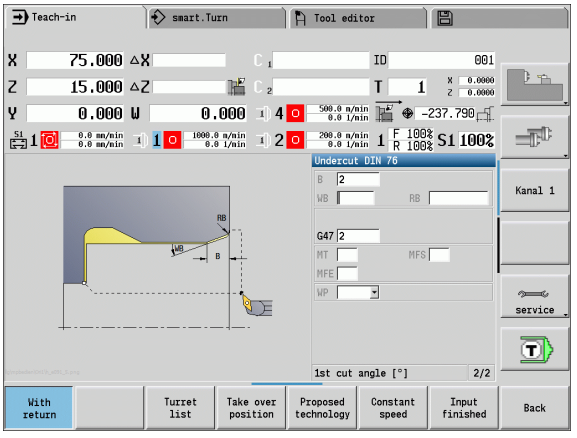
- Threading tool (for external machining)
- WO = 1 (tool orientation)

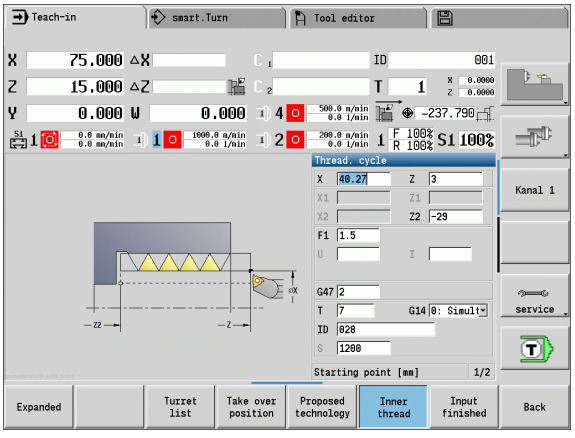


Internal thread and thread undercut









The machining operation is to be performed in two steps. The **thread undercut DIN 76** produces the undercut and thread chamfer. In the second step, the **thread cycle** cuts the thread.

First step

The parameters for the undercut and thread chamfer are programmed in two superimposed input windows.

The CNC PILOT determines the undercut parameters from the standard table.

For the thread chamfer, you only need to enter the chamfer width. The angle of 45° is the default value for the **1st cut angle WB.**

Tool data

- Turning tool (for internal machining)
- WO = 7 (tool orientation)
- A = 93° (tool angle)
- B = 55° (point angle)

Second step

The **Thread cycle (longitudinal)** cuts the thread. The thread pitch is defined. The CNC PILOT automatically determines all other values from the standard table.

You must pay attention to the setting of the **Inner thread** soft key.

Tool data

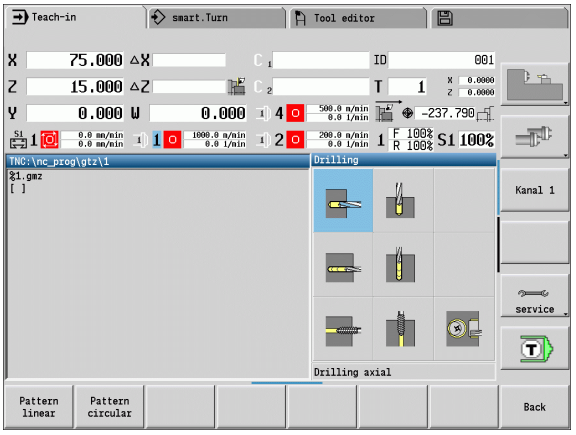
- Threading tool (for internal machining)
- WO = 7 (tool orientation)

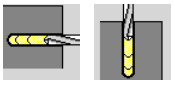
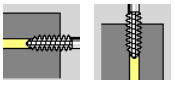
4.7 Drilling cycles



The drilling cycles allow you to machine axial and radial holes.

For pattern machining, see "Drilling and milling patterns" on page 336.



Drilling cycles	Symbol
Axial/radial drilling cycle For drilling single holes and patterns	
Axial/radial deep-drilling cycle For drilling single holes and patterns	
Axial/radial tapping cycle For drilling single holes and patterns	
Thread milling For milling threads in an existing hole	

Drilling, axial



Select **drilling**

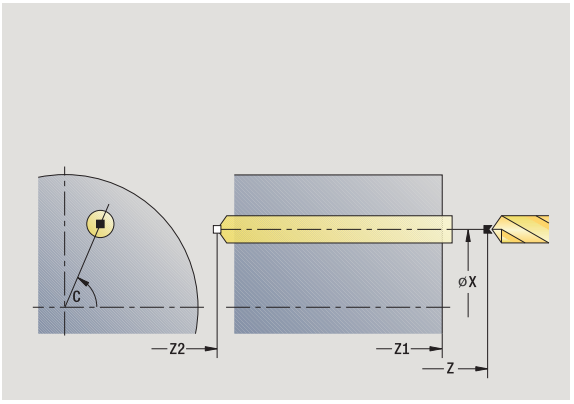


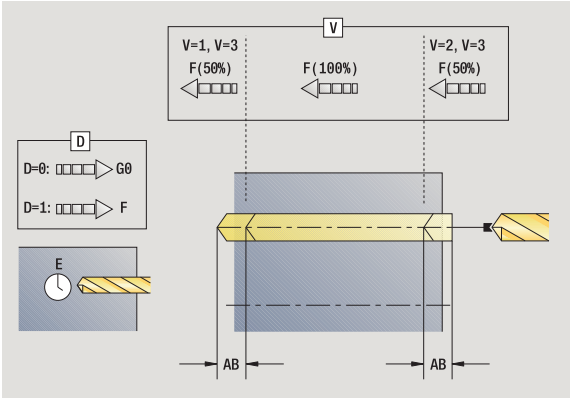
Select **drilling, axial**

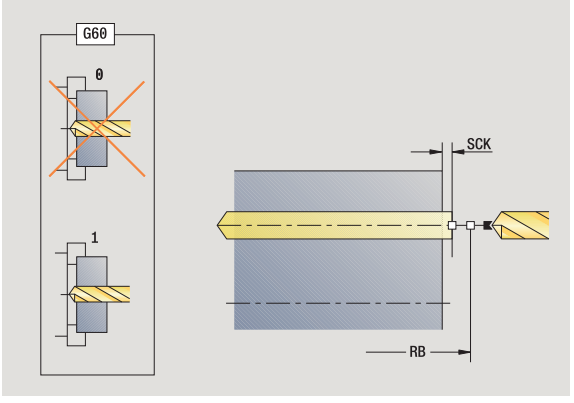
This cycle drills a hole on the face of the workpiece.

Cycle parameters

X, Z	Starting point
C	Spindle angle (C-axis position)
Z1	Starting point of hole (default: drilling starts from Z)
Z2	End point drill (end point of hole)
E	Dwell time for chip breaking at end of hole (default: 0)
D	Retraction type ■ 0: Rapid traverse ■ 1: Feed rate
AB	Drilling lengths (default: 0)
V	Through drilling variants (default: 0) ■ 0: No feed rate reduction ■ 1: Feed rate reduction at the end of the hole ■ 2: Feed rate reduction at the beginning of the hole ■ 3: Feed rate reduction at the beginning and end of the hole
SCK	Safety clearance (see page 128)
G60	Deactivate the protective zone for the drilling operation ■ 0: Active ■ 1: Inactive
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
BP	Break duration: Time span for interruption of the feed. The chip is broken by the (intermittent) interruption of the feed.
BF	Feed duration: Time interval until the next break. The chip is broken by the (intermittent) interruption of the feed.
MT	M after T: M function that is executed after the tool call T







MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Operating mode for technology database access depends on the tool type:

- Twist drill: **Boring and drilling**
- Boring bar with indexable inserts: **Pilot drilling**



- If "AB" **and** "V" are programmed, the feed rate is reduced by 50 % during both pre-drilling and through-boring.
- The CNC PILOT uses the tool parameter **driven tool** to determine whether the programmed spindle speed and feed rate apply to the spindle or the driven tool.

Cycle run

1. Position spindle to **spindle angle C** (in Manual mode, machining starts from the current spindle angle)
2. If defined, move at rapid traverse to **starting point of hole Z1**
3. If defined, spot drill at reduced feed rate
4. Depending on **spot and through drilling variant V**:
   - Through-drilling reduction:
     – Drill at programmed feed rate to position **Z2** – **AB**
     – Drill at reduced feed rate to **hole end point Z2**
   - No through drilling reduction:
     – Drill at programmed feed rate to **hole end point Z2**
     – Remain at end of hole for **dwell time E,** if defined
5. Retract
   - If **Z1** is programmed, to **hole starting point Z1**
   - If **Z1** is **not** programmed, to **starting point Z**
6. Move to the tool change point according to the G14 setting

Radial drilling



Select **drilling**

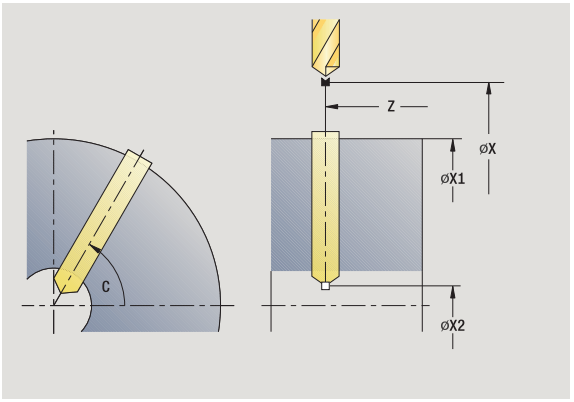


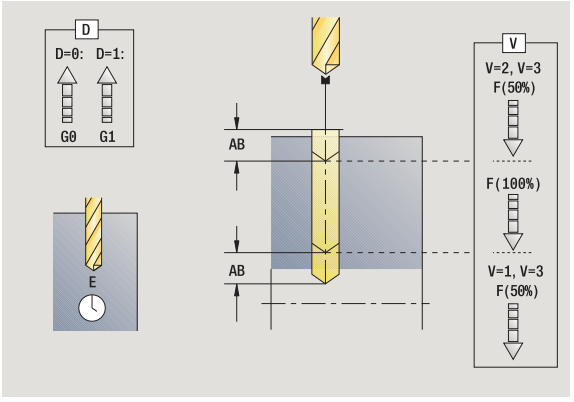
Select **drilling, radial**

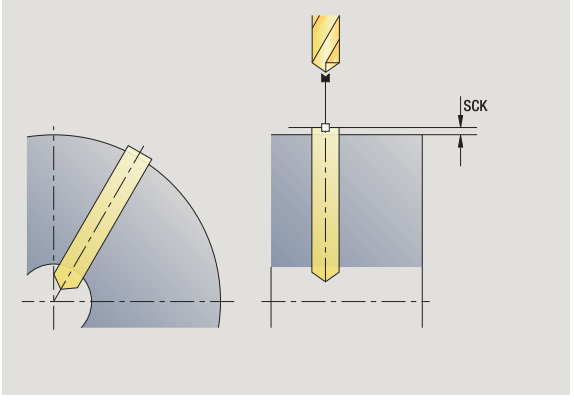
This cycle drills a hole on the lateral surface of the workpiece.

Cycle parameters

X, Z	Starting point
C	Spindle angle (C-axis position)
X1	Starting point of hole (default: drilling starts from X)
X2	End point drill (end point of hole)
E	Dwell time for chip breaking at end of hole (default: 0)
D	Retraction type ■ 0: Rapid traverse ■ 1: Feed rate
AB	Drilling lengths (default: 0)
V	Through drilling variants (default: 0) ■ 0: No feed rate reduction ■ 1: Feed rate reduction at the end of the hole ■ 2: Feed rate reduction at the beginning of the hole ■ 3: Feed rate reduction at the beginning and end of the hole
SCK	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
BP	Break duration: Time span for interruption of the feed. The chip is broken by the (intermittent) interruption of the feed.
BF	Feed duration: Time interval until the next break. The chip is broken by the (intermittent) interruption of the feed.
MT	M after T: M function that is executed after the tool call T







MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Operating mode for technology database access depends on the tool type:

- Twist drill: **Boring and drilling**
- Boring bar with indexable inserts: **Pilot drilling**



If "AB" **and** "V" are programmed, the feed rate is reduced by 50 % during both pre-drilling and through-boring.

Cycle run

1. Position spindle to **spindle angle C** (in Manual mode, machining starts from the current spindle angle)
2. If defined, move at rapid traverse to **hole starting point X1**
3. If defined, spot drill at reduced feed rate
4. Depending on **spot and through drilling variant V**:
   - Through-drilling reduction:
     – Drill at programmed feed rate to position **X2** – **AB**
     – Drill at reduced feed rate to **hole end point X2**
   - No through drilling reduction:
     – Drill at programmed feed rate to **hole end point X2**
     – Remain at end of hole for **dwell time E,** if defined
5. Retract
   - If **X1** is programmed, to **hole starting point X1**
   - If **X1** is **not** programmed, to **starting point X**
6. Move to the tool change point according to the G14 setting

Deep-hole drilling, axial

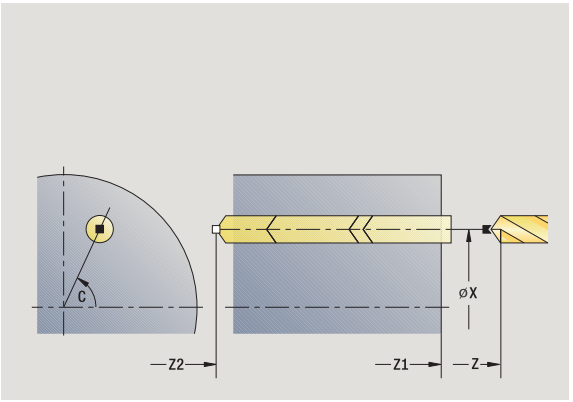


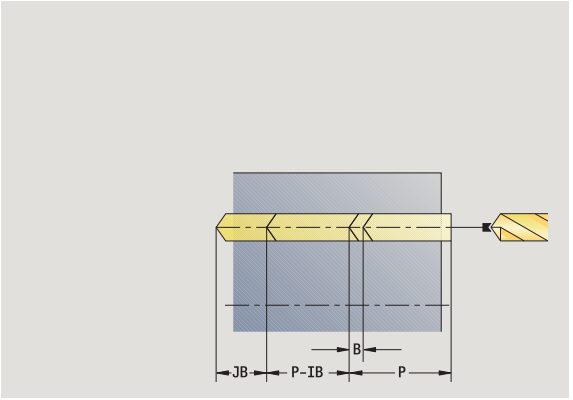
Select **drilling**

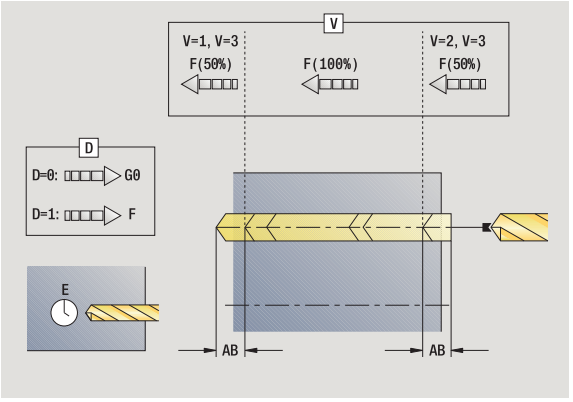


Select **deep-hole drilling, axial**

The cycle produces a bore hole on the face in several passes. After each pass, the drill retracts and, after a dwell time, advances again to the first pecking depth, minus the safety clearance. You define the first pass with **1st hole depth P.** The drilling depth is reduced with each subsequent pass by the **reducing value,** however, without falling below the **minimum drilling depth.**







Cycle parameters

X, Z	Starting point
C	Spindle angle (C-axis position)
Z1	Starting point of hole (default: drilling starts from Z)
Z2	End point drill (end point of hole)
P	1st hole depth (default: hole will be drilled in one pass)
IB	Hole depth reduction value (default: 0)
JB	Minimum hole depth (default: 1/10 of P)
B	Retraction length (default: retract to starting point of hole)
E	Dwell time for chip breaking at end of hole (default: 0)
D	Retraction—retraction speed and infeed within the hole (default: 0)

- 0: Rapid traverse
- 1: Feed rate

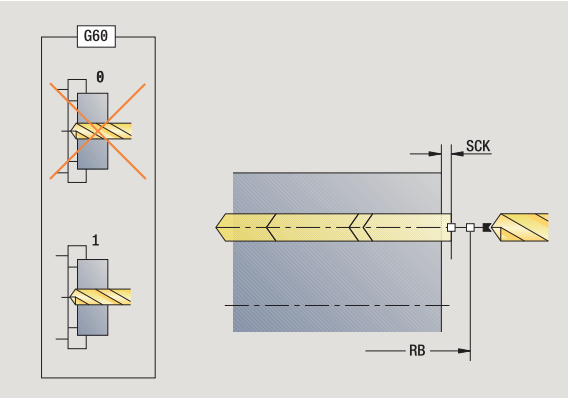
AB	Drilling lengths (default: 0)
V	Through drilling variants (default: 0)

- 0: No feed rate reduction
- 1: Feed rate reduction at the end of the hole
- 2: Feed rate reduction at the beginning of the hole
- 3: Feed rate reduction at the beginning and end of the hole

G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
SCK	Safety clearance (see page 128)
G60	Deactivate the protective zone for the drilling operation

- 0: Active
- 1: Inactive

BP	Break duration: Time span for interruption of the feed. The chip is broken by the (intermittent) interruption of the feed.
BF	Feed duration: Time interval until the next break. The chip is broken by the (intermittent) interruption of the feed.
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent) ■ Main drive ■ Opposing spindle for rear-face machining



Operating mode for technology database access depends on the tool type:

- Twist drill: **Boring and drilling**
- Boring bar with indexable inserts: **Pilot drilling**

- If "AB" **and** "V" are programmed, the feed rate is reduced by 50 % during both pre-drilling and through-boring.
- The CNC PILOT uses the tool parameter **`driven tool`** to determine whether the programmed spindle speed and feed rate apply to the spindle or the driven tool.

Cycle run

1. Position spindle to **spindle angle C** (in Manual mode, machining starts from the current spindle angle)
2. If defined, move at rapid traverse to **starting point of hole Z1**
3. First pass (pecking depth: P)—drill with reduced feed rate, if defined
4. Retract by **retraction length B**—or to the **hole starting point** and advance again to last pecking depth minus safety clearance
5. Next pass (pecking depth: "last depth – IB" or JB).
6. Repeat 4 to 5 until **hole end point Z2** is reached
7. Last drilling pass—depending on **spot and through-drilling variant V**:
   - Through-drilling reduction:
     – Drill at programmed feed rate to position **Z2** – **AB**
     – Drill at reduced feed rate to **hole end point Z2**
   - No through drilling reduction:
     – Drill at programmed feed rate to **hole end point Z2**
     – Remain at end of hole for **dwell time E,** if defined
8. Retract
   - If **Z1** is programmed, to **hole starting point Z1**
   - If **Z1** is **not** programmed, to **starting point Z**
9. Move to the tool change point according to the G14 setting

Deep-hole drilling, radial

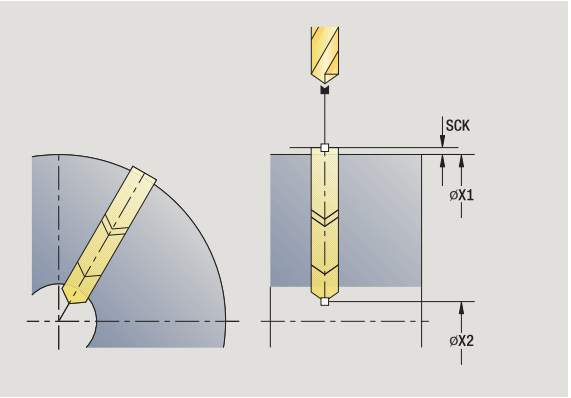


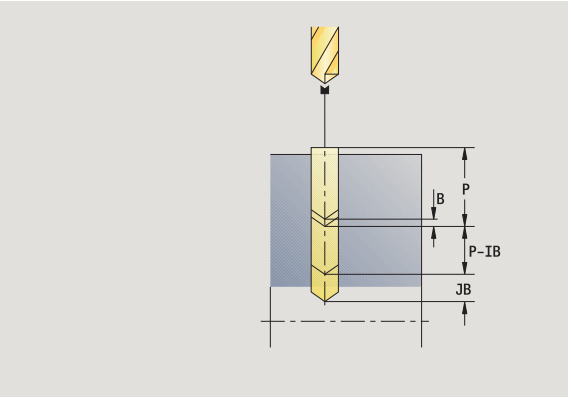
Select **drilling**

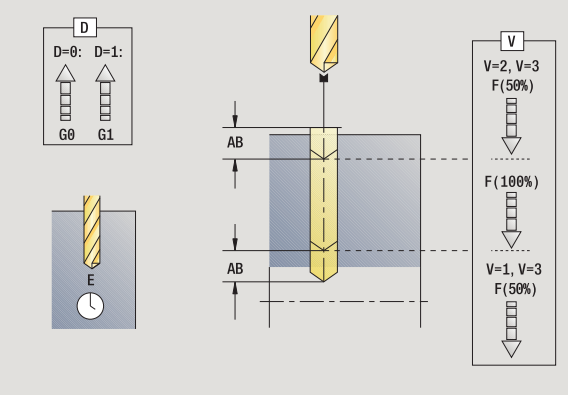


Select **deep hole, radial**

The cycle produces a bore hole on the lateral surface in several passes. After each pass, the drill retracts and, after a dwell time, advances again to the first pecking depth, minus the safety clearance. You define the first pass with **1st hole depth P.** The drilling depth is reduced with each subsequent pass by the **reducing value,** however, without falling below the **minimum drilling depth.**







Cycle parameters

X, Z	Starting point
C	Spindle angle (C-axis position)
X1	Starting point of hole (default: drilling starts from X)
X2	End point drill (end point of hole)
P	1st hole depth (default: hole will be drilled in one pass)
IB	Hole depth reduction value (default: 0)
JB	Minimum hole depth (default: 1/10 of P)
B	Retraction length (default: retract to starting point of hole)
E	Dwell time for chip breaking at end of hole (default: 0)
D	Retraction—retraction speed and infeed within the hole (default: 0) ■ 0: Rapid traverse ■ 1: Feed rate
AB	Drilling lengths (default: 0)
V	Through drilling variants (default: 0) ■ 0: No feed rate reduction ■ 1: Feed rate reduction at the end of the hole ■ 2: Feed rate reduction at the beginning of the hole ■ 3: Feed rate reduction at the beginning and end of the hole
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
SCK	Safety clearance (see page 128)
BP	Break duration: Time span for interruption of the feed. The chip is broken by the (intermittent) interruption of the feed.
BF	Feed duration: Time interval until the next break. The chip is broken by the (intermittent) interruption of the feed.

MT M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Operating mode for technology database access depends on the tool type:

- Twist drill: **Boring and drilling**
- Boring bar with indexable inserts: **Pilot drilling**



If "AB" **and** "V" are programmed, the feed rate is reduced by 50 % during both pre-drilling and through-boring.

Cycle run

1. Position spindle to **spindle angle C** (in Manual mode, machining starts from the current spindle angle)
2. If defined, move at rapid traverse to **hole starting point X1**
3. First pass (pecking depth: P)—drill with reduced feed rate, if defined
4. Retract by **retraction length B**—or to the **hole starting point** and advance again to last pecking depth minus safety clearance
5. Next pass (pecking depth: "last depth – IB" or JB).
6. Repeat 4 to 5 until **hole end point X2** is reached
7. Last drilling pass—depending on **spot and through-drilling variant V**:
   - Through-drilling reduction:
     – Drill at programmed feed rate to position **X2** – **AB**
     – Drill at reduced feed rate to **hole end point X2**
   - No through drilling reduction:
     – Drill at programmed feed rate to **hole end point X2**
     – Remain at end of hole for **dwell time E,** if defined
8. Retract
   - If **X1** is programmed, to **hole starting point X1**
   - If **X1** is **not** programmed, to **starting point X**
9. Move to the tool change point according to the G14 setting

Tapping, axial



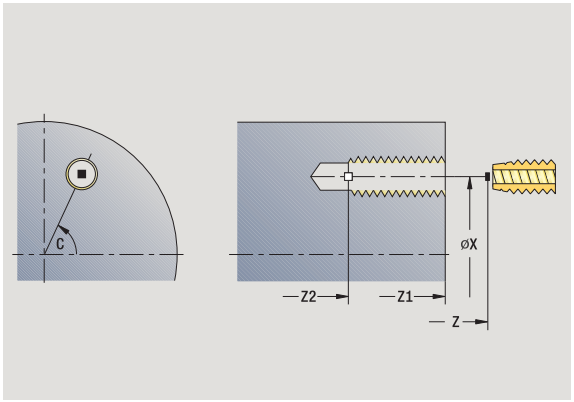
Select **drilling**

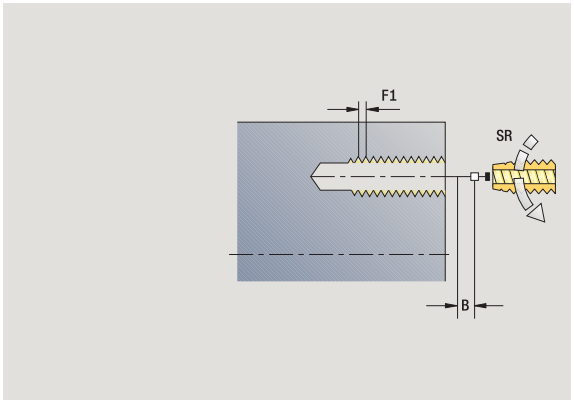


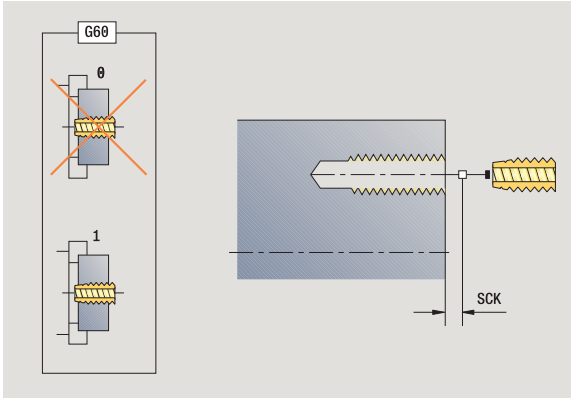
Select **tapping, axial**

This cycle is used to tap a thread on the face of a workpiece.

Meaning of **retraction length:** Use this parameter for floating tap holders. The cycle calculates a new nominal pitch on the basis of the thread depth, the programmed pitch, and the retraction length. The nominal pitch is somewhat smaller than the pitch of the tap. During tapping, the drill is pulled away from the chuck by the retraction length. With this method you can achieve higher service life from the taps.







Cycle parameters

X, Z	Starting point
C	Spindle angle (C-axis position)—(default: current spindle position)
Z1	Starting point of hole (default: drilling starts from Z)
Z2	End point drill (end point of hole)
F1	Thread pitch (= feed rate) (default: feed rate from tool definition)
B	Run-in length to reach the programmed spindle speed and feed rate (default: 2 * thread pitch F1)
SR	Return speed for enabling rapid retraction (default: same spindle speed as for tapping)
L	Retraction length when using floating tap holders (default: 0)
SCK	Safety clearance (see page 128)
G60	Deactivate the protective zone for the drilling operation ■ 0: Active ■ 1: Inactive
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **tapping**



The CNC PILOT uses the tool parameter **driven tool** to determine whether the programmed spindle speed and feed rate apply to the spindle or the driven tool.

Cycle run

1. Position spindle to **spindle angle C** (in Manual mode, machining starts from the current spindle angle)
2. If defined, move at rapid traverse to **starting point of hole Z1**
3. Tap thread to **hole end point Z2**
4. Retract at **return speed SR**
   - If **Z1** is programmed, to **hole starting point Z1**
   - If **Z1** is **not** programmed, to **starting point Z**
5. Move to the tool change point according to the G14 setting

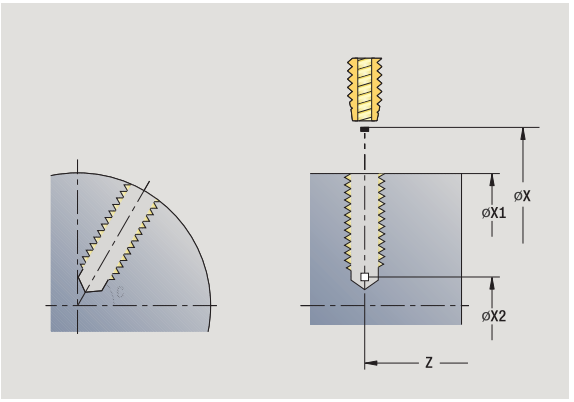
Tapping, radial

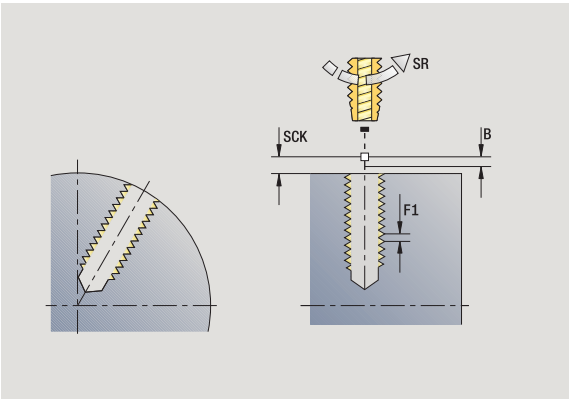


Select **drilling**



Select **tapping, radial**





This cycle is used to tap a thread on the lateral surface of a workpiece.

Meaning of **retraction length:** Use this parameter for floating tap holders. The cycle calculates a new nominal pitch on the basis of the thread depth, the programmed pitch, and the retraction length. The nominal pitch is somewhat smaller than the pitch of the tap. During tapping, the drill is pulled away from the chuck by the retraction length. With this method you can achieve higher service life from the taps.

Cycle parameters

X, Z	Starting point
C	Spindle angle (C-axis position)—(default: current spindle position)
X1	Starting point of hole (default: drilling starts from X)
X2	End point drill (end point of hole)
F1	Thread pitch (= feed rate) (default: feed rate from tool definition)
B	Run-in length to reach the programmed spindle speed and feed rate (default: 2 * thread pitch F1)
SR	Return speed for enabling rapid retraction (default: same spindle speed as for tapping)
L	Retraction length when using floating tap holders (default: 0)
SCK	Safety clearance (see page 128)
G60	Protection zone—deactivates the protection zone for the drilling process ■ 0: Active ■ 1: Inactive
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
MT	M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **tapping**

Cycle run

1. Position spindle to **spindle angle C** (in Manual mode, machining starts from the current spindle angle)
2. If defined, move at rapid traverse to **hole starting point X1**
3. Tap thread to **hole end point X2**
4. Retract at **return speed SR**
   - If **X1** is programmed, to **hole starting point X1**
   - If **X1** is **not** programmed, to **starting point X**
5. Move to the tool change point according to the G14 setting

Thread milling, axial



Select **drilling**



Select **thread milling, axial**

The cycle mills a thread in existing holes.

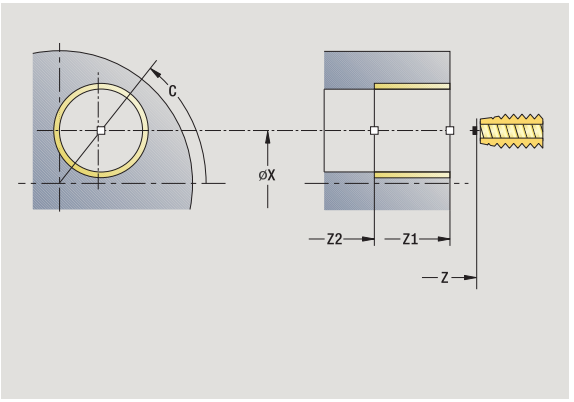


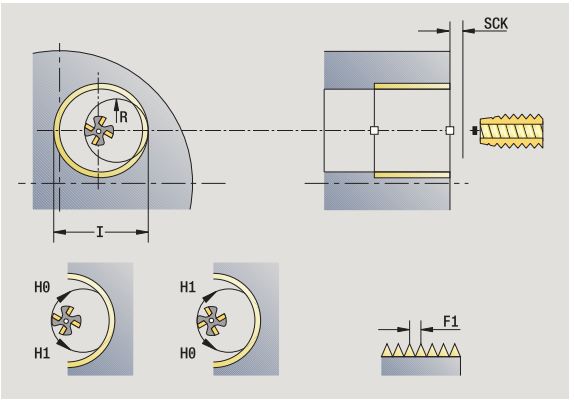
Use threading tools for this cycle.



Danger of collision!

Be sure to consider the hole diameter and the diameter of the milling cutter when programming **approaching radius R.**





Cycle parameters

X, Z	Starting point
C	Spindle angle (C-axis position)—(default: current spindle position)
Z1	Starting point of thread (default: drilling starts from Z)
Z2	End point of thread
F1	Thread pitch (= feed rate)
J	Direction of thread ■ 0: Right ■ 1: Left
I	Thread diameter
R	Approaching radius [default: (I - milling diameter)/2]
H	Cutting direction ■ 0: Up-cut milling ■ 1: Climb milling
V	Milling method ■ 0: The thread is milled in a 360-degree helix ■ 1: The thread is milled in several helical paths (single-point tool)
SCK	Safety clearance (see page 128)

G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent) ■ Main drive ■ Opposing spindle for rear-face machining

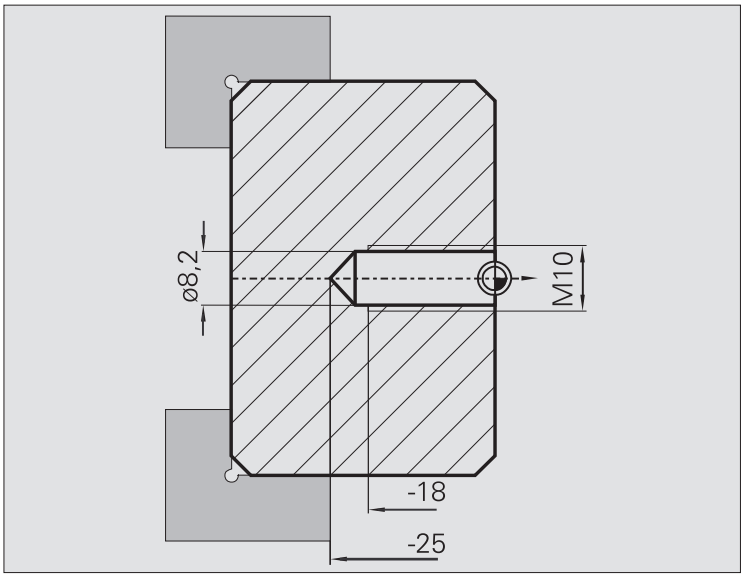
Type of machining for technology database access: **Milling**

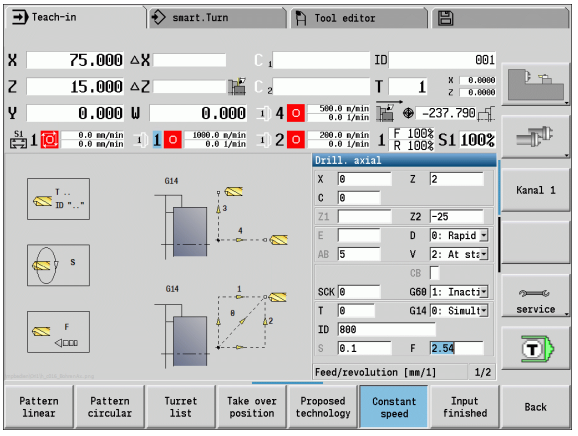
Cycle run

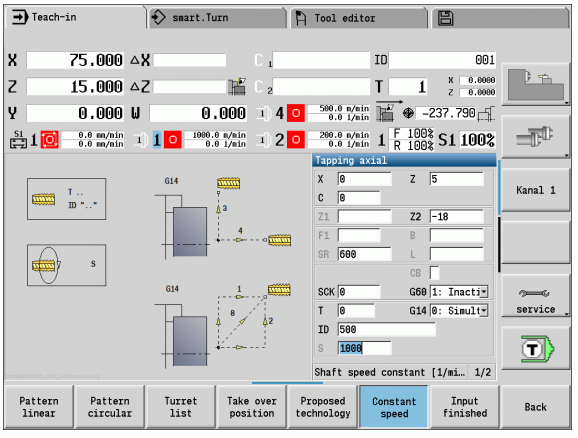
1. Position spindle to **`spindle angle C`** (in Manual mode, machining starts from the current spindle angle)
2. Position the tool to **`thread end point Z2`** (milling floor) inside the hole
3. Approach on **`approach radius R`**
4. Mill the thread in a rotation of 360°, while advancing by the **`thread pitch F1`**
5. Retract the tool and return it to the starting point
6. Move to the tool change point according to the G14 setting

Examples of drilling cycles

Centric drilling and tapping







The machining operation is to be performed in two steps. In the first step, the **Drilling, axial** cycle drills the hole. In the second, the **Tapping, axial** cycle taps the thread.

The drill is positioned at the safety clearance to the workpiece surface (**starting point X, Z**). The hole **starting point Z1** is therefore not programmed. In the parameters "AB" and "V," you program a feed reduction.

The thread pitch is not programmed. The CNC PILOT uses the thread pitch of the tool. The **return speed SR** ensures that the tool is retracted quickly.

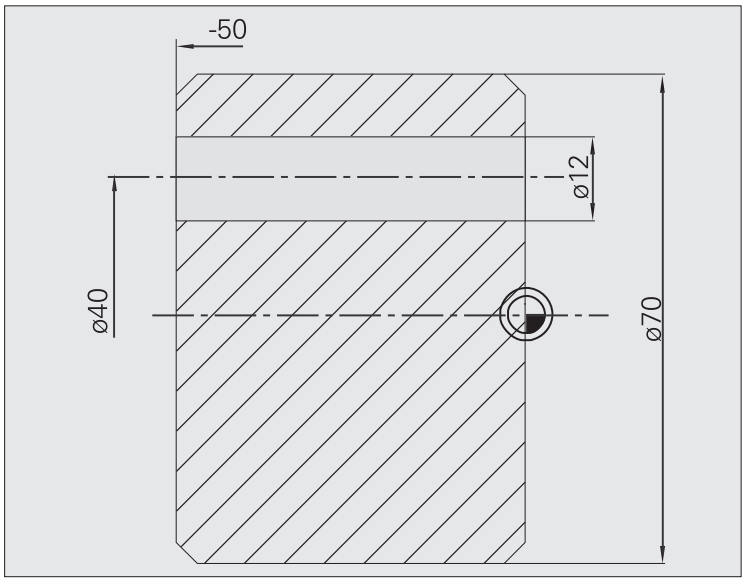
Tool data (drill)

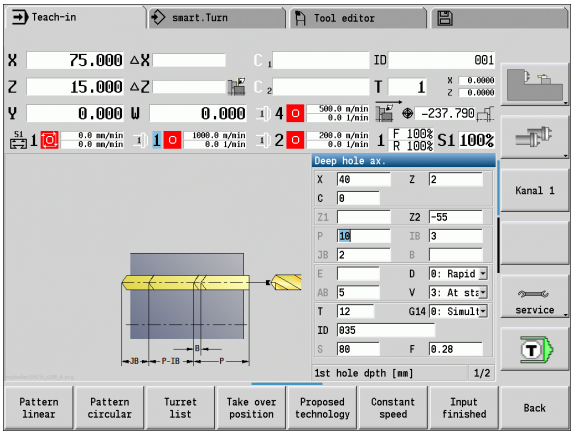
- WO = 8 (tool orientation)
- I = 8.2 (drilling diameter)
- B = 118 (point angle)
- H = 0 (the tool is not a driven tool)

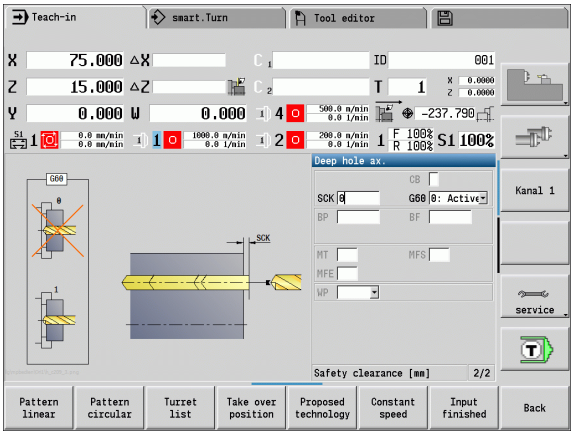
Tool data (tap)

- WO = 8 (tool orientation)
- I = 10 (thread diameter M10)
- F = 1.5 (thread pitch)
- H = 0 (the tool is not a driven tool)

Deep-hole drilling







A hole is to be bored through the workpiece outside the turning center with the cycle **Deep-hole drilling, axial.** This machining operation requires a traversable spindle and driven tools.

The parameters **1st hole depth P** and **hole depth reduction value IB** define the individual passes, and the **minimum hole depth JB** limits the hole reduction value.

As the **return length B** is not defined, the drill therefore retracts to the starting point after each pass, remains there for the programmed dwell time, and then advances again to the safety clearance for the next pass.

Since this example is to illustrate how you drill a through hole, the **hole end point Z2** is programmed such that the tool has to drill all the way through the workpiece before it reaches the end point.

The parameters "AB" and "V" define a feed reduction for both pre-drilling and through-boring.

Tool data

- WO = 8 (tool orientation)
- I = 12 (drilling diameter)
- B = 118 (point angle)
- H = 1 (the tool is a driven tool)

4.8 Milling cycles

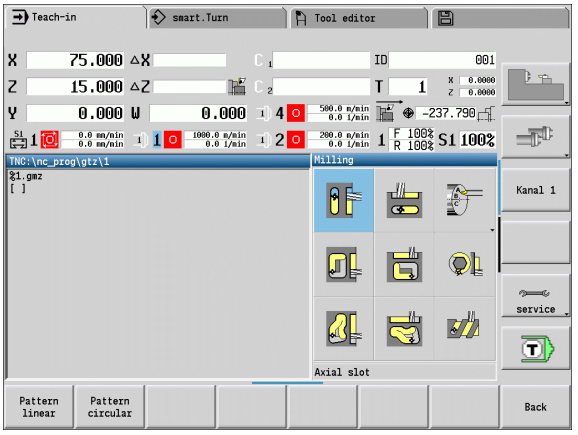


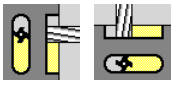
Milling cycles are used to machine axial and radial slots, contours, pockets, surfaces and polygons.

For pattern machining, see "Drilling and milling patterns" on page 336.

In **Teach-in mode** these cycles include the activation/deactivation of the C axis and the positioning of the spindle.

In **Manual mode** you can activate the C axis with **`Rapid traverse positioning`** and position the spindle **before** the actual milling cycle. The milling cycles then automatically deactivate the C axis.



Milling cycles	Symbol
Rapid traverse positioning Activate C axis; position tool and spindle	
Slot axial/radial For milling single slots or slot patterns	
Figure axial/radial For milling a single figure	
Radial/axial ICP contour For milling single ICP contours or contour patterns	
Face milling For milling surfaces or polygons	
Helical-slot milling, radial For milling a helical slot	
Engraving, axial/radial Engraves characters and character strings	

Rapid positioning milling

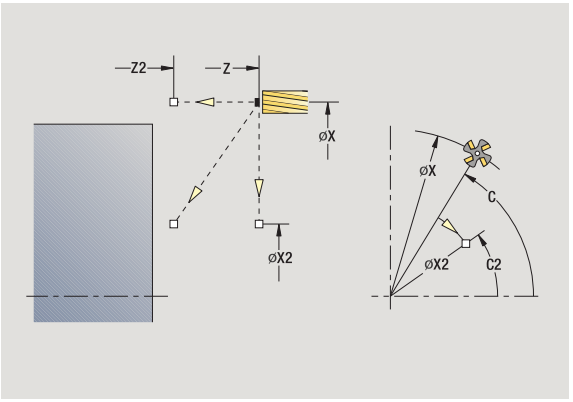


Select **milling**



Select **rapid traverse positioning**

The cycle activates the C axis and positions the spindle (C axis) and the tool.



- **Rapid traverse positioning** is only required in **Manual** mode.
- The C axis is deactivated by a subsequent manual milling cycle.

Cycle parameters

X2, Z2	Target point
C2	End angle (C-axis position)—(default: current spindle position)
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent) ■ Main drive ■ Opposing spindle for rear-face machining

Cycle run

1. Activate C axis
2. Insert the current tool
3. Position the tool at rapid traverse to the **target point X2, Z2** and the **end angle C2**

Slot, axial



Select **milling**



Select **slot, axial**

This cycle mills a slot on the face of the workpiece. The slot width equals the diameter of the milling cutter.

Cycle parameters

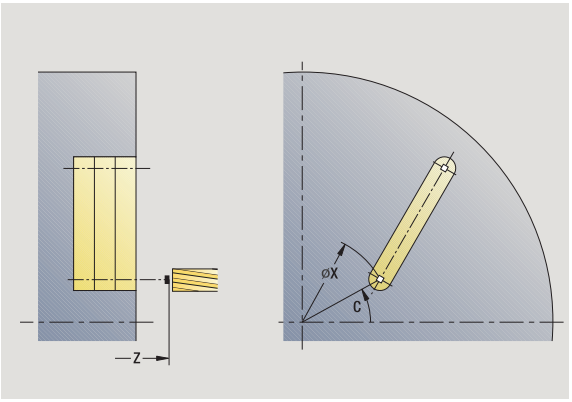
X, Z	Starting point
C	Spindle angle (C-axis position)
X1	Slot target point in X (diameter value)
C1	Angle of slot target point (default: spindle angle C)
L	Slot length
A1	Angle to X axis (default: 0)
Z1	Milling top edge (default: starting point Z)
Z2	Milling floor
P	Infeed depth (default: total depth of one infeed)
FZ	Approach feed (infeed rate) (default: active feed rate)
SCK	Safety clearance (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent)

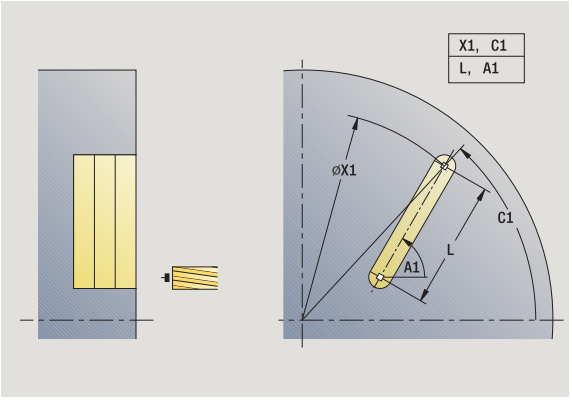
- Main drive
- Opposing spindle for rear-face machining

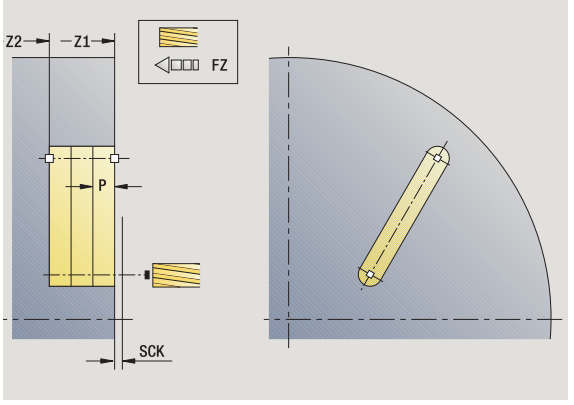
Type of machining for technology database access: **Milling**

Parameter combinations for the position and orientation of the slot:

- X1, C1
- L, A1







Cycle run

1. Activate the C axis and position to **`spindle angle C`** at rapid traverse (only in **Teach-in mode**)
2. Calculate the proportioning of cuts
3. Approach at **`infeed rate FZ`**
4. Machine to end point of slot
5. Approach at **`infeed rate FZ`**
6. Machine to starting point of slot
7. Repeat 3 to 6 until the milling depth is reached
8. Position to **`starting point Z`** and deactivate C axis
9. Move to the tool change point according to the G14 setting

Figure, axial



Select **milling**



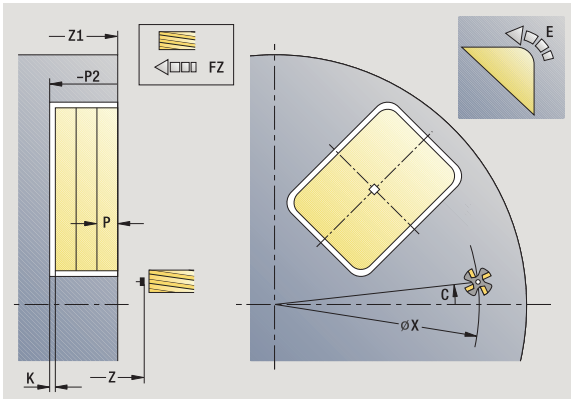
Select **figure, axial**

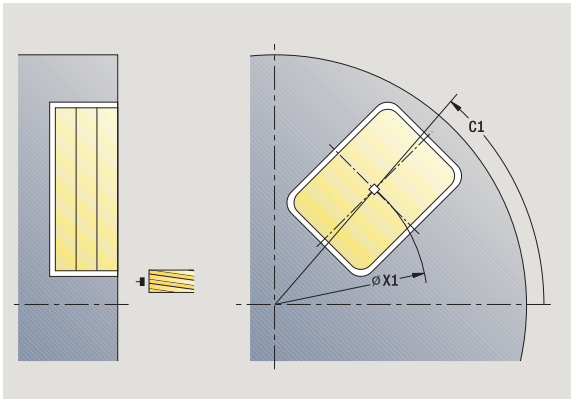
Depending on the parameters, the cycle mills one of the following contours or roughs/finishes a pocket on the face:

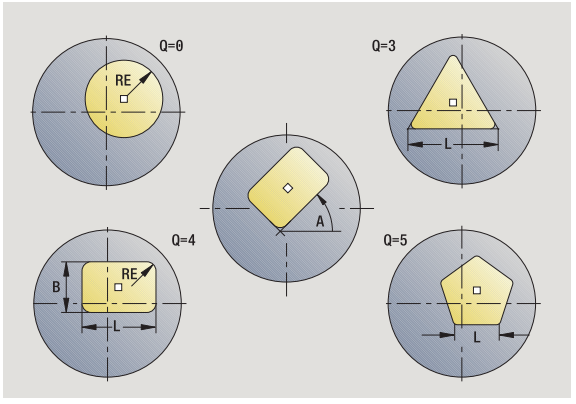
- Rectangle (Q=4, L<>B)
- Square (Q=4, L=B)
- Circle (Q=0, RE>0, L and B: No input)
- Triangle or polygon (Q=3 or Q>4, L<>0)

Cycle parameters (first input window)

X, Z	Starting point
C	Spindle angle (C-axis position)—(default: current spindle position)
X1	Diameter of figure center
C1	Angle of figure center (default: Spindle angle C)
Q	Number of edges (default: 0)
	■ Q=0: Circle ■ Q=4: Rectangle, square ■ Q=3: Triangle ■ Q>4: Polygon
L	Edge length
	■ Rectangle: Length of rectangle ■ Square, polygon: Edge length ■ Polygon: L<0 inside circle diameter ■ Circle: No input
B	Rectangle width
	■ Rectangle: Width of rectangle ■ Square: L=B ■ Polygon, circle: No input
RE	Rounding radius (default: 0)
	■ Rectangle, square, polygon: Rounding radius ■ Circle: Radius of circle
A	Angle to X axis (default: 0)
	■ Rectangle, square, polygon: Position of figure ■ Circle: No input
Z1	Milling top edge (default: starting point Z)
P2	Milling depth
G14	Tool change point (see page 128)







T Turret pocket number
ID Tool ID number
S Spindle speed/cutting speed
F Feed per revolution

Cycle parameters (second input window)

I Contour-parallel oversize
K Infeed-direction oversize
P Infeed depth (default: total depth of one infeed)
FZ Approach feed (infeed rate) (default: active feed rate)
E Reduced feed rate for circular elements (default: active feed rate)
O Roughing or finishing—only for pocket milling

- 0: Roughing
- 1: Finishing

H Cutting direction

- 0: Up-cut milling
- 1: Climb milling

U Overlap factor (range: 0 < U < 1)

- U=0 or no input: Contour milling
- U>0: Pocket milling—minimum overlap of milling paths = U * milling diameter

JK Contour milling (input is evaluated only for contour milling)

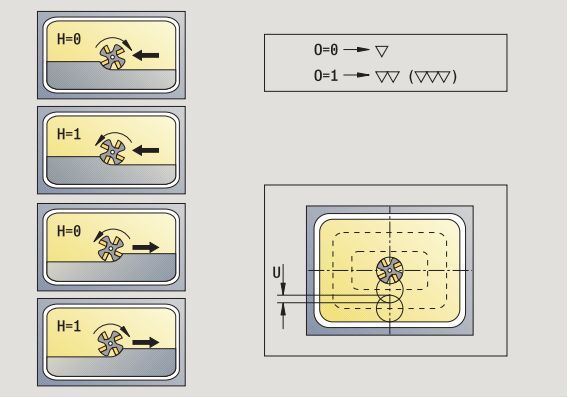
- 0: On the contour
- 1: Within the contour
- 2: Outside the contour

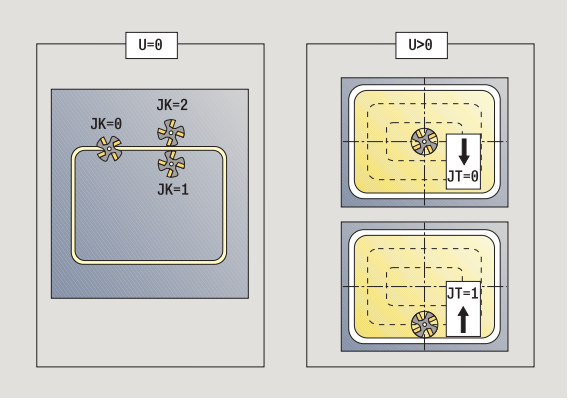
JT Pocket milling (input is evaluated only for pocket milling)

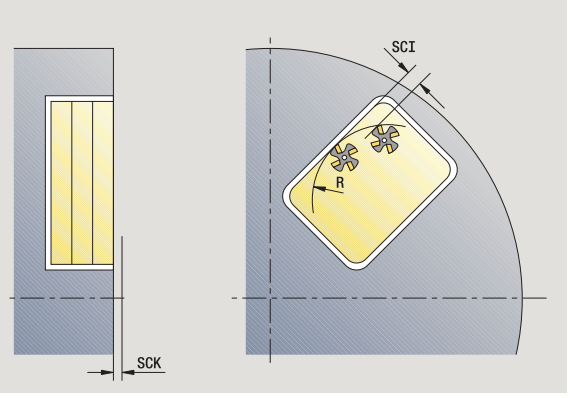
- 0: From the inside out (from the inside towards the outside)
- 1: From the outside in (from the outside towards the inside)

R Approach radius (default: 0)

- R=0: Contour element is approached directly; infeed to starting point above the milling plane—then vertical plunge
- R>0: Tool moves on approaching/departing arc that connects tangentially to the contour element
- R<0 for inside corners: Tool moves on approaching/departing arc that connects tangentially to the contour element
- R<0 for outside corners: Length of linear approaching/departing element; contour element is approached/departed tangentially







SCI Safety clearance in the working plane
SCK Safety clearance in infeed direction (see page 128)
MT M after T: M function that is executed after the tool call T
MFS M at beginning: M function that is executed at the beginning of the machining step
MFE M at end: M function that is executed at the end of the machining step

Cycle parameters (third input window)
WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Milling**

Notes on parameters/functions:

- **Machining of contour or pocket:** defined with **`overlap factor U`**.
- **Milling direction:** depends on definition of **`cutting direction H`** and the direction of tool rotation (see "Milling direction for contour milling" on page 328).
- **Milling cutter compensation:** effective (except for contour milling with J=0).
- **Approach and departure:** For closed contours, the starting point of the first element (in rectangles, the longer element) is the point of approach and departure. The tool approaches directly or on an arc according to **`approaching radius R.`**
- **`Contour milling JK`** defines whether the milling cutter is to machine on the contour (center of milling cutter on the contour) or on the inside/outside of the contour.
- **Pocket milling—roughing (O=0):** Use **`JT`** to define whether a pocket is machined from the inside toward the outside, or vice versa.
- **Pocket milling—finishing (O=1):** First, the edge of the pocket is machined; then the pocket floor is machined. With **`JT`** you define whether a pocket floor is to be finished from the inside towards the outside, or vice versa.

Cycle run

1. Activate the C axis and position to **spindle angle C** at rapid traverse (only in **Teach-in mode**)
2. Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling planes)

Contour milling:

3. Depending on the **approaching radius R,** approach the workpiece and plunge to the first milling plane
4. Mill the first plane
5. Plunge to the next milling plane
6. Repeat 5 to 6 until the milling depth is reached

Pocket milling—roughing:

3. Move to the safety clearance and plunge to the first milling plane
4. Depending on **pocket milling JT,** machine the milling plane either from the inside towards the outside, or vice versa
5. Plunge to the next milling plane
6. Repeat 4 to 5 until the milling depth is reached

Pocket milling—finishing:

3. Depending on the **approaching radius R,** approach the workpiece and plunge to the first milling plane
4. Finish-machine the edge of the pocket—one working plane after the other
5. Depending on **pocket milling JT,** finish the pocket floor either from the inside towards the outside, or vice versa
6. Finish-machine the pocket at the programmed feed rate

All versions:

7. Position to **starting point Z** and deactivate C axis
8. Move to the tool change point according to the G14 setting

ICP contour, axial

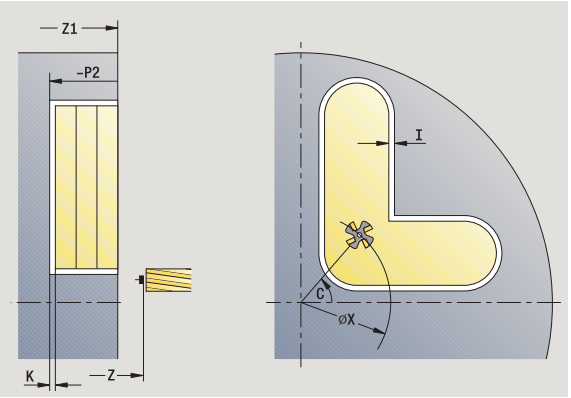


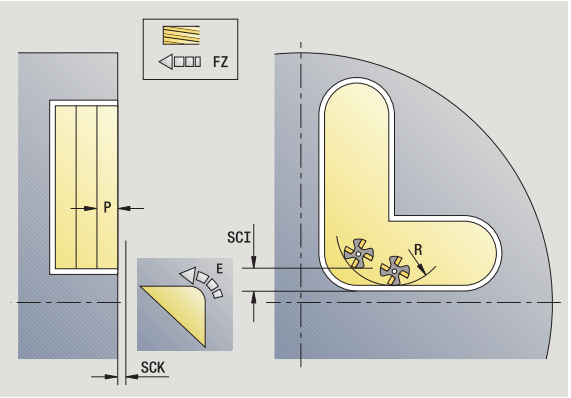
Select **milling**

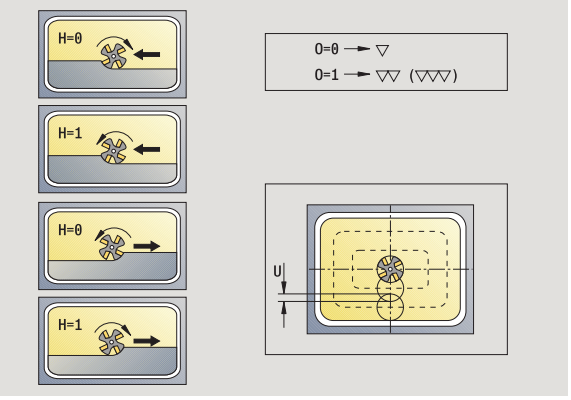


Select **ICP contour, axial**

Depending on the parameters, the cycle mills a contour or roughs/ finishes a pocket on the face.







Cycle parameters (first input window)

X, Z	Starting point
C	Spindle angle (C-axis position)
Z1	Milling top edge (default: starting point Z)
P2	Milling depth
I	Contour-parallel oversize
K	Infeed-direction oversize
P	Infeed depth (default: total depth of one infeed)
FZ	Approach feed (infeed rate) (default: active feed rate)
E	Reduced feed rate for circular elements (default: active feed rate)
FK	ICP contour number
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution

Cycle parameters (second input window)

O — Roughing or finishing—only for pocket milling

- 0: Roughing
- 1: Finishing
- 2: Deburr

H — Cutting direction

- 0: Up-cut milling
- 1: Climb milling

U — Overlap factor (range: 0 < U < 1)

- U=0 or no input: Contour milling
- U>0: Pocket milling—minimum overlap of milling paths = U * milling diameter

JK — Contour milling (input is evaluated only for contour milling)

- 0: On the contour
- 1: Within the contour
- 2: Outside the contour

JT Pocket milling (input is evaluated only for pocket milling)

- 0: From the inside out (from the inside towards the outside)
- 1: From the outside in (from the outside towards the inside)

R Approach radius (default: 0)

- R=0: Contour element is approached directly; infeed to starting point above the milling plane—then vertical plunge
- R>0: Tool moves on approaching/departing arc that connects tangentially to the contour element
- R<0 for inside corners: Tool moves on approaching/departing arc that connects tangentially to the contour element
- R<0 for outside corners: Length of linear approaching/departing element; contour element is approached/departed tangentially

SCI Safety clearance in the working plane

SCK Safety clearance in infeed direction (see page 128)

BG Chamfer width for deburring

JG Preparation diameter

MT M after T: M function that is executed after the tool call T

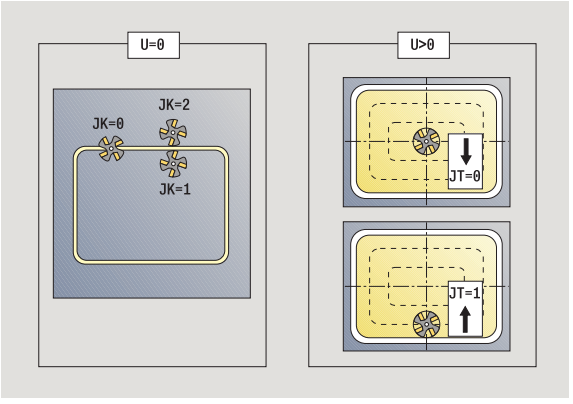
MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Milling**



Notes on parameters/functions:

- **Machining of contour or pocket:** defined with **`overlap factor U`**.
- **Milling direction:** depends on definition of **`cutting direction H`** and the direction of tool rotation (see "Milling direction for contour milling" on page 328).
- **Milling cutter compensation:** effective (except for contour milling with JK=0).
- **Approach and departure:** For closed contours, the starting point of the first element (in rectangles, the longer element) is the point of approach and departure. The tool approaches directly or on an arc according to **`approaching radius R.`**

Notes on parameters/functions:

- **Contour milling JK** defines whether the milling cutter is to machine on the contour (center of milling cutter on the contour) or on the inside/outside of the contour. **Open contours** are machined in direction of contour definition. **JK** defines whether to move to the left or right of the contour.
- **Pocket milling—roughing (O=0):** Use **JT** to define whether a pocket is machined from the inside toward the outside, or vice versa.
- **Pocket milling—finishing (O=1):** First, the edge of the pocket is machined; then the pocket floor is machined. With **JT** you define whether a pocket floor is to be finished from the inside towards the outside, or vice versa.

Cycle run

1 Activate the C axis and position to **spindle angle C** at rapid traverse (only in **Teach-in mode**)

2 Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling planes)

Contour milling:

3 Depending on the **approaching radius R,** approach the workpiece and plunge to the first milling plane

4 Mill the first plane

5 Plunge to the next milling plane

6 Repeat 5 to 6 until the milling depth is reached

Pocket milling—roughing:

3 Move to the safety clearance and plunge to the first milling plane

4 Depending on **pocket milling JT,** machine the milling plane either from the inside towards the outside, or vice versa

5 Plunge to the next milling plane

6 Repeat 4 to 5 until the milling depth is reached

Pocket milling—finishing:

3 Depending on the **approaching radius R,** approach the workpiece and plunge to the first milling plane

4 Finish-machine the edge of the pocket—one working plane after the other

5 Depending on **pocket milling JT,** finish the pocket floor either from the inside towards the outside, or vice versa

6 Finish-machine the pocket at the programmed feed rate

All versions:

7 Position to **starting point Z** and deactivate C axis

8 Move to the tool change point according to the G14 setting

Face milling



Select **milling**



Select the "Face milling" cycle

Depending on the parameters, the cycle mills the following contours on the face.

- One or two surfaces (Q=1 or Q=2, B>0)
- Rectangle (Q=4, L<>B)
- Square (Q=4, L=B)
- Triangle or polygon (Q=3 or Q>4, L<>0)
- Circle (Q=0, RE>0, L and B: No input)

Cycle parameters (first input window)

X, Z	Starting point
C	Spindle angle (C-axis position)
X1	Diameter of figure center
C1	Angle of figure center (default: Spindle angle C)
Z1	Milling top edge (default: starting point Z)
Z2	Milling floor
Q	Number of edges

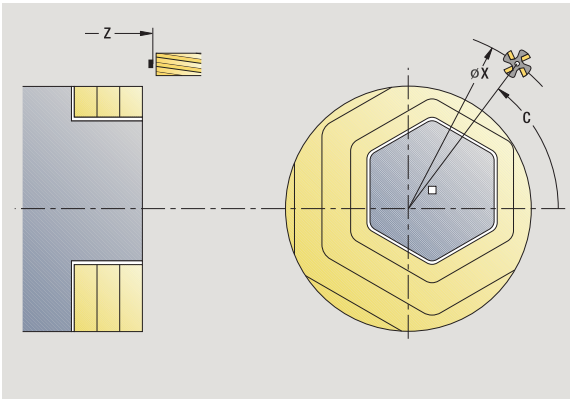
- Q=0: Circle
- Q=1: One surface
- Q=2: Two surfaces offset by 180°
- Q=3: Triangle
- Q=4: Rectangle, square
- Q>4: Polygon

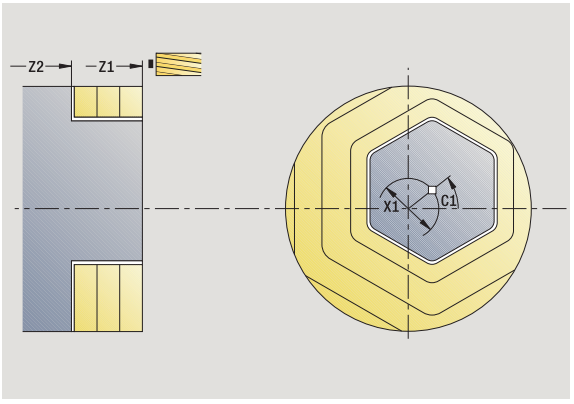
L — Edge length

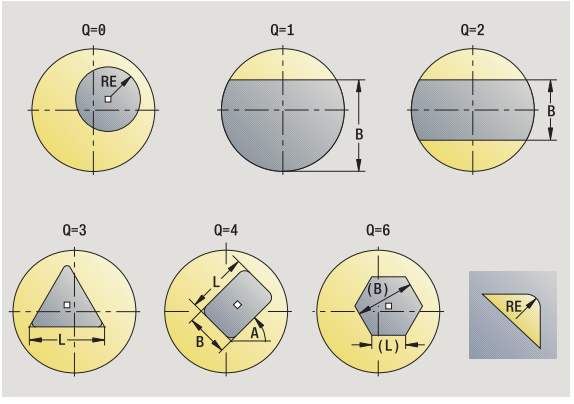
- Rectangle: Length of rectangle
- Square, polygon: Edge length
- Polygon: L<0: Inside circle diameter
- Circle: No input

B — Width across flats:

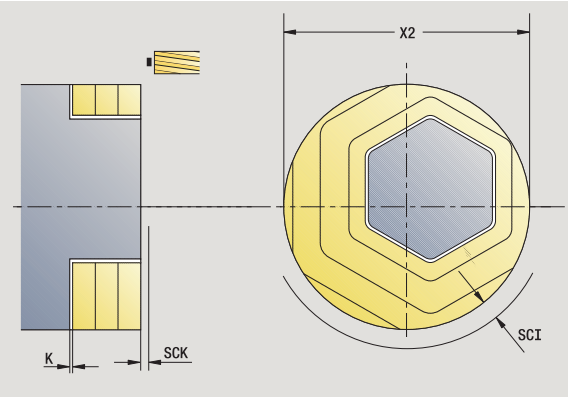
- For Q=1, Q=2: Remaining thickness (remaining material)
- Rectangle: Width of rectangle
- Square, polygon (Q>=4): Width across flats (use only for even number of surfaces; program "B" as an alternative to "L")
- Circle: No input





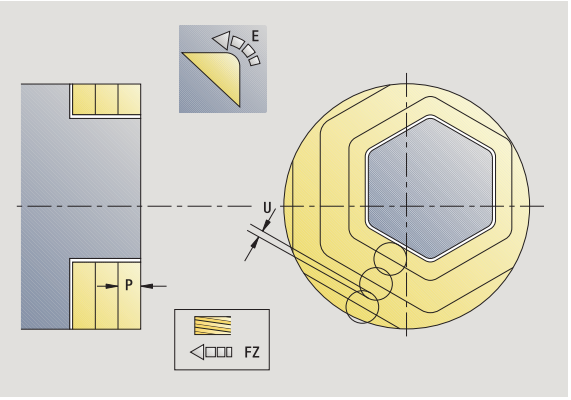


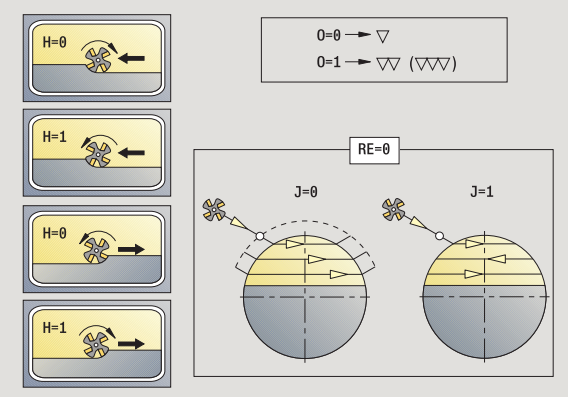
RE	Rounding radius (default: 0) ■ Polygon (Q>2): Rounding radius ■ Circle (Q=0): Radius of circle
A	Angle to X axis (default: 0) ■ Polygon (Q>2): Position of figure ■ Circle: No input
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution



Cycle parameters (second input window)

I	Contour-parallel oversize
K	Infeed-direction oversize
X2	Limit diameter
P	Infeed depth (default: total depth of one infeed)
FZ	Approach feed (infeed rate) (default: active feed rate)
E	Reduced feed rate for circular elements (default: active feed rate)
U	Overlap factor (range: 0 < U < 1; default 0.5)
O	Roughing or finishing ■ 0: Roughing ■ 1: Finishing
H	Cutting direction ■ 0: Up-cut milling ■ 1: Climb milling
SCI	Safety clearance in the working plane
SCK	Safety clearance in infeed direction (see page 128)
MT	M after T: M function that is executed after the tool call T





MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent) ■ Main drive ■ Opposing spindle for rear-face machining

Type of machining for technology database access: **Milling**

Cycle run

1. Activate the C axis and position to **`spindle angle C`** at rapid traverse (only in **Teach-in mode**)
2. Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling planes)
3. Move to the safety clearance and plunge to the first milling plane

Roughing

4. Machine the milling plane, taking **`milling direction J`** (unidirectional or bidirectional) into account
5. Plunge to the next milling plane
6. Repeat 4 to 5 until the milling depth is reached

Finishing:

4. Finish-machine the edge of the island—one working plane after the other
5. Finish-machine the floor from the outside towards the inside

All versions:

6. Position to **`starting point Z`** and deactivate C axis
7. Move to the tool change point according to the G14 setting

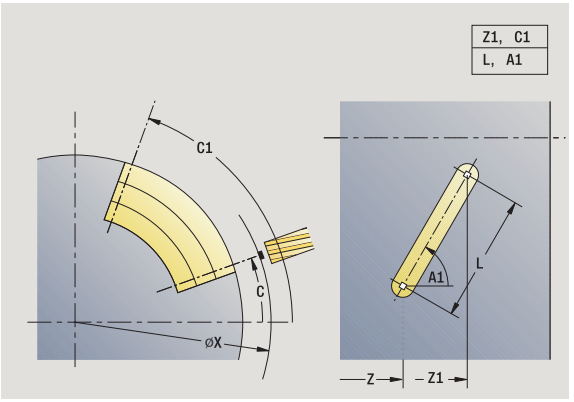
Slot, radial

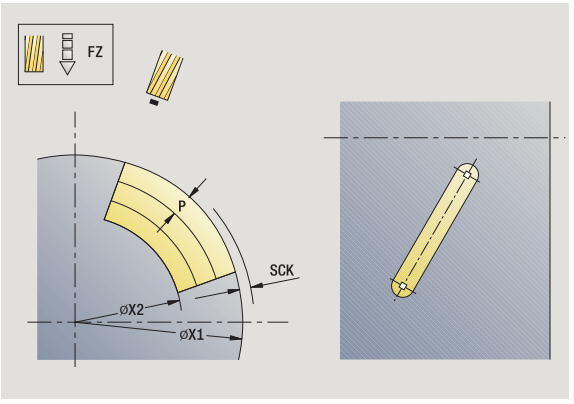


Select **milling**



Select "Slot, radial"





This cycle mills a slot on the lateral surface. The slot width equals the diameter of the milling cutter.

Cycle parameters (first input window)

X, Z	Starting point
C	Spindle angle (C-axis position)
Z1	Slot target point
C1	Angle of slot target point (default: spindle angle C)
L	Slot length
A	Angle to Z axis (default: 0)
X1	Milling top edge (diameter) – (default: Starting point X)
X2	Milling floor
P	Infeed depth (default: total depth of one infeed)
FZ	Approach feed (infeed rate) (default: active feed rate)
SCK	Safety clearance in infeed direction (see page 128)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step
WP	Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Milling**

Parameter combinations for the position and orientation of the slot:

- X1, C1
- L, A1

Cycle run

1. Activate the C axis and position to **`spindle angle C`** at rapid traverse (only in **Teach-in mode**)
2. Calculate the proportioning of cuts
3. Approach at **`infeed rate FZ`**
4. Mill to slot end point at programmed feed rate
5. Approach at **`infeed rate FZ`**
6. Machine to starting point of slot
7. Repeat 3 to 6 until the milling depth is reached
8. Position to **`starting point X`** and deactivate C axis
9. Move to the tool change point according to the G14 setting

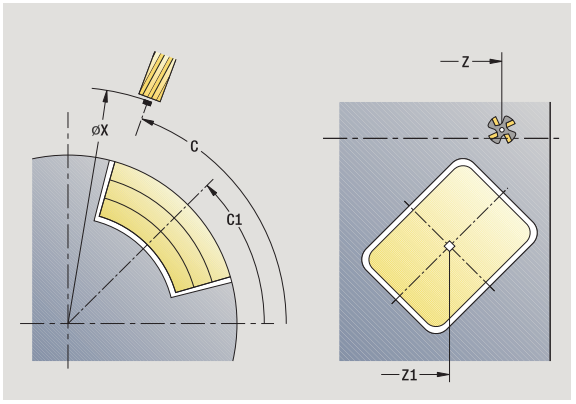
Figure, radial

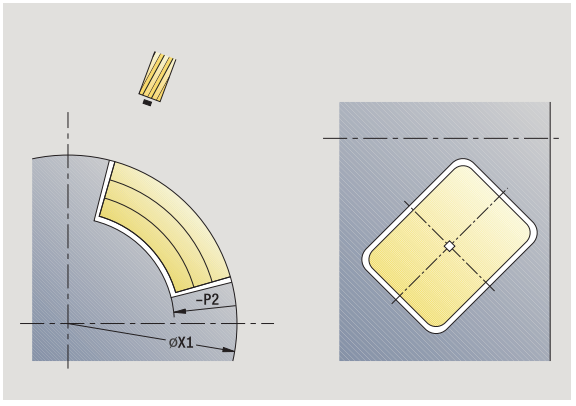


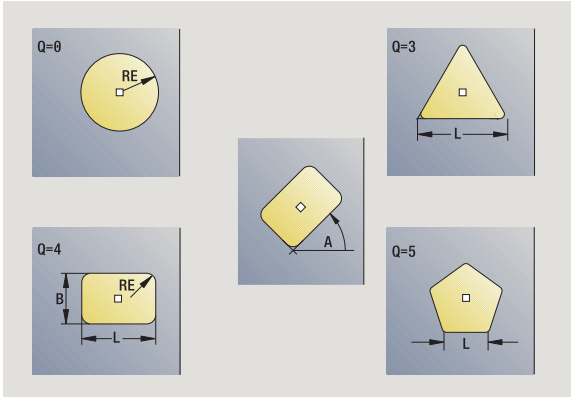
Select **milling**



Select "Figure, radial"







Depending on the parameters, the cycle mills one of the following contours or roughs/finishes a pocket on the lateral surface:

- Rectangle (Q=4, L<>B)
- Square (Q=4, L=B)
- Circle (Q=0, RE>0, L and B: No input)
- Triangle or polygon (Q=3 or Q>4, L>0 or L<0)

Cycle parameters (first input window)

X, Z — Starting point

C — Spindle angle (C-axis position)—(default: current spindle position)

Z1 — Figure center

C1 — Angle of figure center (default: Spindle angle C)

Q — Number of edges (default: 0)

- Q=0: Circle
- Q=4: Rectangle, square
- Q=3: Triangle
- Q>4: Polygon

L — Edge length

- Rectangle: Length of rectangle
- Square, polygon: Edge length
- Polygon: L<0 inside circle diameter
- Circle: No input

B — Rectangle width

- Rectangle: Width of rectangle
- Square: L=B
- Polygon, circle: No input

RE — Rounding radius (default: 0)

- Rectangle, square, polygon: Rounding radius
- Circle: Radius of circle

A — Angle to X axis (default: 0)

- Rectangle, square, polygon: Position of figure
- Circle: No input

X1 — Milling top edge (diameter) – (default: Starting point X)

P2 — Milling depth

G14 — Tool change point (see page 128)

T Turret pocket number
ID Tool ID number
S Spindle speed/cutting speed
F Feed per revolution

Cycle parameters (second input window)

I Contour-parallel oversize
K Infeed-direction oversize
P Infeed depth (default: total depth of one infeed)
FZ Approach feed (infeed rate) (default: active feed rate)
E Reduced feed rate for circular elements (default: active feed rate)
O Roughing or finishing—only for pocket milling

- 0: Roughing
- 1: Finishing

H Cutting direction

- 0: Up-cut milling
- 1: Climb milling

U Overlap factor (range: 0 < U < 1)

- No input: Contour milling
- U>0: Pocket milling—minimum overlap of milling paths = U * milling diameter

JK Contour milling (input is evaluated only for contour milling)

- 0: On the contour
- 1: Within the contour
- 2: Outside the contour

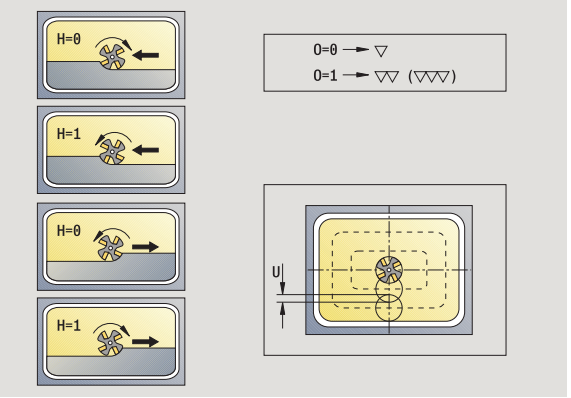
JT Pocket milling (input is evaluated only for pocket milling)

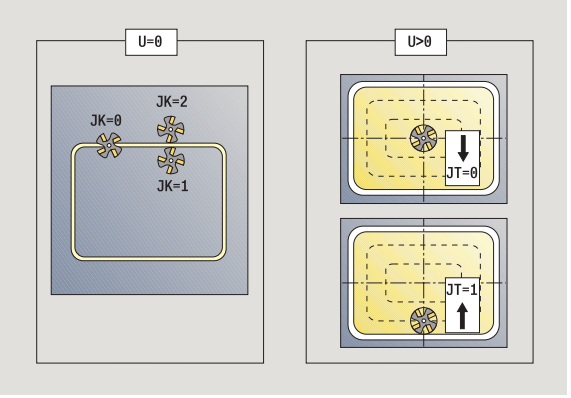
- 0: From the inside out (from the inside towards the outside)
- 1: From the outside in (from the outside towards the inside)

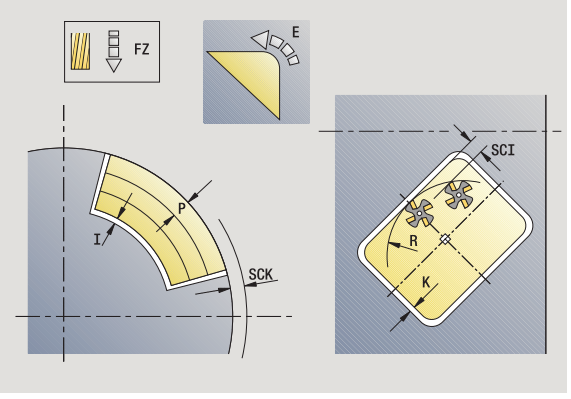
R Approach radius: Radius of approaching/departing arc (default: 0)

- R=0: Contour element is approached directly; infeed to starting point above the milling plane—then vertical plunge
- R>0: Tool moves on approaching/departing arc that connects tangentially to the contour element
- R<0 for inside corners: Tool moves on approaching/departing arc that connects tangentially to the contour element
- R<0 for outside corners: Length of linear approaching/departing element; contour element is approached/departed tangentially

SCI Safety clearance in the working plane
SCK Safety clearance in infeed direction (see page 128)







MT M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

Cycle parameters (third input window)

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Milling**

Notes on parameters/functions:

- **Machining of contour or pocket:** defined with **`overlap factor U`**.
- **Milling direction:** depends on definition of **`cutting direction H`** and the direction of tool rotation (see "Milling direction for contour milling" on page 328).
- **Milling cutter compensation:** effective (except for contour milling with JK=0).
- **Approach and departure:** For closed contours, the starting point of the first element (in rectangles, the longer element) is the point of approach and departure. The tool approaches directly or on an arc according to **`approaching radius R.`**
- **`Contour milling JK`** defines whether the milling cutter is to machine on the contour (center of milling cutter on the contour) or on the inside/outside of the contour.
- **Pocket milling—roughing (O=0):** Use **`JT`** to define whether a pocket is machined from the inside toward the outside, or vice versa.
- **Pocket milling—finishing (O=1):** First, the edge of the pocket is machined; then the pocket floor is machined. With **`JT`** you define whether a pocket floor is to be finished from the inside towards the outside, or vice versa.

Cycle run

1. Activate the C axis and position to **spindle angle C** at rapid traverse (only in **Teach-in mode**)
2. Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling planes)

Contour milling:

3. Depending on the **approaching radius R,** approach the workpiece and plunge to the first milling plane
4. Mill the first plane
5. Plunge to the next milling plane
6. Repeat 5 to 6 until the milling depth is reached

Pocket milling—roughing:

3. Move to the safety clearance and plunge to the first milling plane
4. Depending on **JT,** machine a milling plane either from the inside towards the outside, or vice versa
5. Plunge to the next milling plane
6. Repeat 4 to 5 until the milling depth is reached

Pocket milling—finishing:

3. Depending on the **approaching radius R,** approach the workpiece and plunge to the first milling plane
4. Finish-machine the edge of the pocket—one working plane after the other
5. Depending on **JT,** finish the pocket floor either from the inside towards the outside, or vice versa
6. Finish-machine the pocket at the programmed feed rate

All versions:

7. Position to **starting point Z** and deactivate C axis
8. Move to the tool change point according to the G14 setting

ICP contour, radial

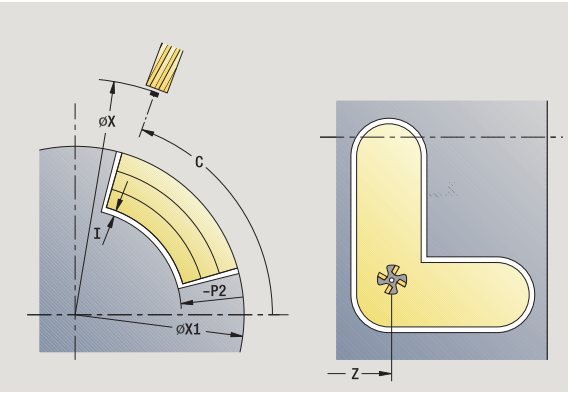


Select **milling**



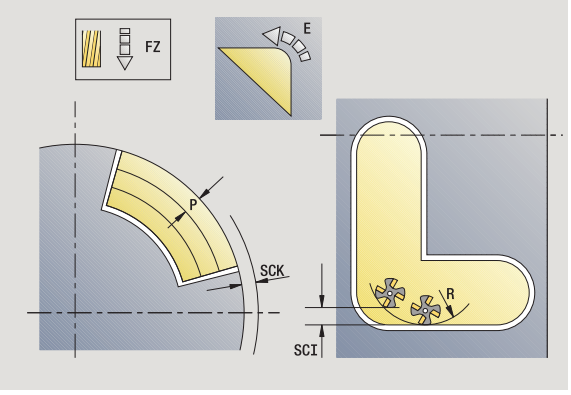
Select **ICP contour, radial**

Depending on the parameters, the cycle mills a contour or roughs/ finishes a pocket on the lateral surface.



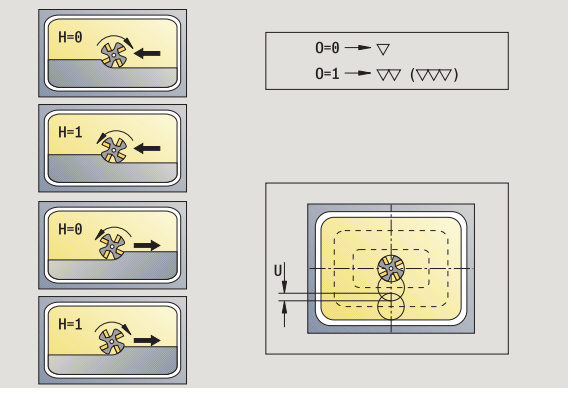
Cycle parameters (first input window)

X, Z	Starting point
C	Spindle angle (C-axis position)
X1	Milling top edge (diameter) – (default: Starting point X)
P2	Milling depth
I	Contour-parallel oversize
K	Infeed-direction oversize
P	Infeed depth (default: total depth of one infeed)
FZ	Approach feed (infeed rate) (default: active feed rate)
E	Reduced feed rate for circular elements (default: active feed rate)
FK	ICP contour number
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution



Cycle parameters (second input window)

O	Roughing or finishing—only for pocket milling
	■ 0: Roughing
	■ 1: Finishing
	■ 2: Deburr
H	Cutting direction
	■ 0: Up-cut milling
	■ 1: Climb milling
U	Overlap factor (range: 0 < U < 1)
	■ No input: Contour milling
	■ U>0: Pocket milling—minimum overlap of milling paths = U * milling diameter
JK	Contour milling (input is evaluated only for contour milling)
	■ 0: On the contour
	■ 1: Within the contour
	■ 2: Outside the contour



JT Pocket milling (input is evaluated only for pocket milling)

- 0: From the inside out (from the inside towards the outside)
- 1: From the outside in (from the outside towards the inside)

R Approach radius: Radius of approaching/departing arc (default: 0)

- R=0: Contour element is approached directly; infeed to starting point above the milling plane—then vertical plunge
- R>0: Tool moves on approaching/departing arc that connects tangentially to the contour element
- R<0 for inside corners: Tool moves on approaching/departing arc that connects tangentially to the contour element
- R<0 for outside corners: Length of linear approaching/departing element; contour element is approached/departed tangentially

SCI Safety clearance in the working plane

SCK Safety clearance in infeed direction (see page 128)

BG Chamfer width for deburring

JG Preparation diameter

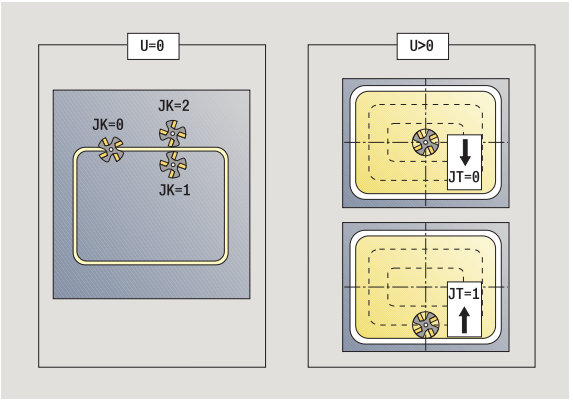
MT M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining



Type of machining for technology database access: **Milling**

Notes on parameters/functions:

- **Machining of contour or pocket:** defined with **`overlap factor U`**.
- **Milling direction:** depends on definition of **`cutting direction H`** and the direction of tool rotation (see "Milling direction for contour milling" on page 328).
- **Milling cutter compensation:** effective (except for contour milling with JK=0).
- **Approach and departure:** For closed contours, the starting point of the first element (in rectangles, the longer element) is the point of approach and departure. The tool approaches directly or on an arc according to **`approaching radius R.`**

Notes on parameters/functions:

- **Contour milling JK** defines whether the milling cutter is to machine on the contour (center of milling cutter on the contour) or on the inside/outside of the contour. **Open contours** are machined in direction of contour definition. **JK** defines whether to move to the left or right of the contour.
- **Pocket milling—roughing (O=0):** Use **JT** to define whether a pocket is machined from the inside toward the outside, or vice versa.
- **Pocket milling—finishing (O=1):** First, the edge of the pocket is machined; then the pocket floor is machined. With **JT** you define whether a pocket floor is to be finished from the inside towards the outside, or vice versa.

Cycle run

1 Activate the C axis and position to **spindle angle C** at rapid traverse (only in **Teach-in mode**)

2 Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling planes)

Contour milling:

3 Depending on the **approaching radius R,** approach the workpiece and plunge to the first milling plane

4 Mill the first plane

5 Plunge to the next milling plane

6 Repeat 5 to 6 until the milling depth is reached

Pocket milling—roughing:

3 Move to the safety clearance and plunge to the first milling plane

4 Depending on **pocket milling JT,** machine the milling plane either from the inside towards the outside, or vice versa

5 Plunge to the next milling plane

6 Repeat 4 to 5 until the milling depth is reached

Pocket milling—finishing:

3 Depending on the **approaching radius R,** approach the workpiece and plunge to the first milling plane

4 Finish-machine the edge of the pocket—one working plane after the other

5 Depending on **pocket milling JT,** finish the pocket floor either from the inside towards the outside, or vice versa

6 Finish-machine the pocket at the programmed feed rate

All versions:

7 Position to **starting point Z** and deactivate C axis

8 Move to the tool change point according to the G14 setting

Helical-slot milling, radial

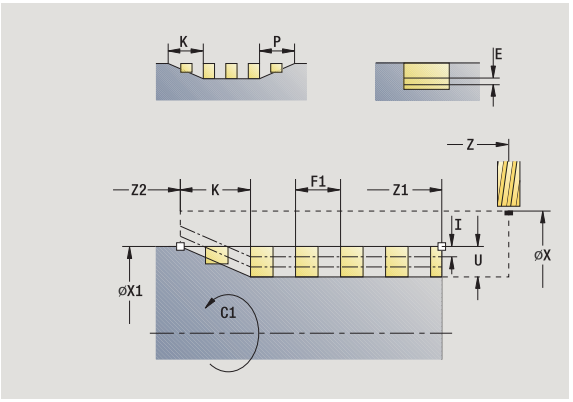


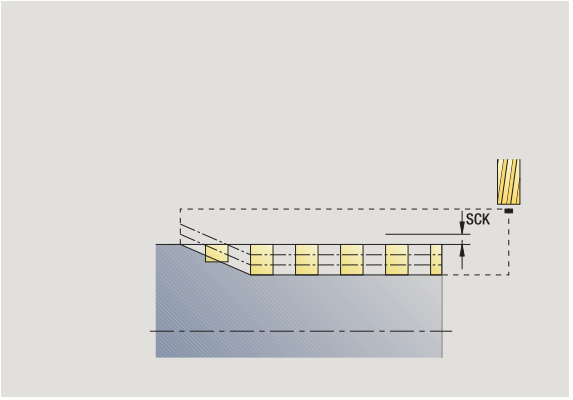
Select **milling**



Select **helical-slot milling, radial**

The cycle mills a helical slot from the **contour starting point** to the **thread end point.** The **starting angle** defines the starting position for the slot. The slot width equals the diameter of the milling cutter.





Cycle parameters

X, Z	Starting point
C	Spindle angle (C-axis position)
X1	Thread diameter
C1	Starting angle
Z1	Starting point of thread
Z2	End point of thread
F1	Thread pitch ■ F1 positive: Right-hand thread ■ F1 negative: Left-hand thread
U	Thread depth
I	Maximum infeed. The infeed movements are reduced down to >= 0.5 mm according to the following calculation. Following that, each infeed movement will amount to 0.5 mm. ■ Infeed 1: "I" ■ Infeed n: I * (1 – (n–1) * E)
E	Cutting depth reduction
P	Run-in length (ramp at the beginning of the slot)
K	Run-out length (ramp at the end of the slot)
G14	Tool change point (see page 128)
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
D	No. of gears (threads per unit)
SCK	Safety clearance in infeed direction (see page 128)
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

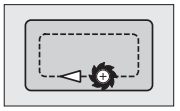
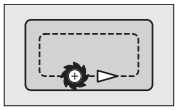
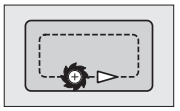
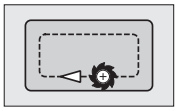
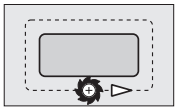
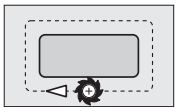
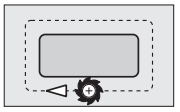
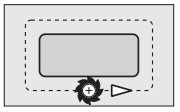
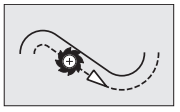
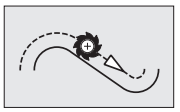
- Main drive
- Opposing spindle for rear-face machining

Type of machining for technology database access: **Milling**

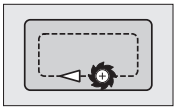
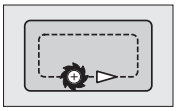
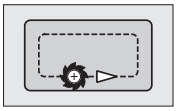
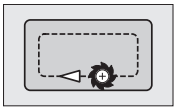
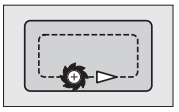
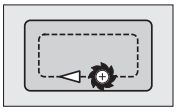
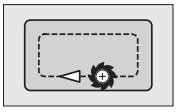
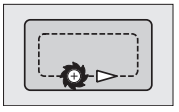
Cycle run

1. Activate the C axis and position to **`spindle angle C`** at rapid traverse (only in **Teach-in mode**)
2. Calculate current infeed
3. Position the tool for the first pass
4. Machine up to the **`thread end point Z2`** at the programmed feed rate, taking the ramps at the beginning and end of the slot into account
5. Return on paraxial path and approach for next pass
6. Repeat 4 to 5 until the slot depth is reached
7. Move to the tool change point according to the G14 setting

Milling direction for contour milling

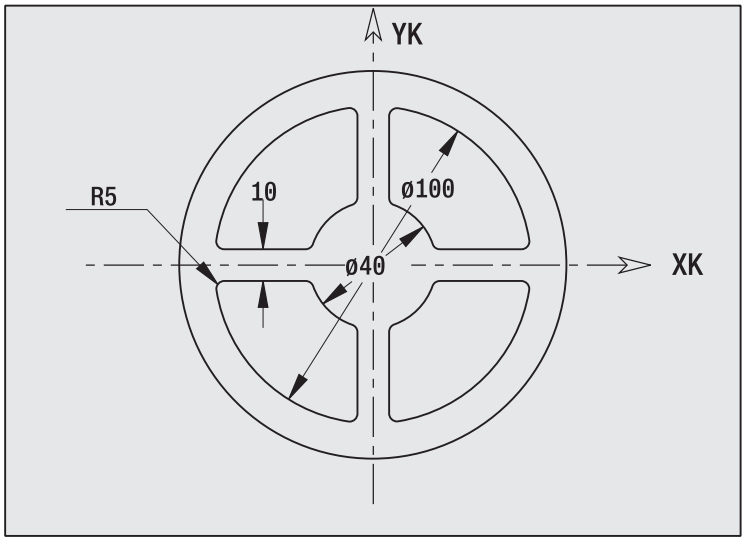
Milling direction for contour milling				
Cycle type	**Cutting direction**	**Direction of tool rotation**	**TRC**	**Execution**
Inside (JK=1)	Up-cut milling (H=0)	Mx03	Right	
Inside	Up-cut milling (H=0)	Mx04	Left	
Inside	Climb milling (H=1)	Mx03	Left	
Inside	Climb milling (H=1)	Mx04	Right	
Outside (JK=2)	Up-cut milling (H=0)	Mx03	Right	
Outside	Up-cut milling (H=0)	Mx04	Left	
Outside	Climb milling (H=1)	Mx03	Left	
Outside	Climb milling (H=1)	Mx04	Right	
Right (JK=2)	For open contours without function. Machining in direction of contour definition	Without effect	Right	
Left (JK=1)	For open contours without function. Machining in direction of contour definition	Without effect	Left	

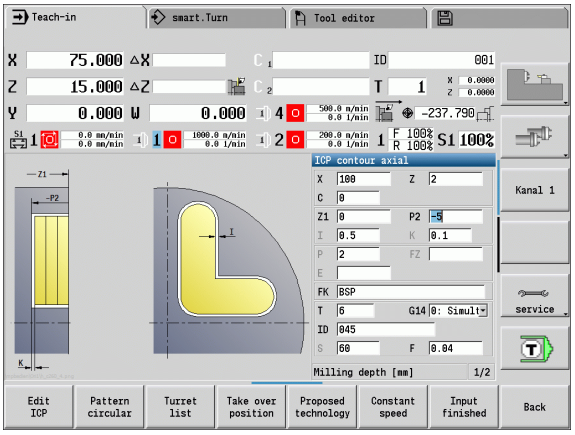
Milling direction for pocket milling

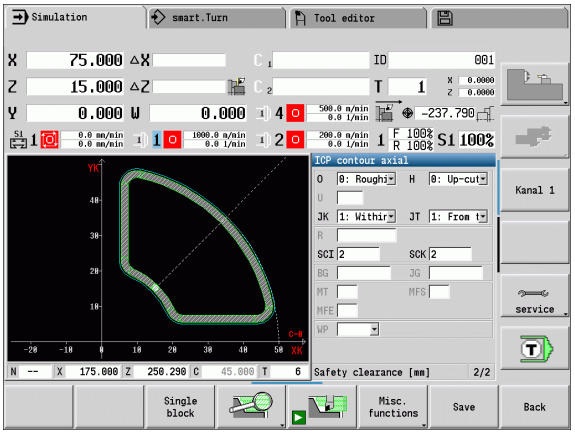
Milling direction for pocket milling				
Machining	**Cutting direction**	**Machining direction**	**Direction of tool rotation**	**Execution**
Roughing Finishing	Up-cut milling (H=0)	From inside toward the outside (JT=0)	Mx03	
Roughing Finishing	Up-cut milling (H=0)	From inside toward the outside (JT=0)	Mx04	
Roughing	Climb milling (H=0)	From outside toward inside (JT=1)	Mx03	
Roughing	Up-cut milling (H=0)	From outside toward inside (JT=1)	Mx04	
Roughing Finishing	Climb milling (H=1)	From inside toward the outside (JT=0)	Mx03	
Roughing Finishing	Climb milling (H=1)	From inside toward the outside (JT=0)	Mx04	
Roughing	Climb milling (H=1)	From outside toward inside (JT=1)	Mx03	
Roughing	Up-cut milling (H=1)	From outside toward inside (JT=1)	Mx04	

Example of milling cycle

Milling on the face







In this example, a pocket is milled. The milling example in "9.8 ICP Example, Milling Cycle" illustrates the complete machining process on the face, including contour definition.

The machining process is performed with the cycle **ICP contour, axial.** To describe a contour, define the basic contour first. Then superimpose the rounding arcs.

Tool data (milling cutter)

- WO = 8 (tool orientation)
- I = 8 (milling diameter)
- K = 4 (number of teeth)
- TF = 0.025 (feed per tooth)

Engraving, axial

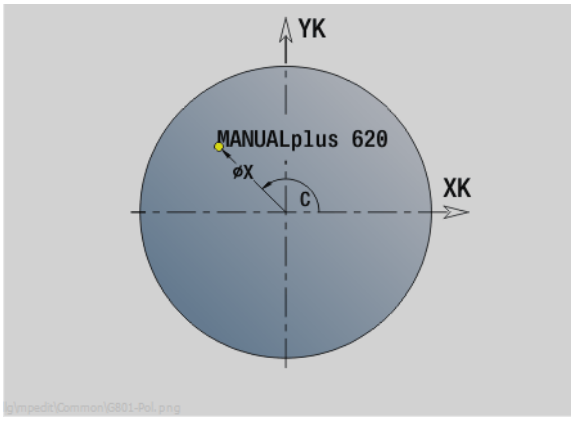
The "Radial engraving" cycle engraves character strings in linear or polar layout on the face of the workpiece. For character table and more information, see page 335

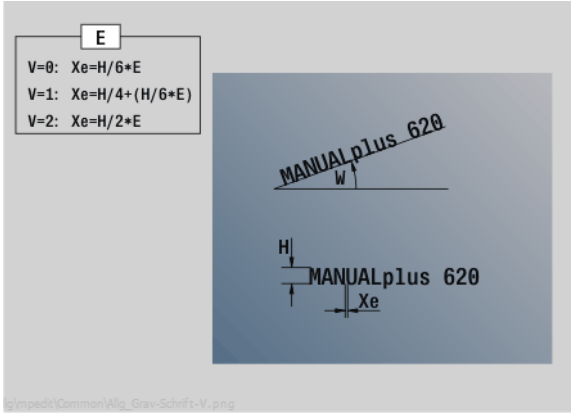
You define the starting point of the character string in the cycle. If you do not define a starting point, the cycle starts at the current tool position.

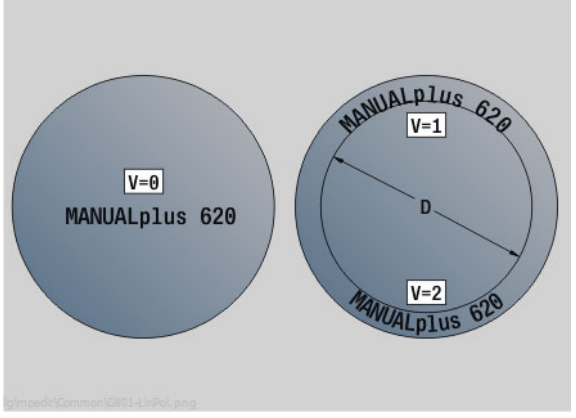
You can also engrave a logotype with several calls. For this purpose, specify the starting point with the first call. All other calls are programmed without a starting point.

Parameters:

X	Starting point (diameter dimension): Tool pre-positioning
Z	Starting point: Tool pre-positioning
C	Spindle angle: Workpiece spindle pre-positioning
TX	Text to be engraved
NF	Character number: ASCII code of the character to be engraved
Z2	End point Z position, infeed depth during engraving.
X1	Starting point (polar), first character
C1	Start angle (polar), first character
XK	Starting point (Cartesian), first character
YK	Starting point (Cartesian), first character
H	Font height
E	Distance factor (for calculation see figure)
T	Turret pocket number
G14	Tool change point (see page 128)
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
W	Inclination angle of character string
FZ	Plunging feed rate factor (plunging feed rate = current feed rate * F)
V	Execution linear, curved above or below
D	Reference diameter







Parameters:

RB Retraction plane. Z position retracted to for positioning.
SCK Safety clearance (see page 128)
MT M after T: M function that is executed after the tool call T
MFS M at beginning: M function that is executed at the beginning of the machining step
MFE M at end: M function that is executed at the end of the machining step
WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining



The engraving cycles are not available in manual operation.

Cycle run

1. Activate the C axis and position to **spindle angle C** at rapid traverse, **starting point X** and **Z**
2. Position to starting point if defined
3. Approach at **plunging feed rate FZ**
4. Engrave with programmed feed rate
5. Position tool to **retraction plane RB** or if no **RB** is defined, to **starting point Z**
5. Position tool to the next character
6. Repeat steps 3 to 5 until all characters are engraved
7. Position to **starting point X**, **Z** and deactivate C axis
8. Move to the tool change point according to the G14 setting

Engraving, radial

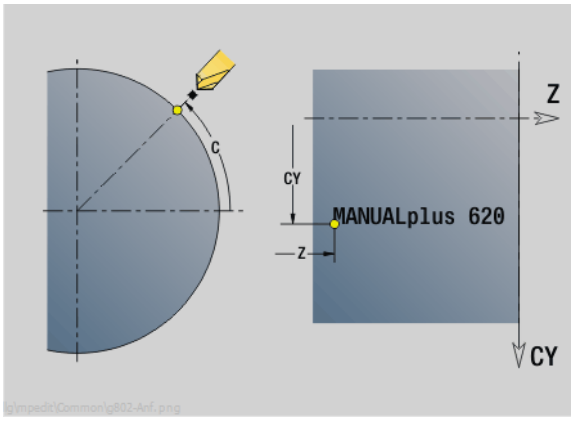
The "Radial engraving" cycle engraves character strings in linear layout on the face of the workpiece. For character table and more information, see page 335

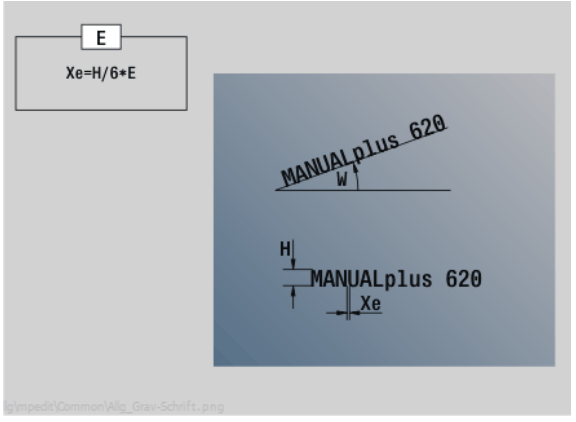
You define the starting point of the character string in the cycle. If you do not define a starting point, the cycle starts at the current tool position.

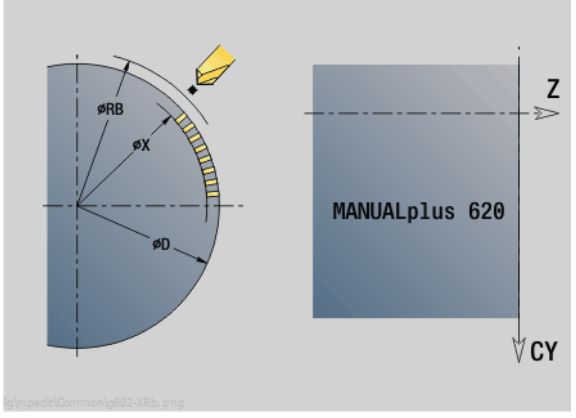
You can also engrave a logotype with several calls. For this purpose, specify the starting point with the first call. All other calls are programmed without a starting point.

Parameters:

X	Starting point (diameter dimension): Tool pre-positioning
Z	Starting point: Tool pre-positioning
C	Spindle angle: Workpiece spindle pre-positioning
TX	Text to be engraved
NF	Character number: ASCII code of the character to be engraved
X2	End point (diameter dimension): X position, infeed depth during engraving.
Z1	Starting point, first character
C1	Start angle, first character
CY	Starting point, first character
D	Reference diameter
H	Font height
E	Distance factor (for calculation see figure)
T	Turret pocket number
G14	Tool change point (see page 128)
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
W	Inclination angle of character string
FZ	Plunging feed rate factor (plunging feed rate = current feed rate * F)
RB	Retraction plane. X position retracted to for positioning.







Parameters:

SCK Safety clearance (see page 128)

MT M after T: M function that is executed after the tool call T

MFS M at beginning: M function that is executed at the beginning of the machining step

MFE M at end: M function that is executed at the end of the machining step

WP Displays which workpiece spindle is used to process the cycle (machine-dependent)

- Main drive
- Opposing spindle for rear-face machining



The engraving cycles are not available in manual operation.

Cycle run

1. Activate the C axis and position to **spindle angle C** at rapid traverse, **starting point X** and **Z**
2. Position to starting point if defined
3. Approach at **plunging feed rate FZ**
4. Engrave with programmed feed rate
5. Position tool to **retraction plane RB** or if no **RB** is defined, to **starting point X**
5. Position tool to the next character
6. Repeat steps 3 to 5 until all characters are engraved
7. Position to **starting point X**, **Z** and deactivate C axis
8. Move to the tool change point according to the G14 setting



Engraving, axial/radial

The CNC PILOT can realize the characters listed in the following table. The text to be engraved is entered as a character string. Diacritics and special characters that you cannot enter in the editor can be defined, character by character, in **NF**. If text is defined in **ID** and a character is defined in **NF**, the text is engraved before the character.



The engraving cycles are not available in manual operation.

Small letters		Capital letters		Numerals, diacritics		Special characters		
NF	**Character**	**NF**	**Character**	**NF**	**Character**	**NF**	**Character**	**Meaning**
97	a	65	A	48	0	32		Blank space
98	b	66	B	49	1	37	%	Per cent sign
99	c	67	C	50	2	40	(	Opening parenthesis
100	d	68	D	51	3	41	)	Closing parenthesis
101	e	69	E	52	4	43	+	Plus character
102	f	70	F	53	5	44	,	Comma
103	g	71	G	54	6	45	–	Minus sign
104	h	72	H	55	7	46	.	Point
105	i	73	I	56	8	47	/	Forward slash
106	j	74	J	57	9	58	:	Colon
107	k	75	K			60	<	Less than character
108	l	76	L	196	Ä	61	=	Equal sign
109	m	77	M	214	Ö	62	>	Greater than character
110	n	78	N	220	Ü	64	@	at
111	o	79	O	223	ß	91	[	Opening brackets
112	p	80	P	228	ä	93	]	Closing brackets
113	q	81	Q	246	ö	95	_	Underscore
114	r	82	R	252	ü	8364		Euro sign
115	s	83	S			181	µ	Micro
116	t	84	T			186	°	Degrees
117	u	85	U			215	*	Multiplication sign
118	v	86	V			33	!	Exclamation point
119	w	87	W			38	&	Ampersand and
120	x	88	X			63	?	Question mark
121	y	89	Y			174	®	Trademark
122	z	90	Z			216	Ø	Diameter sign

4.9 Drilling and milling patterns

Note on using drilling/milling patterns:

- **Drilling patterns:** The CNC PILOT generates the machine commands M12, M13 (apply/release block brake) under the following conditions: the drill/tap must be entered as driven tool (parameters **`driven tool AW`** and **`direction of rotation MD`** must be defined).
- **ICP milling contours:** If the contour starting point is outside the coordinate datum, the distance between contour starting point and coordinate datum is added to the pattern position (see "Examples of pattern machining" on page 353).

Linear drilling pattern, axial

DRILLING PATTERN LINEAR, AXIAL



Select **drilling**



Select **drilling, axial**



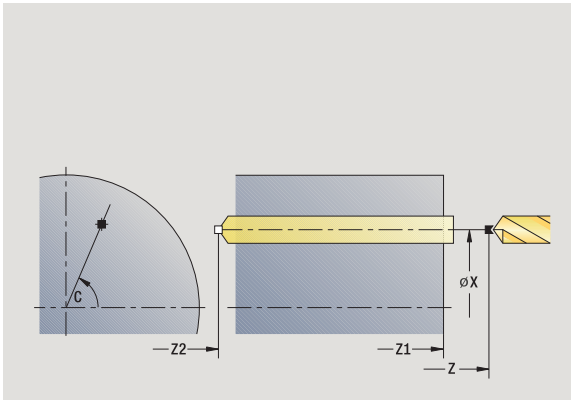
Select **deep-hole drilling, axial**

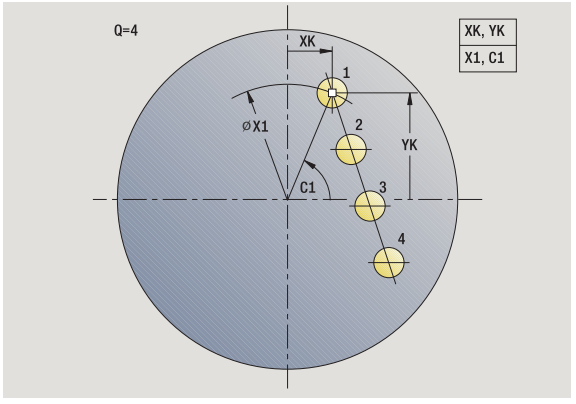


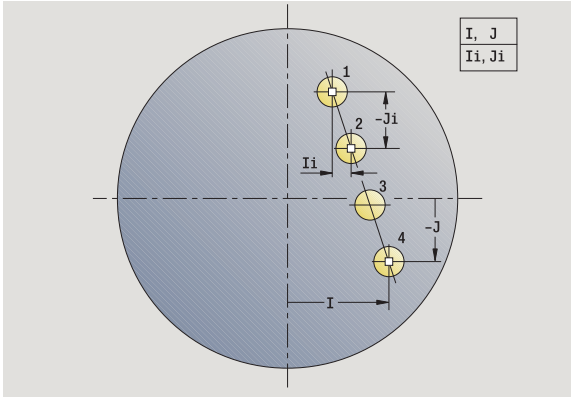
Select **tapping, axial**

Pattern linear

Press the **Pattern linear** soft key







Press **Pattern linear** to machine drilling patterns in which the individual features are arranged at a regular spacing in a straight line on the face.

Cycle parameters

X, Z	Starting point
C	Spindle angle (C-axis position)—(default: current spindle position)
Q	Number of holes
X1, C1	Starting point of pattern in polar coordinates
XK, YK	Starting point of pattern in Cartesian coordinates
I, J	End point of pattern in Cartesian coordinates
Ii, Ji:	Pattern spacing (incremental)

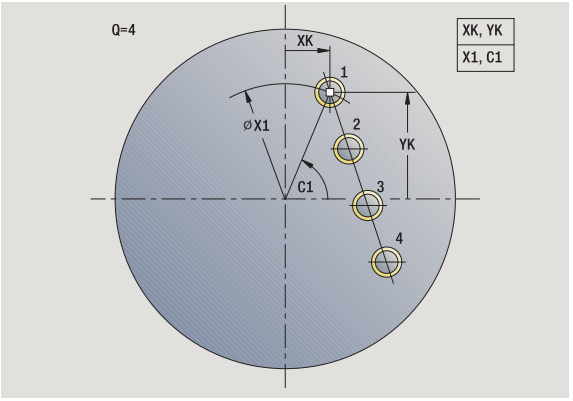
The parameters of the hole also requested.

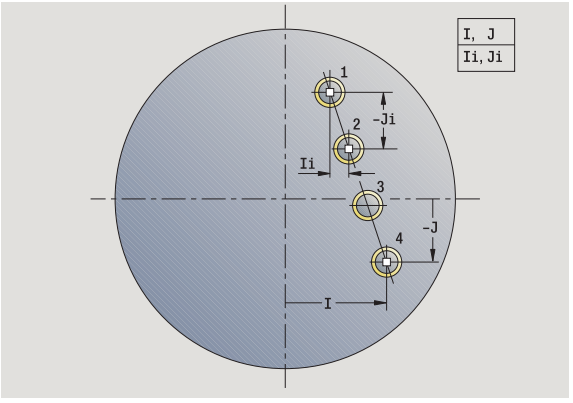
Use the following parameter combinations to define the:

- Starting point of pattern
  - X1, C1 or
  - XK, YK
- Pattern positions:
  - Ii, Ji and Q
  - I, J and Q

Cycle run

1. Positioning (depending on the machine configuration):
   - Without C axis: Position to **spindle angle C**
   - With C axis: Activate C axis and position to **spindle angle C** at rapid traverse
   - Manual mode: Machining starts from current spindle angle
2. Calculate the pattern positions
3. Position to **starting point of pattern**
4. Execute drilling operation
5. Position for the next machining operation
6. Repeat steps 4 and 5 until all machining operations have been completed
7. Return to starting point
8. Move to the tool change point according to the G14 setting





Milling pattern linear, axial

LINEAR MILLING PATTERN, AXIAL



Select **milling**

Pattern linear

Press the **Pattern linear** soft key



Select **slot, axial**



Select **recessing, axial ICP**

Press **Pattern linear** to machine milling patterns in which the individual features are arranged at a regular spacing in a straight line on the face.

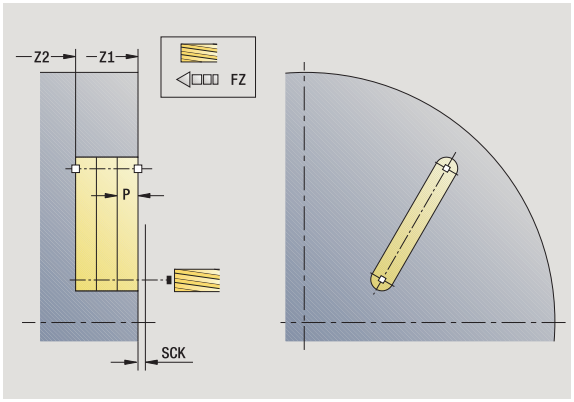
Cycle parameters

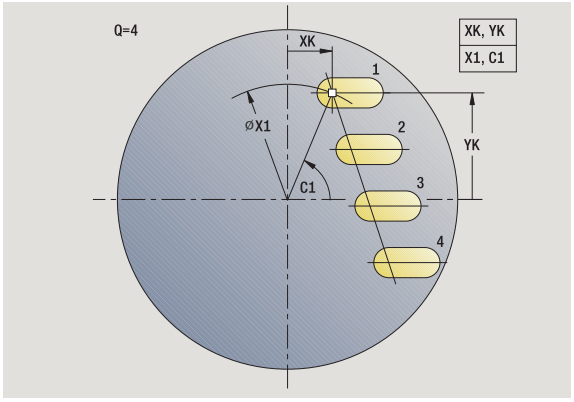
X, Z	Starting point
C	Spindle angle (C-axis position)—(default: current spindle position)
Q	Number of slots
X1, C1	Starting point of pattern in polar coordinates
XK, YK	Starting point of pattern in Cartesian coordinates
I, J	End point of pattern in Cartesian coordinates
Ii, Ji:	Pattern spacing (incremental)

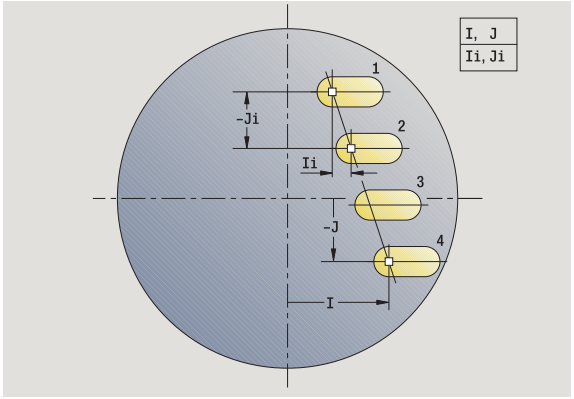
The parameters of the milling operation are also requested.

Use the following parameter combinations to define the:

- Starting point of pattern
  - X1, C1 or
  - XK, YK
- Pattern positions:
  - Ii, Ji and Q
  - I, J and Q

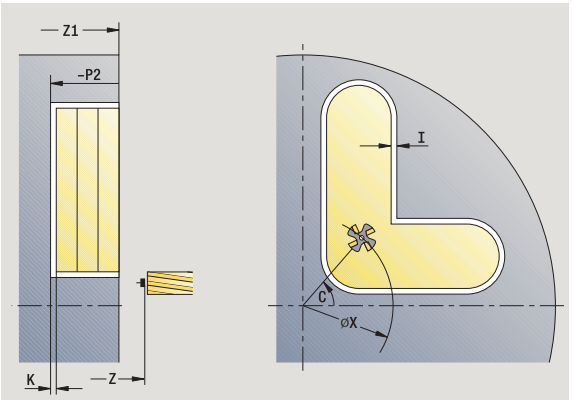


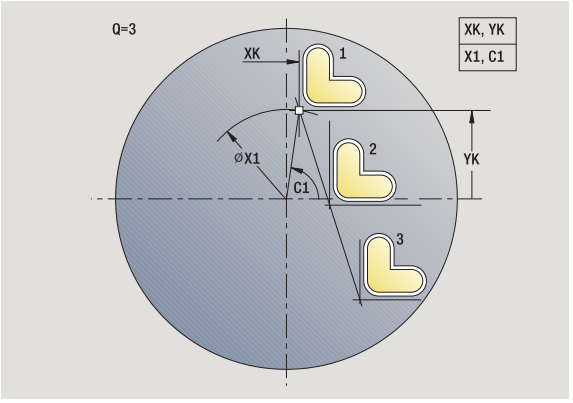


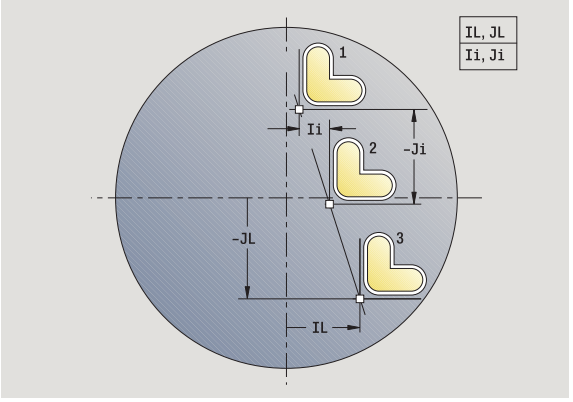


Cycle run

1. Positioning (depending on the machine configuration):
   - Without C axis: Position to **spindle angle C**
   - With C axis: Activate C axis and position to **spindle angle C** at rapid traverse
   - Manual mode: Machining starts from current spindle angle
2. Calculate the pattern positions
3. Position to **starting point of pattern**
4. Execute the milling operation
5. Position for the next machining operation
6. Repeat steps 4 and 5 until all machining operations have been completed
7. Return to starting point
8. Move to the tool change point according to the G14 setting







Drilling pattern circular, axial

CIRCULAR DRILLING PATTERN, AXIAL



Select **drilling**



Select **drilling, axial**



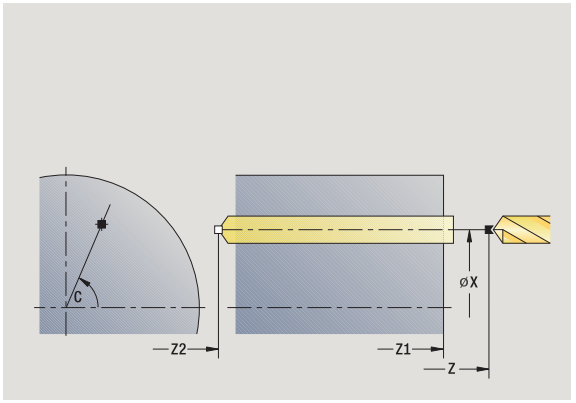
Select **deep-hole drilling, axial**

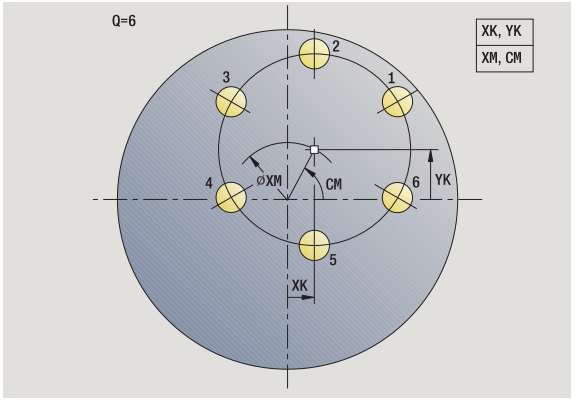


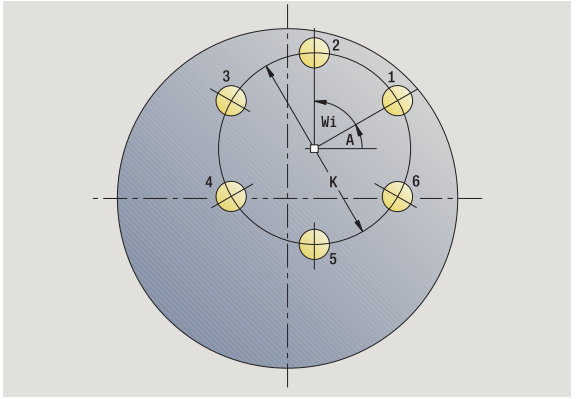
Select **tapping, axial**

Pattern circular

Press the **Pattern circular** soft key







Press **Pattern circular** to machine drilling patterns in which the individual features are arranged at a regular spacing in a circle or circular arc on the face.

Cycle parameters

X, Z	Starting point
C	Spindle angle (C-axis position)—(default: current spindle position)
Q	Number of holes
XM, CM	Center of pattern in polar coordinates
XK, YK	Center of pattern in Cartesian coordinates
K	Pattern diameter
A	Angle of 1st hole (default: 0°)
Wi	Angle increment (pattern spacing)—(default: holes are arranged at a regular spacing in a circle)

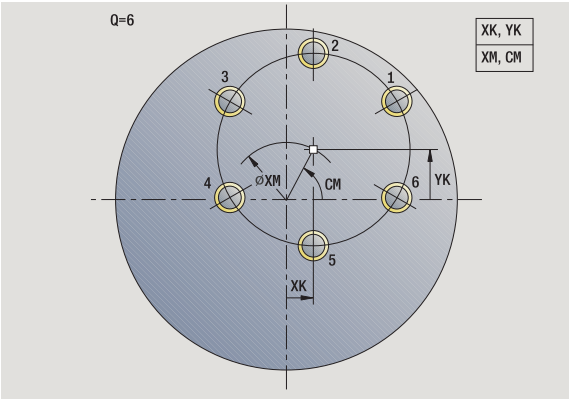
The parameters for creating the hole are also requested.

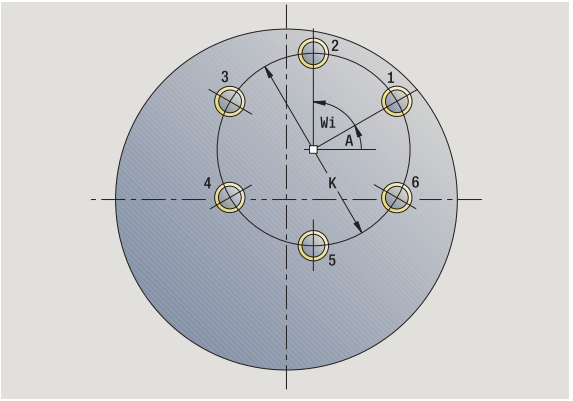
Use the following parameter combinations to define the center of the pattern:

- XM, CM or
- XK, YK

Cycle run

1. Positioning (depending on the machine configuration):
   - Without C axis: Position to **spindle angle C**
   - With C axis: Activate C axis and position to **spindle angle C** at rapid traverse
   - Manual mode: Machining starts from current spindle angle
2. Calculate the pattern positions
3. Position to **starting point of pattern**
4. Execute drilling operation
5. Position for the next machining operation
6. Repeat steps 4 and 5 until all machining operations have been completed
7. Return to starting point
8. Move to the tool change point according to the G14 setting





Milling pattern circular, axial

CIRCULAR MILLING PATTERN, AXIAL



Call the milling menu



Select **slot, axial**



Select **ICP contour, axial**

Pattern circular

Press the **Pattern circular** soft key

Press **Pattern circular** to machine milling patterns in which the individual features are arranged at a regular spacing in a circle or circular arc on the face.

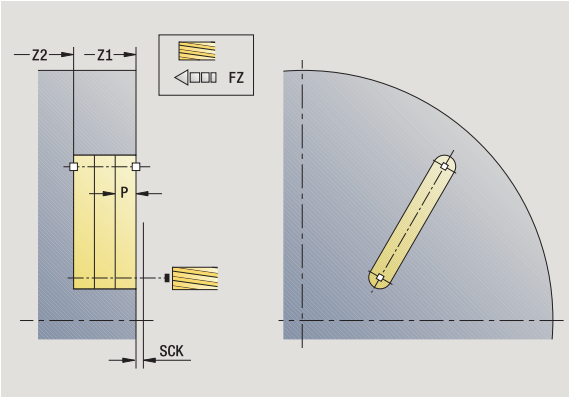
Cycle parameters

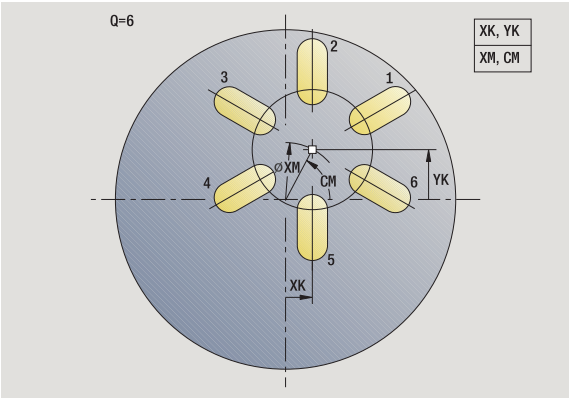
X, Z	Starting point
C	Spindle angle (C-axis position)—(default: current spindle position)
Q	Number of slots
XM, CM	Center of pattern in polar coordinates
XK, YK	Center of pattern in Cartesian coordinates
K	Pattern diameter
A	Angle of 1st slot (default: 0°)
Wi	Angle increment (pattern spacing) – (default: milling operations are arranged at a regular spacing in a circle)

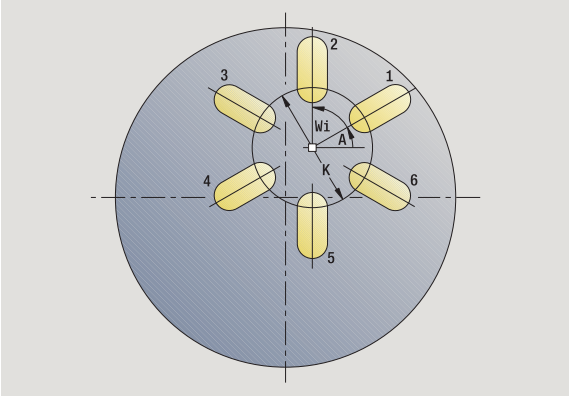
The parameters for creating the milling operation are also requested.

Use the following parameter combinations to define the center of the pattern:

- XM, CM or
- XK, YK

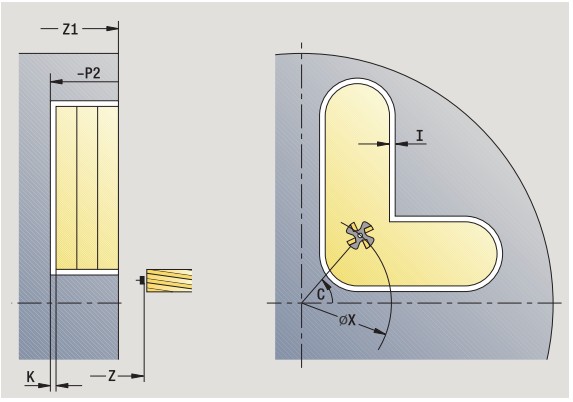


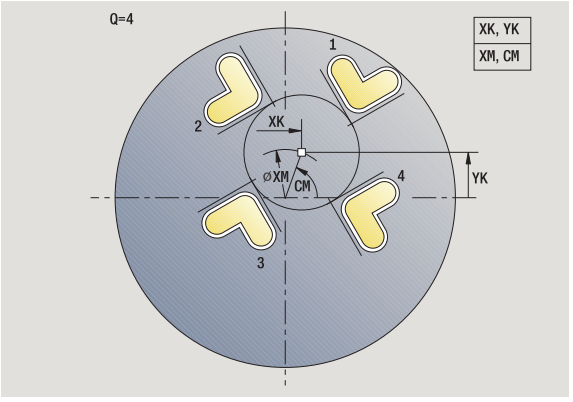


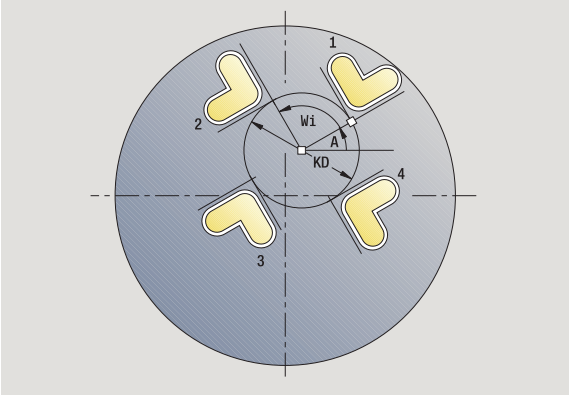


Cycle run

1 Positioning (depending on the machine configuration):
   - Without C axis: Position to **spindle angle C**
   - With C axis: Activate C axis and position to **spindle angle C** at rapid traverse
   - Manual mode: Machining starts from current spindle angle
2 Calculate the pattern positions
3 Position to **starting point of pattern**
4 Execute the milling operation
5 Position for the next machining operation
6 Repeat steps 4 and 5 until all machining operations have been completed
7 Return to starting point
8 Move to the tool change point according to the G14 setting







Drilling pattern linear, radial

DRILLING PATTERN LINEAR, RADIAL



Select **drilling**



Select **drilling, radial**



Select **deep-hole drilling, radial**



Select **tapping, radial**

Pattern linear

Press the **Pattern linear** soft key

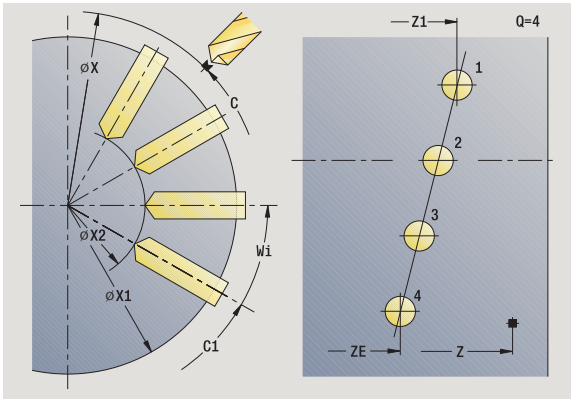
Press **Pattern linear** during drilling cycles to machine drilling patterns in which the individual features are arranged at a regular spacing in a straight line on the lateral surface.

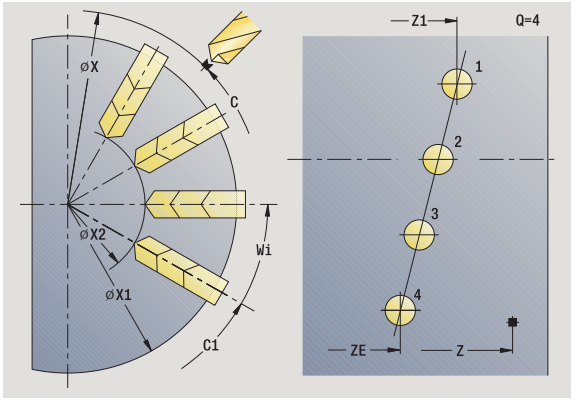
Cycle parameters

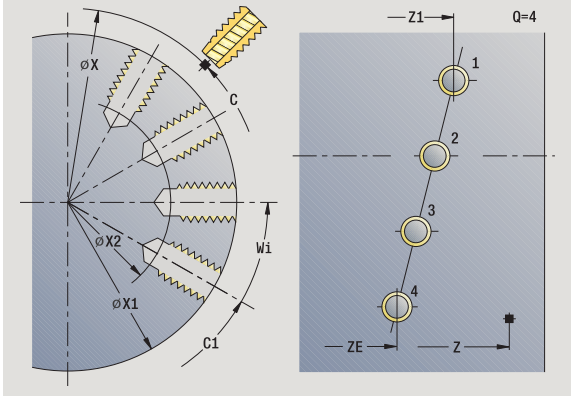
X, Z	Starting point
C	Spindle angle (C-axis position)—(default: current spindle position)
Q	Number of holes
Z1	Starting point of pattern (position of 1st hole)
ZE	End point of pattern (default: Z1)
C1	Angle of 1st hole (starting angle)
Wi	Angle increment (pattern spacing)—(default: holes are arranged at a regular spacing in a circle)

You define the pattern position with **end point of pattern** and **angle increment** or **angle increment** and **number of holes**.

The parameters of the hole also requested.







Cycle run

1. Positioning (depending on the machine configuration):
   - Without C axis: Position to **spindle angle C**
   - With C axis: Activate C axis and position to **spindle angle C** at rapid traverse
   - Manual mode: Machining starts from current spindle angle
2. Calculate the pattern positions
3. Position to **starting point of pattern**
4. Execute drilling operation
5. Position for the next machining operation
6. Repeat steps 4 and 5 until all machining operations have been completed
7. Position to **starting point Z** and deactivate C axis
8. Move to the tool change point according to the G14 setting

Linear milling pattern, radial

MILLING PATTERN LINEAR, RADIAL



Select **milling**



Press the **Pattern linear** soft key



Select **slot, radial**



Select **ICP contour, radial**

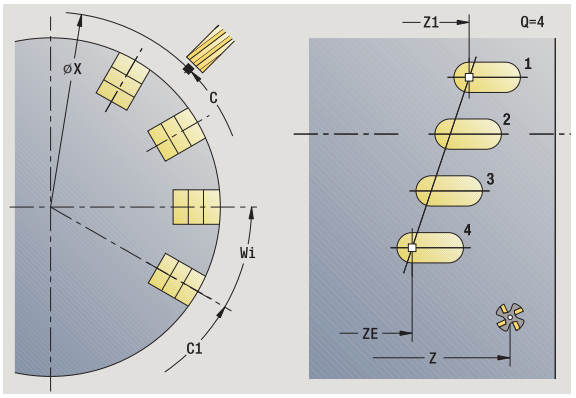
Press **Pattern linear** during milling cycles to machine milling patterns in which the individual features are arranged at a regular spacing in a straight line on the lateral surface.

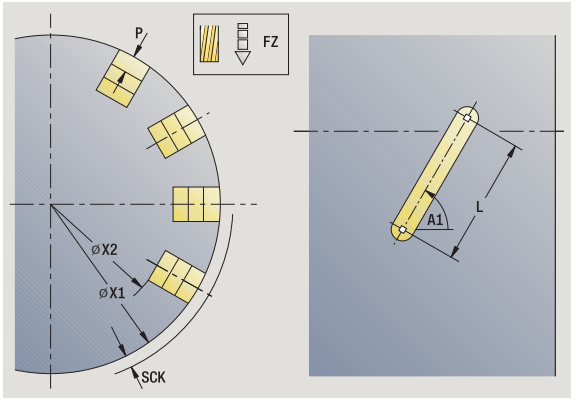
Cycle parameters

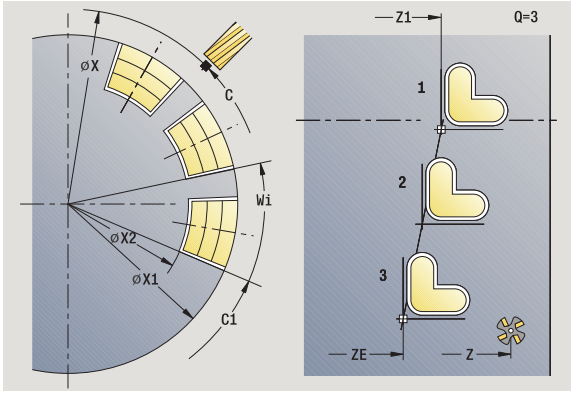
X, Z	Starting point
C	Spindle angle (C-axis position)—(default: current spindle position)
Q	Number of slots
Z1	Starting point of pattern (position of 1st slot)
ZE	End point of pattern (default: Z1)
C1	Angle of 1st slot (starting angle)
Wi	Angle increment (pattern spacing) – (default: milling operations are arranged at a regular spacing in a circle)

You define the pattern position with **end point of pattern** and **angle increment** or **angle increment** and **number of slots**.

The parameters of the milling operation are also requested.







Cycle run

1. Positioning (depending on the machine configuration):
   - Without C axis: Position to **spindle angle C**
   - With C axis: Activate C axis and position to **spindle angle C** at rapid traverse
   - Manual mode: Machining starts from current spindle angle
2. Calculate the pattern positions
3. Position to **starting point of pattern**
4. Execute the milling operation
5. Position for the next machining operation
6. Repeat steps 4 and 5 until all machining operations have been completed
7. Position to **starting point Z** and deactivate C axis
8. Move to the tool change point according to the G14 setting

Drilling pattern circular, radial

CIRCULAR DRILLING PATTERN, RADIAL



Select **drilling**



Select **drilling, radial**



Select **deep-hole drilling, radial**



Select **tapping, radial**

Pattern circular

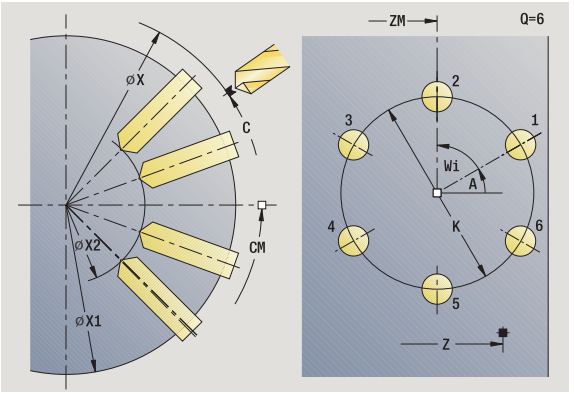
Press the **Pattern circular** soft key

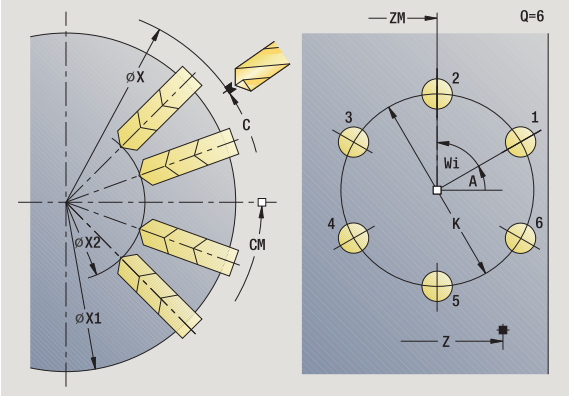
Press **Pattern circular** to machine drilling patterns in which the individual features are arranged at a regular spacing in a circle or circular arc on the lateral surface.

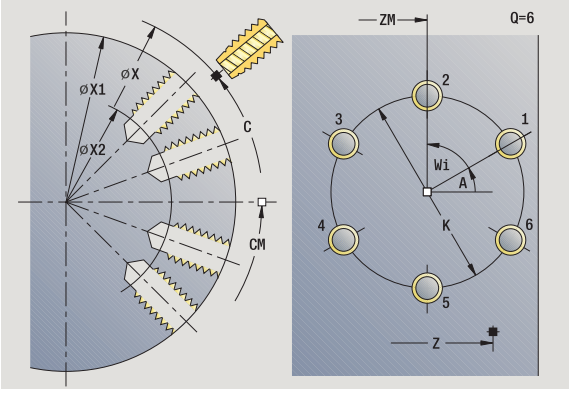
Cycle parameters

X, Z	Starting point
C	Spindle angle (C-axis position)—(default: current spindle position)
Q	Number of slots
ZM, CM	Center of pattern: Position, angle
K	Pattern diameter
A	Angle of 1st hole (default: 0°)
Wi	Angle increment (pattern spacing)—(default: holes are arranged at a regular spacing in a circle)

The parameters that are required for machining the respective elements are also requested (see corresponding cycle descriptions).

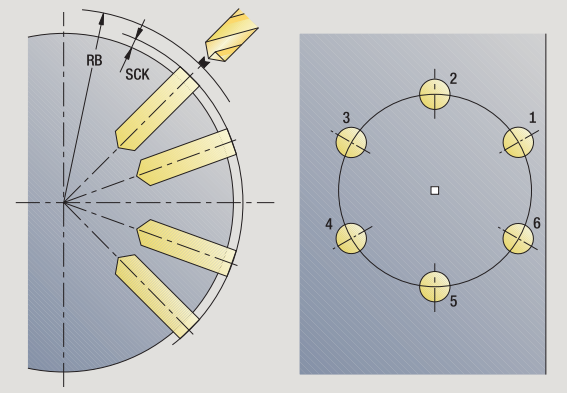






Cycle run

1. Positioning (depending on the machine configuration):
   - Without C axis: Position to **spindle angle C**
   - With C axis: Activate C axis and position to **spindle angle C** at rapid traverse
   - Manual mode: Machining starts from current spindle angle
2. Calculate the pattern positions
3. Position to **starting point of pattern**
4. Execute drilling operation
5. Position for the next machining operation
6. Repeat steps 4 and 5 until all machining operations have been completed
7. Position to **starting point Z** and deactivate C axis
8. Move to the tool change point according to the G14 setting



Circular milling pattern, radial

MILLING PATTERN CIRCULAR, RADIAL



Select **milling**



Select **slot, radial**



Select **ICP contour, radial**

Pattern circular

Press the **Pattern radial** soft key

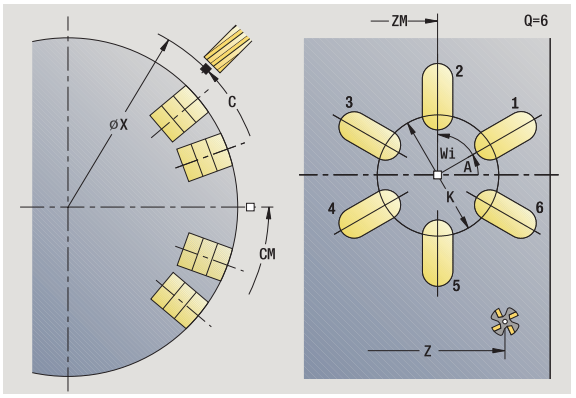
Press **Pattern circular** to machine milling patterns in which the individual features are arranged at a regular spacing in a circle or circular arc on the lateral surface.

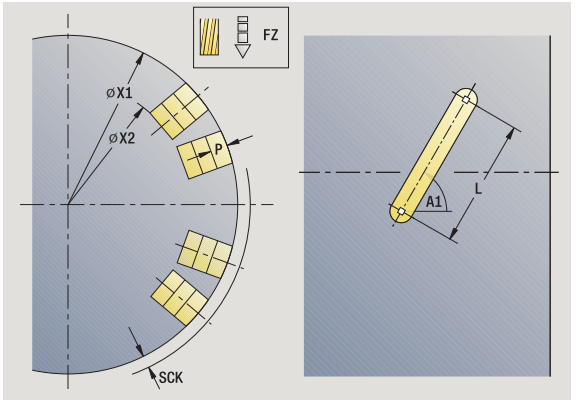
Cycle parameters

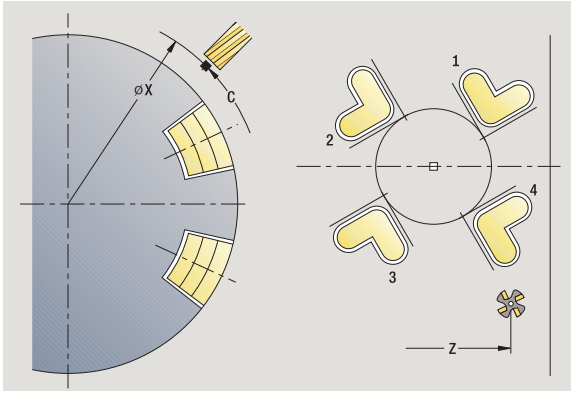
X, Z	Starting point
C	Spindle angle (C-axis position)—(default: current spindle position)
Q	Number of slots
ZM, CM	Center of pattern: Position, angle
K	Pattern diameter
A	Angle of 1st slot (default: 0°)
Wi	Angle increment (pattern spacing) – (default: milling operations are arranged at a regular spacing in a circle)

The parameters for programming the milling operation are also requested (see corresponding cycle descriptions).

The starting point of a pattern to be assigned to an ICP contour must lie on the positive XK axis.





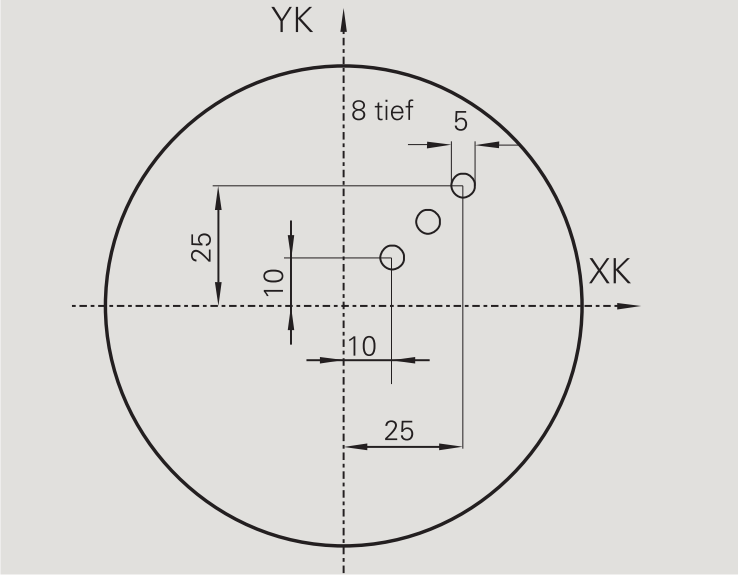


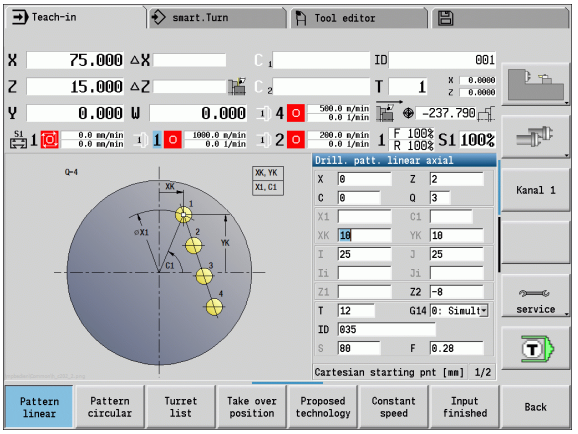
Cycle run

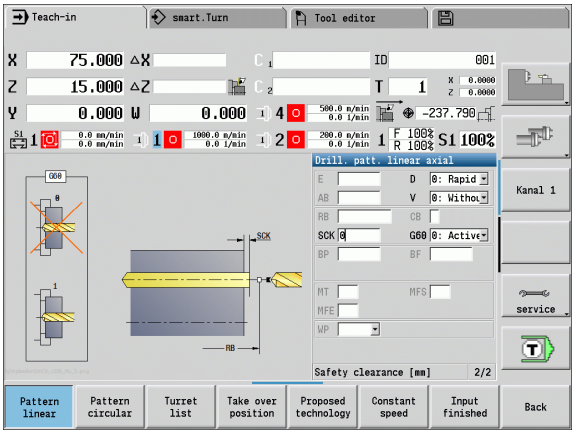
1. Positioning (depending on the machine configuration):
   - Without C axis: Position to **spindle angle C**
   - With C axis: Activate C axis and position to **spindle angle C** at rapid traverse
   - Manual mode: Machining starts from current spindle angle
2. Calculate the pattern positions
3. Position to **starting point of pattern**
4. Execute the milling operation
5. Position for the next machining operation
6. Repeat steps 4 and 5 until all machining operations have been completed
7. Position to **starting point Z** and deactivate C axis
8. Move to the tool change point according to the G14 setting

Examples of pattern machining

Linear hole pattern on face







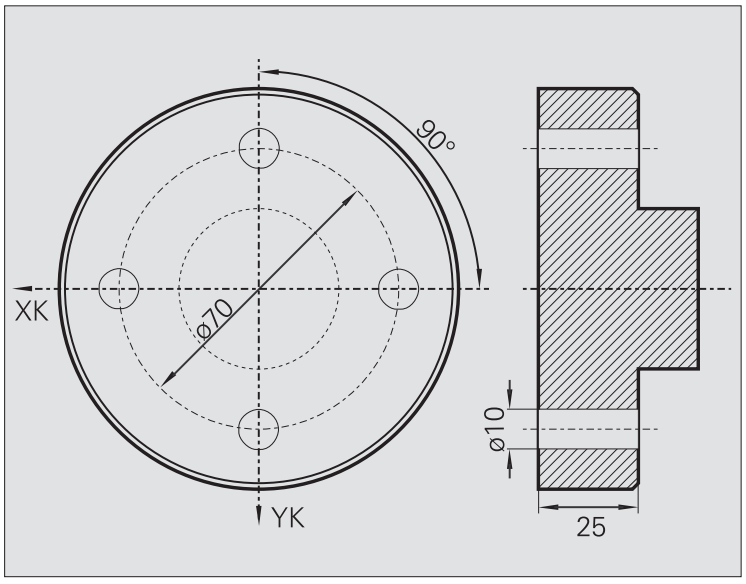
A linear hole pattern is to be machined on the face of the workpiece with the **Drilling, axial** cycle. This machining operation requires a traversable spindle and driven tools.

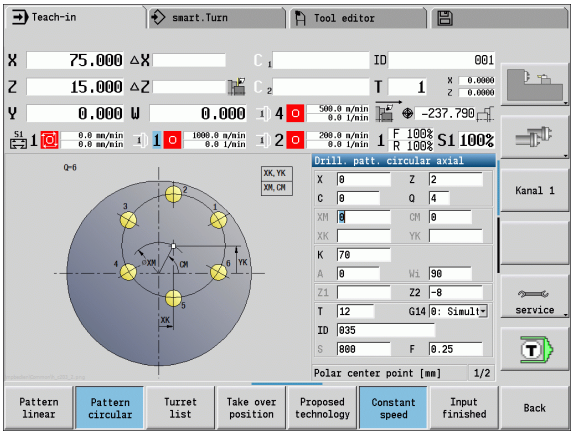
The pattern is programmed by entering the coordinates of the first and last hole, and the number of holes. Only the depth is indicated for the drilling cycle.

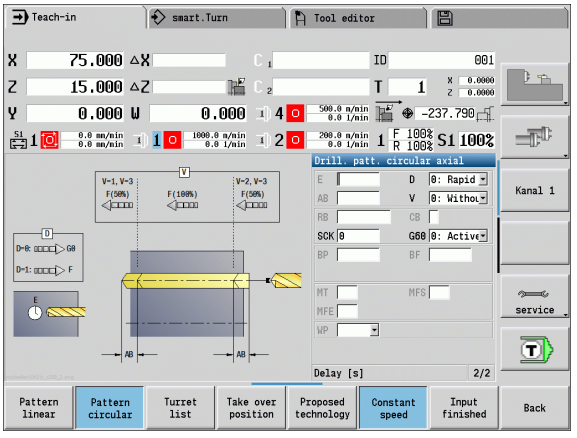
Tool data

- WO = 8 (tool orientation)
- DV = 5 (drilling diameter)
- BW = 118 (point angle)
- AW = 1 (The tool is a driven tool)

Circular hole pattern on face







A circular hole pattern is to be machined on the face of the workpiece with the **Drilling, axial** cycle. This machining operation requires a traversable spindle and driven tools.

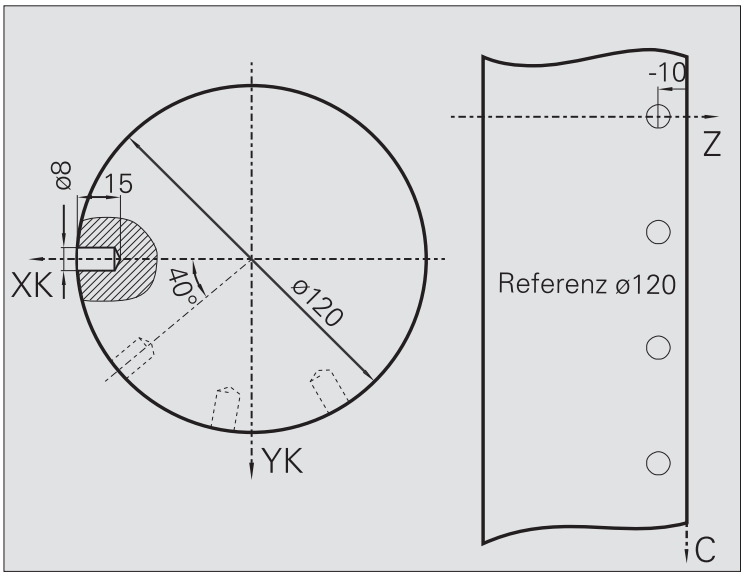
The **center of the pattern** is entered in Cartesian coordinates.

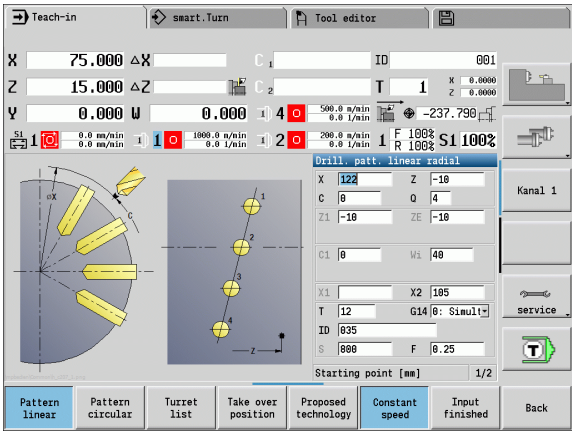
Since this example is to illustrate how you drill a through hole, the **hole end point Z2** is programmed such that the tool has to drill all the way through the workpiece before it reaches the end point. The parameters "AB" and "V" define a feed reduction for both pre-drilling and through-boring.

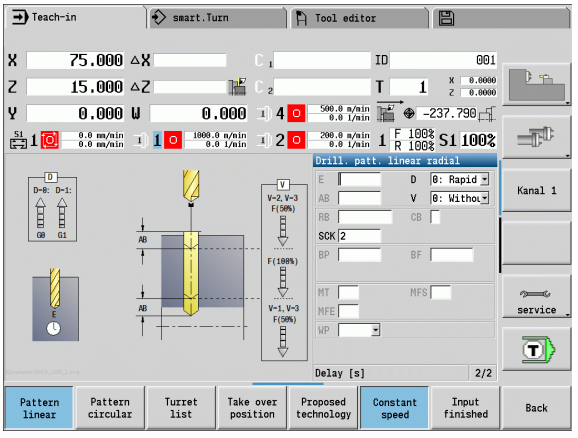
Tool data

- WO = 8 (tool orientation)
- DV = 5 (drilling diameter)
- BW = 118 (point angle)
- AW = 1 (The tool is a driven tool)

Linear hole pattern on lateral surface







A linear hole pattern is to be machined on the lateral surface of the workpiece with the **Drilling, axial** cycle. This machining operation requires a traversable spindle and driven tools.

The drilling pattern is defined by the coordinates of the first hole, the number of holes, and the spacing between the holes. Only the depth is indicated for the drilling cycle.

Tool data

- WO = 2 (tool orientation)
- DV = 8 (drilling diameter)
- BW = 118 (point angle)
- AW = 1 (The tool is a driven tool)

4.10 DIN cycles

DIN cycle



Select DIN cycle

This function allows you to select a DIN cycle (DIN subprogram) and integrate it in a cycle program. The dialogs of the parameters defined in the subprogram are then shown in the form.

The technology data that are programmed in the DIN cycle (in Manual mode, the currently active technology data) become effective as soon as you start the DIN subprogram. You can change the machine data (T, S, F) at any time by editing the DIN subprogram.

Cycle parameters

L	DIN macro number
Q	Number of repetitions (default: 1)
LA-LF	Transfer values
LH-LK	Transfer values
LO-LP	Transfer values
LR-LS	Transfer values
LU	Transfer value
LW-LZ	Transfer values
LN	Transfer value
T	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
MT	M after T: M function that is executed after the tool call T
MFS	M at beginning: M function that is executed at the beginning of the machining step
MFE	M at end: M function that is executed at the end of the machining step

Operating mode for technology database access depends on the tool type:

- Turning tool: **Roughing**
- Button tool: **Roughing**
- Threading tool: **Thread turning**
- Recessing tool: **Contour recessing**
- Twist drill: **Boring and drilling**
- Boring bar with indexable inserts: **Pilot drilling**
- Tap: **Tapping**
- Milling cutter: **Milling**

In the DIN subprogram you can assign **texts** and **help graphics** to the transfer values (see "Subprograms" Chapter in the "smart.Turn- and DIN Programming" User's Manual).

Danger of collision!

- **Cycle programming:** With DIN subprograms, the zero point shift is reset at the end of the cycle. Therefore, do not use any DIN subprograms with zero point shifts in cycle programming.
- In the DIN cycle, no starting point is defined. Please keep in mind that the tool moves on a diagonal path from the current position to the first position that is programmed in the DIN subprogram.



5

ICP programming

5.1 ICP contours

The Interactive Contour Programming (ICP) feature provides graphic support when you are defining the workpiece contours. (ICP is the abbreviation of "Interactive Contour Programming".) Contours created with ICP are used in the following:

- In **ICP cycles** (Teach-in, Manual Operation)
- In **smart.Turn**

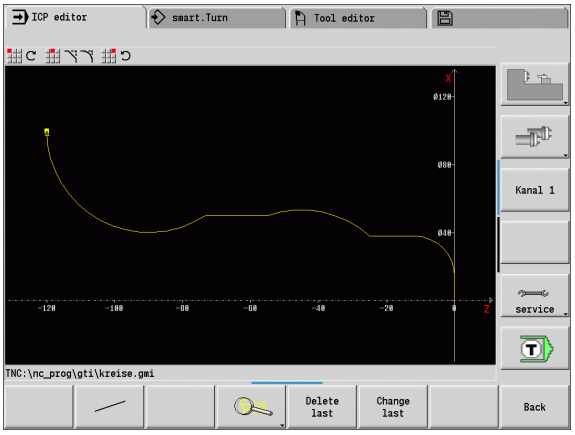
Each contour begins with a starting point. The following contours are defined using linear and circular contour elements as well as form elements like chamfers, rounding arcs, and undercuts.

ICP is called from smart.Turn and cycle dialogs.

ICP contours created in **cycle mode** are saved by the CNC PILOT in **independent files**. You can enter file names (contour names) with up to 40 characters. The ICP contours are integrated in ICP cycles. The following contour types are possible:

- Turning contours: *.gmi
- Contours of workpiece blanks: *.gmr:
- Milling contours on face: *.gms
- Milling contours on lateral surface: *.gmm

The CNC PILOT integrates ICP contours created in **smart.Turn** into the respective NC program. Contour descriptions are saved as G commands.



- In cycle mode, ICP contours created are managed in independent files. These contours are machined exclusively with ICP.
- In smart.Turn, contours are part of the NC program. They can be machined with the ICP editor **or** smart.Turn editor.

Loading contours

ICP contours that you have created **for cycle programs** can be loaded in smart.Turn. ICP converts the contours into G commands and integrates them in the smart.Turn program. The contour is now part of the smart.Turn program.

Contours that exist in **DXF format** can be imported with the ICP editor. While they are imported, the contours are converted from DXF format to ICP format. You can use DXF contours both for cycle operation and for smart.Turn.

Form elements

- **Chamfers and rounding arcs** can be inserted at each corner of the contour.
- **Undercuts** according to DIN 76, DIN 509 E, and DIN 509 F can be inserted at paraxial, orthogonal contour corners. No deviations are tolerated in elements in the X direction.

You can insert chamfers and rounding arcs at each corner of the contour. Undercuts according to DIN 76, DIN 509 E, DIN 509 F are only possible at paraxial, orthogonal contour corners. Here no deviations are tolerated in horizontal elements (X direction).

You have the following alternatives for entering form elements:

- Enter all contour elements, including the form elements, in the sequence in which they are given in the workpiece drawing.
- First define the **rough contour** without form elements. Then you superimpose the form elements (see also "Superimposing form elements" on page 378).

Machining attributes

You can assign the following attributes to the contour elements:

Parameters

U	Oversize (additive to other oversizes) The ICP generates a G52 Pxx H1.
F	Special feed rate for finishing The ICP generates a G95 Fxx.
D	Number of the additive D compensation for the finishing operation (D = 01 to 16). The ICP generates a G149 D9xx.
FP	Edit element during automatic program generation with TURN PLUS (not available in Teach-in mode) ■ 0: No ■ 1: Yes
IC	Oversize for measuring cut (not available in Teach-in mode)
KC	Length of measuring cut (not available in Teach-in mode)
HC	Measuring cut counter: Number of workpieces after which a measurement is performed (not available in Teach-in mode)



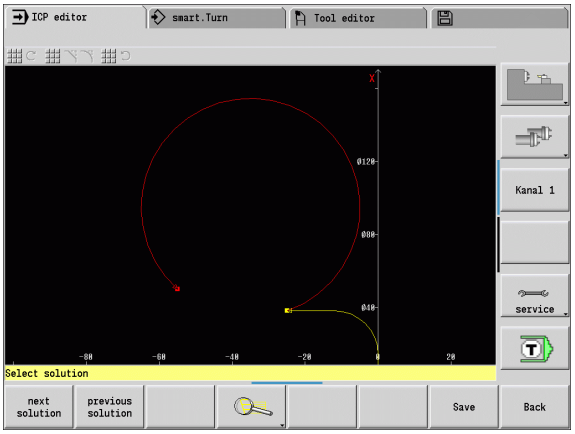
The machining attributes are effective only for the element in which the attributes have been entered in ICP.

Calculation of contour geometry

The CNC PILOT automatically calculates all missing coordinates, points of intersection, center points, etc. that can be derived mathematically.

If the entered data permit several mathematically possible solutions, you can inspect the individual solutions and select the proposal that matches the drawing.

Each **unresolved contour element** is represented by a small symbol below the graphic window. The control displays all contour elements that can be drawn, even if they are not yet fully defined.



5.2 ICP editor in cycle mode

In cycle mode you can create:

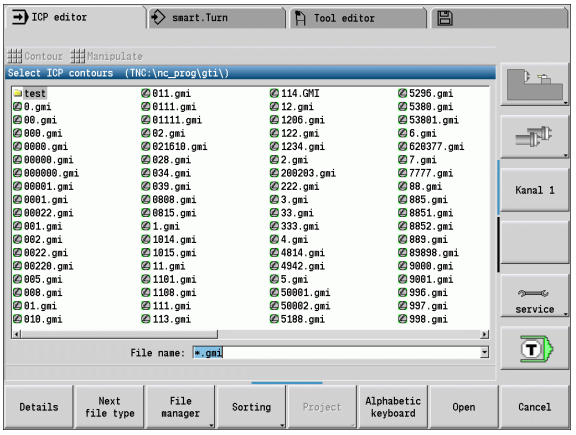
- Complex workpiece blank contours
- Contours for turning
  - For ICP turning cycles
  - For ICP recessing cycles
  - For ICP recess-turning cycles
- Complex contours for milling with the C axis
  - For the face
  - For the lateral surface

You activate the ICP editor with the **Edit ICP** soft key. This can only be selected when editing ICP turning cycles or milling cycles or the ICP workpiece blank contour cycle.

The description depends on the contour type. Using the cycle, ICP distinguishes between:

- Contour for turning or the workpiece blank contour: See "Contour elements of a turning contour" on page 387.
- Contour for the face: See "Face contours in smart.Turn" on page 413.
- Contour for the lateral surface: See "Lateral surface contours in smart.Turn" on page 421.

If you create/edit two or more ICP contours in sequence, the last edited "ICP contour number" is loaded to the cycle after the ICP editor is exited.



Editing contours for cycles

The ICP contours of the cycle editing function are assigned names. The contour name is the same as the file name. The contour name is also used in the calling cycle.

You can define the contour name in the following ways:

- Define the contour name in the cycle dialog **before** calling the ICP editor (input field **FK**). ICP loads this name.
- Define the contour name in the ICP editor. The input field **FK** has to be empty when you call the ICP editor.
- Load the existing contour. When you exit the ICP editor, the name of the contour last edited is loaded into the input field **FK**.

Creating a new contour

Edit ICP

Define the contour name in the cycle dialog and press the **Edit ICP** soft key. The ICP editor switches to entering the contour.

Edit ICP

Press the **Edit ICP** soft key. The ICP editor opens the window "Selection of ICP contours."

Open

Define the contour name in the "file name" field press the **Open** soft key. The ICP editor switches to entering the contour.



Press the **Contour** menu key.

Insert element

Press the **Insert element** soft key.

The ICP waits for you to enter a contour name.

File organization with the ICP editor

In the file organization you can copy, rename or delete the ICP contours.

Edit ICP

Press the **Edit ICP** soft key.

Contour list

Press the **Contour list** soft key. The ICP editor opens the window "Selection of ICP contours."

Organi-zation

Press the **Organization** soft key. The ICP editor switches the soft-key row to the file organization function.

5.3 ICP editor in smart.Turn

In smart.Turn you can make:

- Blank contours and auxiliary blank contours
- Finished part contours and auxiliary contours
- Standard figures and complex contours for C-axis machining
  - on the face
  - on the lateral surface
- Standard figures and complex contours for Y-axis machining
  - on the XY plane
  - on the YZ plane

Blank contours and auxiliary blank contours: You describe complex blanks element by element—like finished parts. You select the standard forms bar and tube and describe them with a few parameters (see "Defining the workpiece blank" on page 386).

Figures and patterns for C and Y axis machining: You describe complex milling contours element for element. The following standard figures are prepared. You select the figures by menu and describe them with a few parameters:

- Circle
- Rectangle
- Polygon
- Linear slot
- Circular slot
- Hole

You can arrange these figures and holes as linear or circular patterns on the face or lateral surface as well as in the XY or YZ plane.

You can import **DXF contours** and integrate them in the smart.Turn program.

You can load **cycle programming contours** and integrate them in the smart.Turn program: smart.Turn supports the loading of the following contours:

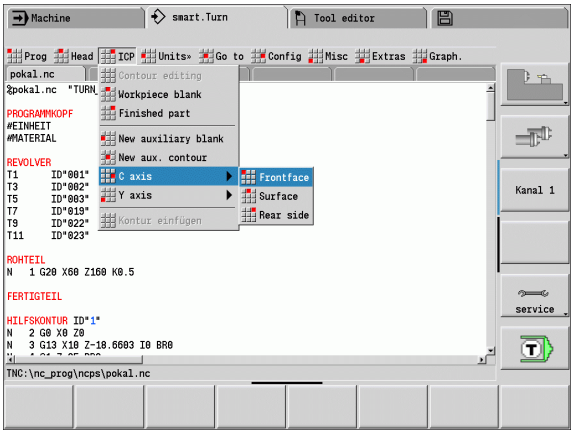
- Workpiece blank description (extension *.gmr): Loaded as blank contour or auxiliary blank contour
- Contour for turning (extension *.gmi): Loaded as finished part contour or auxiliary contour
- Face contour (extension *.gms)
- Lateral surface contour (extension *.gmm)



ICP uses G codes to describe the contours created in the smart.Turn program.

Editing a contour in smart.Turn

Creating a blank contour



Press the **ICP** menu key, then in the ICP submenu select **Blank** or **Auxiliary blank**.

Press the **Contour** menu key. The ICP editor switches to entering the complex blank contour.

Press the **Bar** menu key.

Describe the standard workpiece blank "bar."

Press the **Tube** menu key.

Describe the standard workpiece blank "tube."

Making a new contour for turning

Press the **ICP** menu key, then in the ICP submenu, select the contour type.

Press the **Contour** menu key.

Insert element

Press the **Insert element** soft key.

The ICP waits for you to enter a contour name.

Loading a contour from the cycle editing



Press the **ICP** menu key, then in the ICP submenu, select the contour type.

Contour list

Press the **Contour list** soft key. The ICP editor shows the list of the contours created in cycle mode.

Select and load the contour.

Editing an existing contour

Position the cursor in the corresponding program section.



Press the **ICP** menu key, then:



.. Select **Contour editing** in the ICP submenu.

Change ICP contour

Press the **Change ICP contour** soft key.

The ICP editor displays the selected existing contour for subsequent editing.

5.4 Creating an ICP contour

An ICP contour consists of individual contour elements. You program the contour by entering the individual contour elements one after the other in the correct sequence. The **starting point** is defined before you describe the first contour element. The **end point** is determined by the target point of the last contour element.

The contour elements / subcontours are displayed as soon as they are programmed. With the zoom and panning functions, you can adjust the graphics as required.

The principle described in the following applies for all ICP contours regardless of whether they are used for cycle programming, smart.Turn, for turning or for milling.

Soft keys in the ICP editor main menu	
Contour list	Opens the file selection dialog for ICP contours.
Turn contour	Inverts the definition direction of the contour.
	Retroactively inserts form elements.
Insert element	Adds an element to the existing contour.
Back	Returns to the dialog that has called ICP.

Entering an ICP contour

If the contour is new, the CNC PILOT asks first for the coordinates of the **contour starting point**.

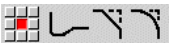
Linear contour element: Use the menu symbol to select the direction of the element and assign it a dimension. When defining horizontal and vertical linear elements, it is not necessary to enter the X and Z coordinates, respectively, provided that there are no unresolved elements.

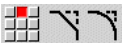
Circular contour elements: Use the menu symbol to select the direction of arc rotation and give the arc a dimension.

After selecting a contour element, you enter the known parameters. The CNC PILOT automatically calculates parameters that have not been defined from the adjoining contour elements. You usually program the contour elements with the dimensions given in the production drawing.

During input of linear or circular elements, the control does display the **`starting point`**, but it is not editable. The starting point is the end point of the preceding element.

You can toggle between the **lines** and **arcs menus** by soft key. Form elements (chamfers, rounding arcs, and undercuts) are selected with the menu key.

Menu items of the lines menu	
	Line with angle in displayed quadrant
	Horizontal line in displayed direction
	Line with angle in displayed quadrant
	Vertical line in displayed direction
	Call the form element menu

Menu items of arcs menu	
	Circular arc in the displayed direction
	Call the form element menu

ICP CONTOUR PROGRAMMING



Press the **Contour** menu key.

Press the **Insert element** soft key.

Specify the starting point.

Select the line menu.

Select the arc menu.

Select the "Form elements" menu item.

Select the element type and enter the known parameters of the contour element.

Soft keys for switching between lines menus and arcs menus

	Select the line menu.
	Select the arc menu.

Absolute or incremental dimensioning

The setting of the **Increment** soft key determines which type of coordinate is active. Incremental parameters will have the appendix "i" (Xi, Zi, etc.).

Soft key switchover for incremental

Increment	Activates the incremental dimension for the current value

Transitions between contour elements

A transition between two contour elements is called **tangential** when one contour element makes a smooth and continuous transition to the next. There is no visible kink or corner at the intersection. With geometrically complex contours, tangential transitions are useful for reducing the input of dimensional data to a minimum and eliminating the possibility of mathematically contradictory entries.

To be able to calculate unresolved contour elements, the CNC PILOT must know the type of transition that connects the contour elements. The transition to the next contour element is determined by soft key.

Soft key for tangential transition

	Activates the tangential condition for the transition in the end point of the contour element

Error messages that occur during definition of the ICP contour are often caused by "forgotten" tangential transitions.

Fits and inside threads

With the **Inside thread fit** soft key, you can display an input form for calculating the machining diameter for fits and inside threads. After entering the required values (nominal diameter and tolerance class or thread type), you can apply the calculated value as the target point for the contour element.



The machining diameter can only be calculated for suitable contour elements, e.g. for a linear element in X direction when calculating the fit for a shaft.

When calculating inside threads of thread type 9, 10 or 11, you can select the nominal diameter for inch threads in the **Nominal diameter list L**.

Calculate the fit for a hole or shaft:

- Press the **Fit** soft key
- Enter the nominal diameter
- Enter the fit data in the **Fit** form
- Press the **ENT** key to calculate the values
- Press the **Apply** soft key. The calculated center of tolerance is entered in corresponding field of the open dialog box

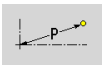
Calculate the core hole diameter for inside threads:

- Press the **Inner thread** soft key
- Enter the nominal diameter
- Enter the thread data in the **Inside thread calculator** form
- Press the **ENT** key to calculate the values
- Press the **Apply** soft key. The calculated core hole diameter is entered in corresponding field of the open dialog box

Polar coordinates

Entry of Cartesian coordinates is expected as standard. With the soft keys for polar coordinates you switch individual coordinates to polar coordinates.

You can mix Cartesian coordinates and polar coordinates to define a point.

Soft keys for polar coordinates	
	Switches the field to entering the angle **W**.
	Switches the field to entering the radius **P**.

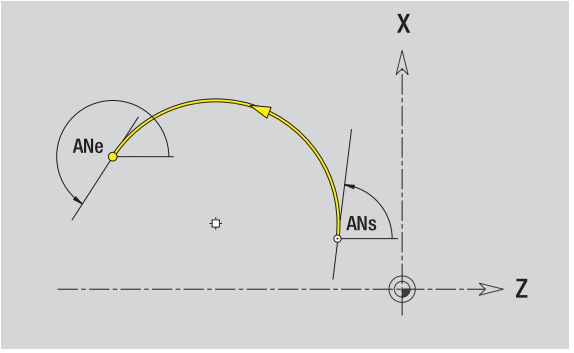
Angular input

Select the desired angle input by soft key.

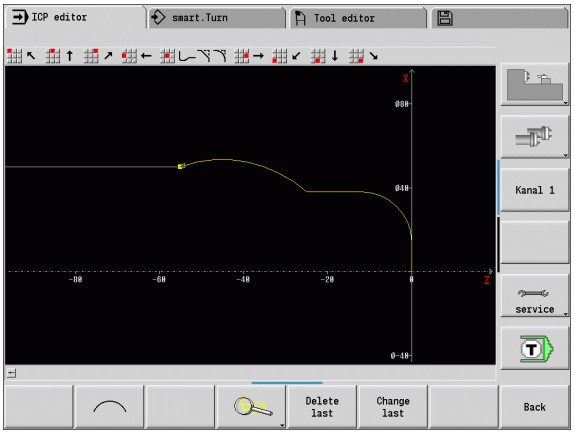
- **Linear elements**
  - **AN** Angle to the Z axis(AN<=90°—within the preselected quadrants)
  - **ANn** Angle to the following element
  - **ANp** Angle to the previous element

- **Circular arcs**
  - **ANs** Tangential angle in the starting point of the circle
  - **ANe** Tangential angle in the end point of the circle
  - **ANn** Angle to the following element
  - **ANp** Angle to the previous element

Soft keys for angular input	
	Angle to successor
	Angle to predecessor



Contour graphics



As soon as you have entered a contour element, the CNC PILOT checks whether the element is **resolved** or **unresolved**.

- A **resolved element** is a contour element that is fully and unambiguously defined. It is drawn immediately.
- An **unresolved element** has not yet been fully defined by the entered data. Use of the ICP editor:
  - It places a symbol below the graphics window. It reflects the element type and the line direction / direction of rotation.
  - Represents an unresolved linear element if the starting point and direction are known.
  - Represents an unresolved circular element as a full circle if the circle center and the radius are known.

As soon as it can be calculated, the CNC PILOT converts an unresolved contour element to a resolved contour element. The symbol is then deleted.

An incorrect contour element is displayed if possible. In addition, an error message is issued.

Unresolved contour elements: If during further contour input an error results because there is not enough information for a contour element, the unresolved element can be selected and the missing information added.

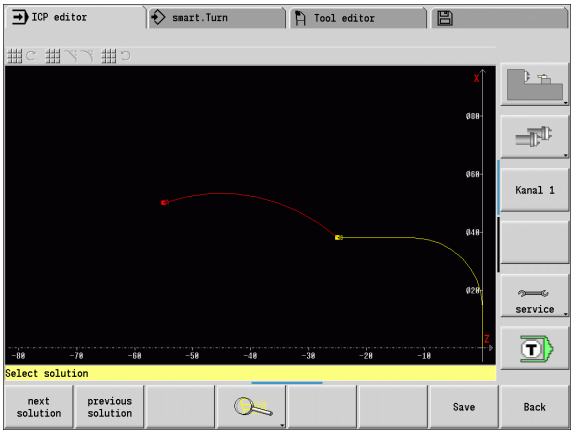
If a contour contains "unresolved" contour elements, the "resolved" elements cannot be changed. You can, however, set or delete the "tangential transition" for the contour element located directly before the unresolved contour area.

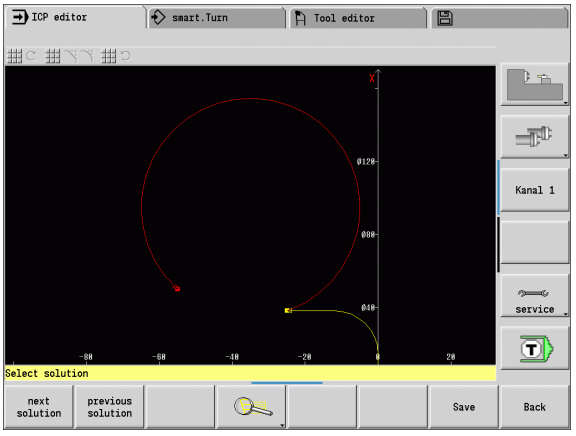
- If the element to be edited is an unresolved element, the associated symbol is marked "selected."
- The element type and the direction of rotation of a circular arc cannot be changed. In this case, the contour element has to be deleted and then added.

Selection of solutions

If the data entered for unresolved contour elements permit several possible solutions, you can check all mathematically possible solutions with the **Next solution** / **Previous solution** soft keys. You then confirm the correct solution by soft key.

If the contour still contains unsolved contour elements when you exit the editing mode, the CNC PILOT will ask you whether to discard these elements.





Colors in contour graphics

Resolved and unresolved contour elements as well as selected contour elements, selected contour corners, and remaining contours are depicted in different colors. (The selection of contour elements / contour corners and remaining contours is important when you are editing ICP contours.)

Colors:

- White: Workpiece blank contour, auxiliary blank contour
- Yellow: Finished part contours (turned contours, contour for C and Y axis machining)
- Blue: auxiliary contours
- Gray: For unresolved or incorrect but displayable elements
- Red: Selected solution, selected element or selected corner

Selection functions

In the ICP editor, the CNC PILOT provides various functions for selecting contour elements, form elements, contour corners and contour areas. You can call these functions by soft key.

Selected contour corners or contour elements are shown in **red**.

Selecting a contour area

Select the first element of the contour section.



Activate the section selection.



Press the **Next element** soft key, until the entire section is marked.



Press the **Previous element** soft key, until the entire section is marked.

Selecting contour elements



Next element (or the left arrow key) selects the next element in the direction of contour definition.



Previous element (or the right arrow key) selects the previous element in the direction of contour definition.



Select range: Activates the range selection.

Selecting contour corners (for form elements)



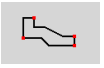
Next contour edge (or the left arrow key) selects the next corner in the direction of contour definition.



Previous contour edge (or the right arrow key) selects the previous corner in the direction of contour definition.



Mark all corners: Marks all contour corners.



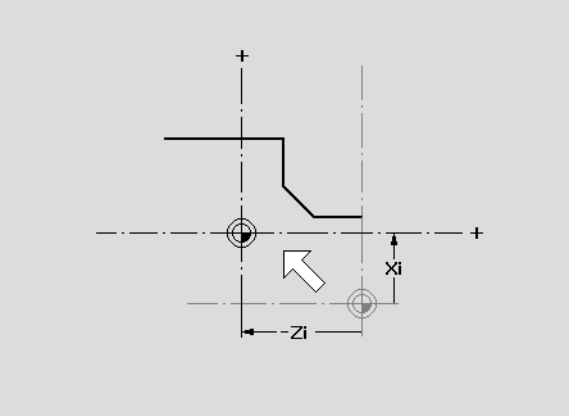
Corner selection: If the corner selecting function is activated, multiple contour corners can be marked.



Mark: If the corner selecting function is active, individual contour corners can be selected and marked or unmarked with this soft key.

Zero point shift

With this function, you can move a complete turning contour.



Activate zero point shift:

- Select "Zero point > Shift" in the finished part menu
- Enter the contour shift to move the defined contour
- Press the **Save** soft key

Deactivate zero point shift:

- Select "Zero point > Reset" in the finished part menu to reset the zero point of the coordinate system to the original position

The zero point shift cannot be reset after you exit the ICP editor. When you exit the ICP editor, the contour is recalculated with the values of the zero point shift and saved. In this case, you can only reset the zero point by a doing zero point shift in the opposite direction.

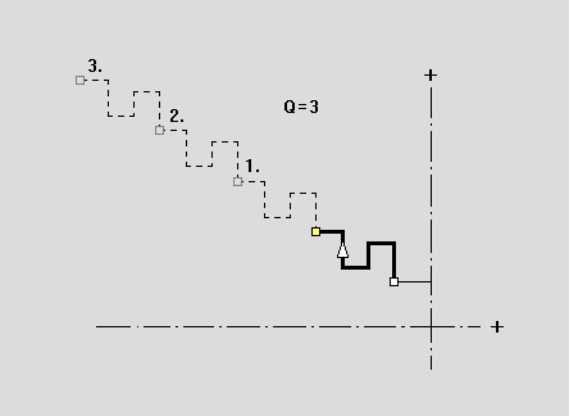
Parameters

Xi Target point – value by which the zero point is shifted

Zi Target point – value by which the zero point is shifted

Copying a contour section in linear series

With this function, you can define a contour section and append it to the existing contour.



- Select "Copy > Row Linear" in the finished part menu
- Use the **Next element** or **Previous element** soft key to select contour elements
- Press the **Select** soft key
- Enter the number of repeats
- Press the **Save** soft key

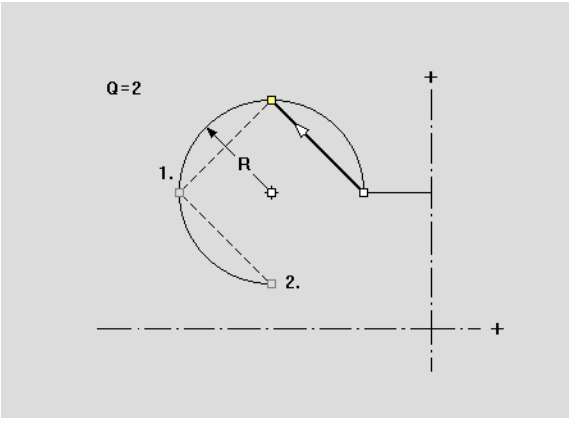
Parameter

Q Number of repeats

Copying a contour section in circular series

With this function, you can define a contour section and append it to the existing contour in a circular series.

- Select "Copy > Row Circular" in the finished part menu
- Use the **Next element** or **Previous element** soft key to select contour elements
- Press the **Select** soft key
- Enter the number of repeats and the radius
- Press the **Save** soft key



Parameters

Q Number of copies (the contour section is copied Q times)

R Radius

The control uses the entered radius for creating a circle both around the starting point and the end point of the contour section. The points of intersection of the circles are the two possible centers of rotation.

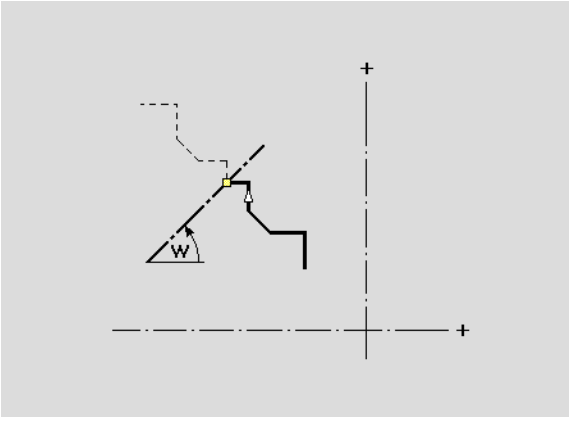
The angle of rotation is calculated from the distance between starting point and end point of the contour section.

Use the **Next solution** or **Previous solution** soft key to select one of the mathematically possible solutions.

Copying a contour section by mirroring

With this function, you can define a contour section, mirror it and add it to the existing contour.

- Select "Copy > Mirror" in the finished part menu
- Use the **Next element** or **Previous element** soft key to select contour elements
- Enter the angle of the mirror axis
- Press the **Save** soft key



Parameter

W Angle of the mirror axis. The mirror axis intersects the current end point of the contour.

Reference of the angle: positive Z axis

Inverting

With the Invert function, you can reverse the programmed direction of a contour.

Contour direction (cycle programming)

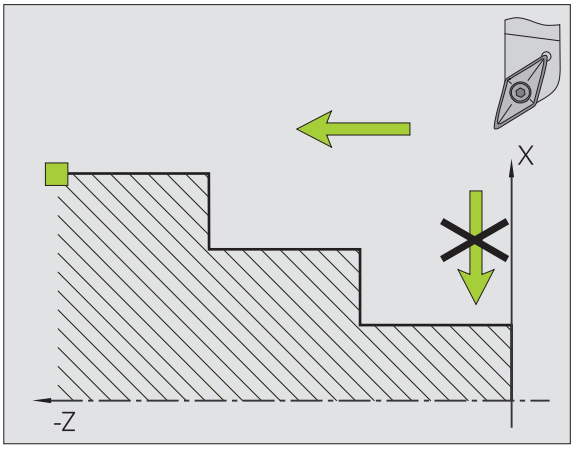
The cutting direction during cycle programming depends on the direction of the contour. If the contour is described in the **–Z direction**, a tool with the orientation 1 must be used for longitudinal machining. (See "General tool parameters" on page 497.) The cycle used determines whether machining is transverse or longitudinal.

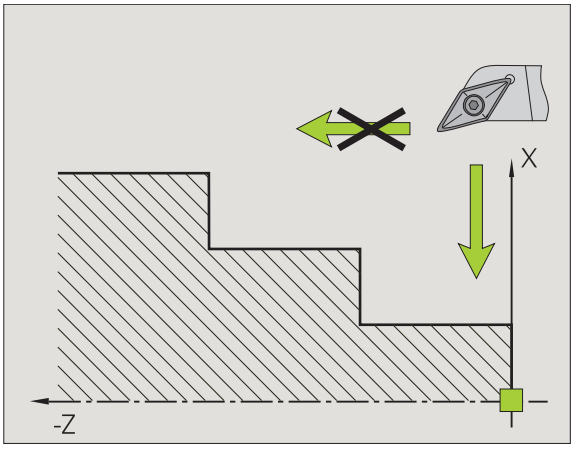
If the contour is described in the **–X direction**, a tool with the orientation 3 must be used for transverse machining.

- **ICP cut, longitudinal/transverse (roughing)**: The CNC PILOT machines the workpiece in the contour direction.
- **ICP finishing, longitudinal/transverse**: the CNC PILOT finishes in the contour direction.



An ICP contour which was defined for a roughing operation with the "ICP cut longitudinal" cycle cannot be used for machining with the "ICP cut transverse" cycle. Reverse the contour direction with the **Turn contour** soft key.





Soft keys in the ICP editor main menu

Turn contour	Inverts the definition direction of the contour.

5.5 Editing ICP contours

The CNC PILOT offers the following possibilities for extending or changing a programmed contour.

Superimposing form elements



Press the soft key.



Select the desired **form element**.





Select the corner to be changed.

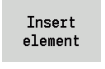


Confirm the corners for the form element and enter the **data** for the form element.



Adding contour elements

You can **add to** an ICP contour by entering additional contour elements that are "appended" to the existing contour. A small square indicates the end of the contour and an arrow indicates the direction.



Press the soft key

"Append" additional contour elements to the existing contour.

Editing or deleting the last contour element

To edit the last contour element: When the **Change last** soft key is pressed the data of the "last" contour element are presented for editing.

Depending on the adjoining contour elements, corrections of linear or circular elements are either transferred immediately or the corrected contour is displayed for inspection. ICP highlights the affected contour elements in color. If the change permits several possible solutions, you can check all mathematically possible solutions with the **Next solution** / **Previous solution** soft keys.

The change will not become effective until you confirm by soft key. If you discard the change, the old description becomes effective again.

The type of contour element (linear or circular element), the direction of a linear element, or the direction of rotation of a circular element cannot be changed. Should this be necessary, you must delete the element and add a new contour element.

To delete the last contour element: When the **delete last** soft key is pressed the data of the "last" contour element are discarded. You can use this function repeatedly to delete several successive contour elements.

Deleting a contour element



Press **Manipulate**. The menu displays functions for trimming, editing and deleting contours.



Menu item **Delete ...**



Select **... Element/range**.





Select the contour element to be deleted.



Delete the contour element.

You can delete several successive contour elements.

Editing contour elements

The CNC PILOT provides various ways to change an existing contour. The procedure is illustrated in the following example of editing the length of an element. The other functions work similar to the procedure described here.

The **Manipulate** menu provides the following functions for editing existing contour elements:

- **Trimming**
  - Length of element
  - Length of contour (only for closed contours)
  - Radius
  - Diameter
- **Change**
  - Contour element
  - Form element
- **Delete**
  - Element/Range
  - Move element/range as well
  - Contour/Pocket/Figure/Pattern
  - Form element
  - All form elements
- **Transform**
  - Shifting a contour
  - Contour rotation
  - Contour mirroring: You can define the position of the mirror axis by entering the starting point and end point or the starting point and angle

Changing the length of the contour element

Press **Manipulate**. The menu displays functions for trimming, editing and deleting contours.

Menu item **Edit ...**

... Select the **contour element**.

Select the contour element to be edited.

Present the selected contour element for editing.

Make the changes.

Load the changes.

Save

The contour or, if applicable, the possible solutions are displayed for inspection. Changes in form elements and unresolved elements are loaded immediately (original contour in yellow, changed contour in red for comparison).

Load the desired solution.

Changing the paraxial line

When changing a paraxial line, an additional soft key is offered with which you can change the second end point as well. From an originally straight line you can make a diagonal in order to make corrections.

Changing the "fixed" end point. By pressing repeatedly you select the direction of the diagonal.

Shifting a contour

Press **Manipulate**. The menu displays functions for trimming, editing and deleting contours.

Menu item **Edit ...**

... Select the **contour element**.

Select the contour element to be edited.

Present the selected contour element for moving.

Enter the new starting point of the reference element.

Over-write

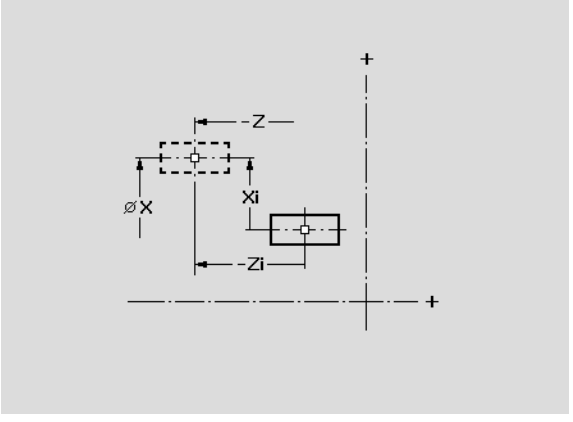
Assume the new starting point (= new position). The CNC PILOT shows the shifted contour.

Over-write

Confirm the new position for the contour.

Transformations – Shifting

With this function, you can move a contour by entering incremental or absolute coordinates.

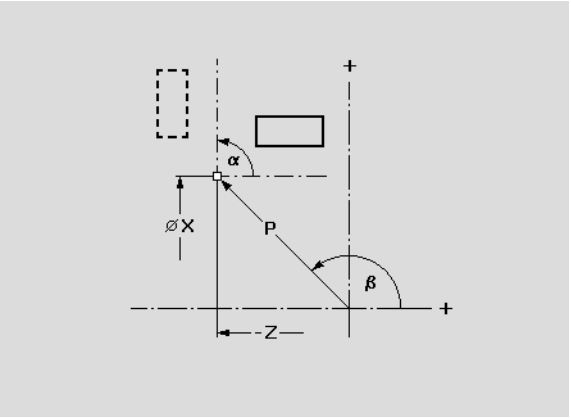


Parameters

X	Target point
Z	Target point
Xi	Target point – incremental
Zi	Target point – incremental
H	Original (only with C-axis contours):
	■ 0: Delete: Original contour is deleted
	■ 1: Copy: Original contour is maintained
ID	Contour name (only with C-axis contours)

Transformations – Rotating

With this function you can rotate a contour about a center of rotation.



Parameters

X Center of rotation in Cartesian coordinates

Z Center of rotation in Cartesian coordinates

W Center of rotation in polar coordinates

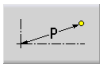
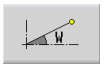
P Center of rotation in polar coordinates

A Angle of rotation

H Original (only with C-axis contours):

- 0: Delete: Original contour is deleted
- 1: Copy: Original contour is maintained

ID Contour name (only with C-axis contours)

Soft keys	
	Polar dimension of the center of rotation: Angle
	Polar dimension of the center of rotation: Radius

Transformations – mirroring

This function mirrors the contour. Define the position of the **mirror axis** by entering the starting point and end point or the starting point and angle.

Parameter

XS Starting point in Cartesian coordinates

ZS Starting point in Cartesian coordinates

X End point in Cartesian coordinates

Z End point in Cartesian coordinates

A Angle of rotation

WS Starting point in polar coordinates

PS Starting point in polar coordinates

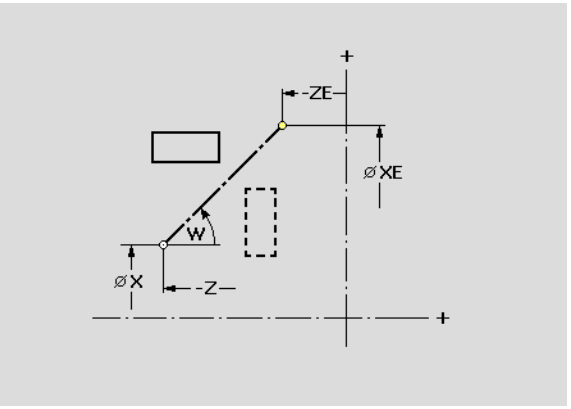
W End point in polar coordinates

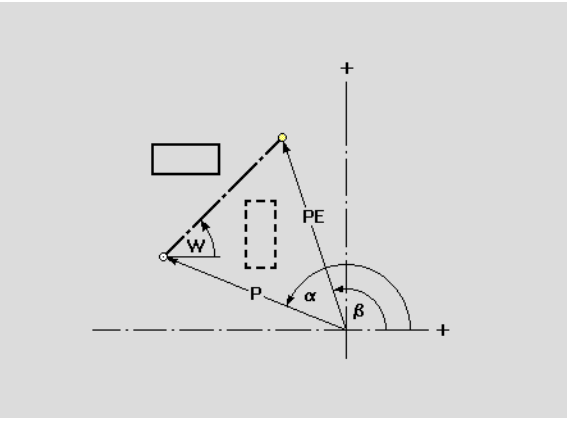
P End point in polar coordinates

H Original (only with C-axis contours):

- 1: Copy: Original contour is maintained
- 0: Delete: Original contour is deleted

ID Contour name (only with C-axis contours)





Soft keys for polar dimensions	
WS	Polar dimension of the starting point: Angle
PS	Polar dimension of the starting point: Radius
P	Polar dimension of the end point: Angle
W	Polar dimension of the end point: Radius

5.6 The zoom function in the ICP editor

The zoom functions make it possible to change the visible section by using **soft keys**, the **arrow keys**, and the **PgUp** and **PgDn** keys. The zoom function can be called in all ICP windows.

The CNC PILOT sizes the graphic section depending on the programmed contour. With the zoom function you can select another graphic section.

Changing the view

Modifying the section by using keys

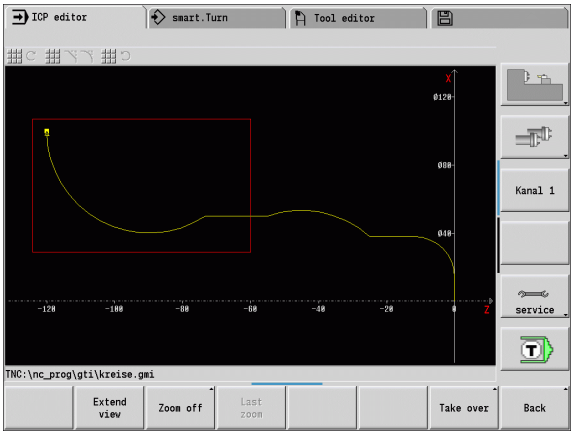
- The displayed section can be modified with the **arrow keys**, the **PgUp** key and the **PgDn** key without opening the zoom menu.

Keys for modifying the displayed section	
↓ ↑ ← →	The arrow keys move the workpiece in the indicated direction.
PG DN	Reduces the displayed workpiece (zoom –).
PG UP	Magnifies the displayed workpiece (zoom +).

Modifying the section with the zoom menu

- If the zoom function is selected, a red rectangle appears in the contour window. This red frame indicates the zoom area, which you can select using the **Take over** soft key or the **Enter** key. Use the following keys to change the frame size and position:

Keys for modifying the red frame	
↓ ↑ ← →	The arrow keys move the frame in the indicated direction.
PG DN	Reduces the displayed frame (zoom +).
PG UP	Enlarges the displayed frame (zoom -).



Soft keys in the zoom menu	
	Activates the zoom function.
Extend view	Extends the displayed section directly (zoom –).
Zoom off	Switches back to the standard screen section and closes the zoom menu.
Last zoom	Switches to the last screen section selected.
Take over	Applies the area selected by the red frame as the new section and closes the zoom menu.
Back	Closes the zoom menu without changing the selected section.

5.7 Defining the workpiece blank

In smart.Turn, the standard forms "bar" and "tube" are described with a G function.

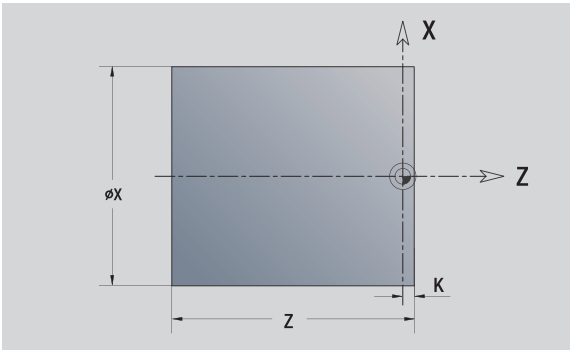
"Bar" blank

The function describes a cylinder.

Parameters

X Cylinder diameter
Z Length of the blank
K Right edge (distance between workpiece zero point and right edge)

ICP generates in smart.Turn a G20 in the BLANK section.



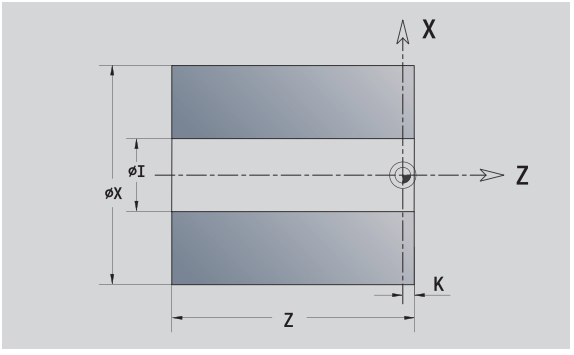
"Tube" blank

The function describes a hollow cylinder.

Parameters

X Diameter of hollow cylinder
Z Length of the blank
K Right edge (distance between workpiece zero point and right edge)
I Inside diameter

ICP generates in smart.Turn a G20 in the BLANK section.



5.8 Contour elements of a turning contour

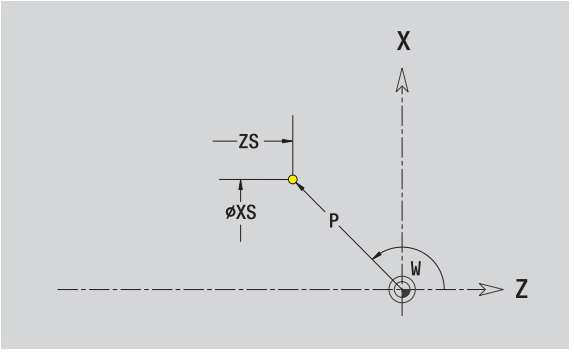
With the "contour elements of a turning contour" you can create the following:

- In cycle mode
  - Complex workpiece blank contours
  - Contours for turning
- In smart.Turn
  - Complex blank contours and auxiliary blank contours
  - Finished part contours and auxiliary contours

Basic elements of a turning contour

Specify the starting point

Enter the coordinates for the starting point and target point in the first contour element of the turning contour. Entering the starting point is only possible in the first contour element. In subsequent contour elements, the starting point results from the previous contour element in each case.



Contour

Press the **Contour** menu key.

Insert element

Press the **Insert element** soft key.

Select the contour element

Parameters for defining the starting point

XS, ZS	Starting point of contour
W	Polar starting point of contour (angle)
P	Polar starting point of contour (radius)

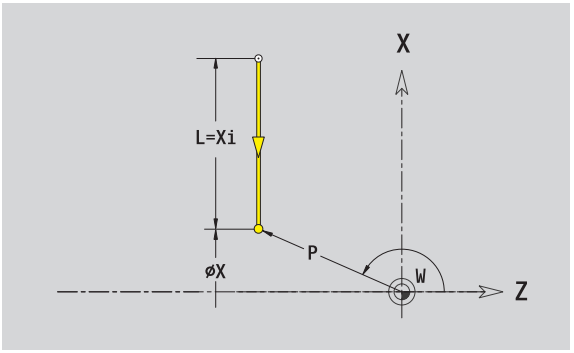
In smart.Turn, ICP generates a G0.

Vertical lines



Select the line direction.

Enter the line dimensions and define the transition to the next element.



Parameters

X	Target point
Xi	Incremental target point (distance from starting point to target point)
W	Polar target point (angle)
P	Polar target point (radius)
L	Length of line

U, F, D, FP, IC, KC, HC: See machining attributes on Page 361

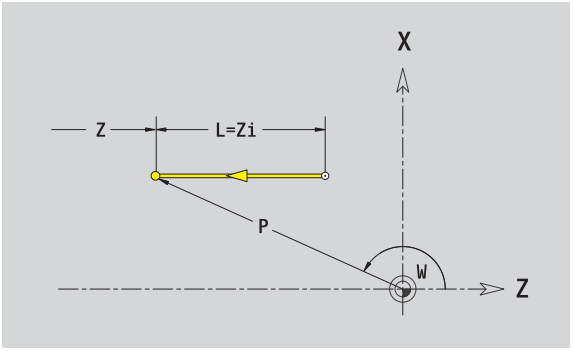
In smart.Turn, ICP generates a G1.

Horizontal lines



Select the line direction.

Enter the line dimensions and define the transition to the next contour element.



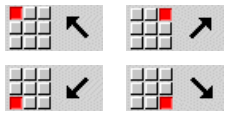
Parameters

Z	Target point
Zi	Incremental target point (distance from starting point to target point)
W	Polar target point (angle)
P	Polar target point (radius)
L	Length of line

U, F, D, FP, IC, KC, HC: See machining attributes on Page 361

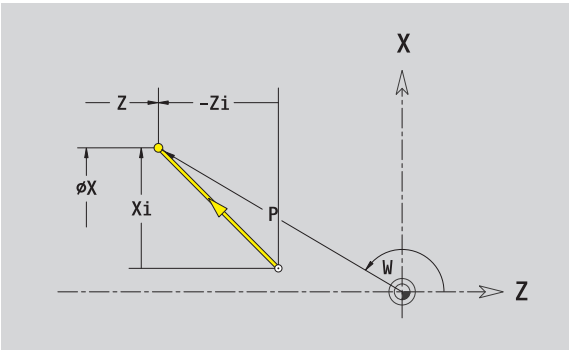
In smart.Turn, ICP generates a G1.

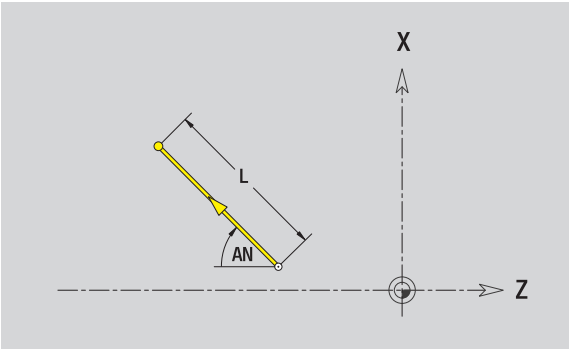
Line at angle



Select the line direction.

Enter the line dimensions and define the transition to the next element. Always enter the **angle AN** (<=90°) within the selected quadrant.





Parameters

X, Z	Target point
Xi, Zi:	Incremental target point (distance from starting point to target point)
W	Polar target point (angle)
P	Polar target point (radius)
L	Length of the line
AN	Angle to Z axis
ANn	Angle to the subsequent element
ANp	Angle to the previous element

U, F, D, FP, IC, KC, HC: See machining attributes on Page 361

In smart.Turn, ICP generates a G1.

Circular arc



Select the arc's direction of rotation.

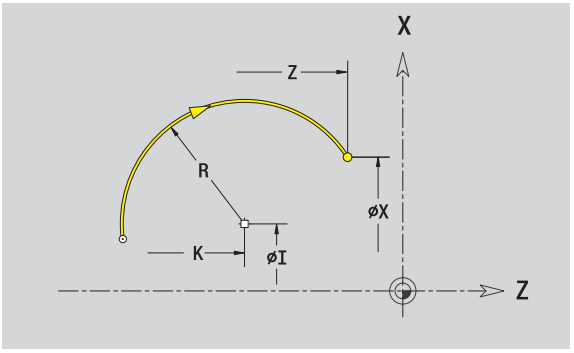
Enter the arc dimensions and define the transition to the next element.

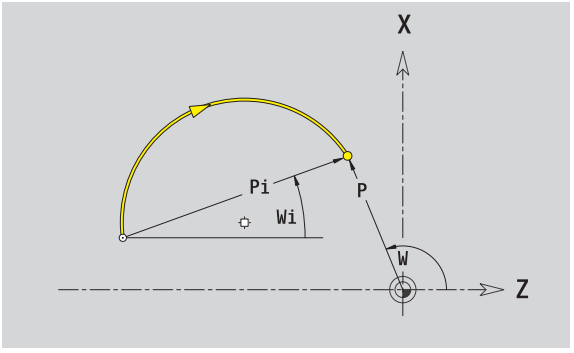
Parameters

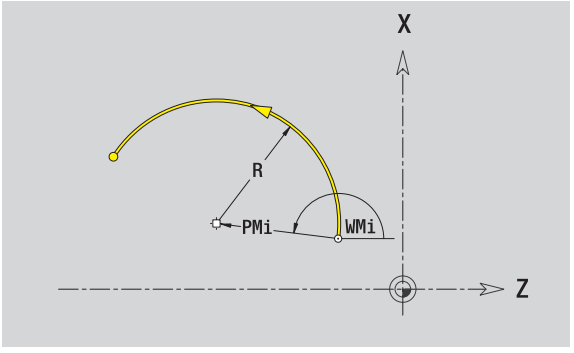
X, Z	Target point (end point of the circular arc)
Xi, Zi:	Incremental target point (distance from starting point to target point)
W	Polar target point (angle)
Wi	Incremental polar target point and angle (relative to the starting point)
P	Polar target point (radius)
Pi	Incremental polar target point (distance from starting point to target point)
I, K	Center of arc
Ii, Ki	Incremental center of arc (distance from starting point to center in X, Z direction)
PM	Polar center of arc (radius)
PMi	Incremental polar center (distance from starting point to center)
WM	Polar center of arc (angle)
WMi	Incremental polar center and angle (relative to the starting point)
R	Radius
ANs	Tangential angle in the starting point
ANe	Tangential angle in the target point
ANp	Angle to the previous element
ANn	Angle to the subsequent element

U, F, D, FP: See machining attributes Page 361

ICP generates a G2 or G3 in smart.Turn.

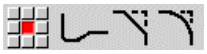






Contour form elements

Chamfer/rounding arc



Select the form elements.





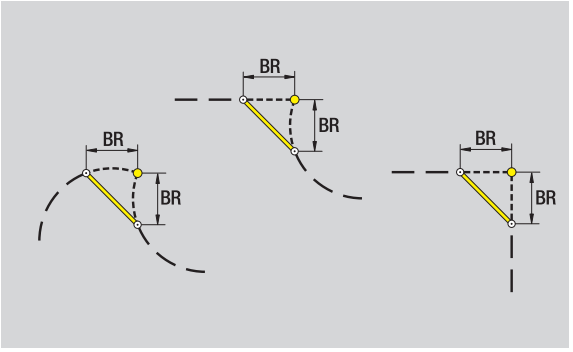
Select a chamfer.

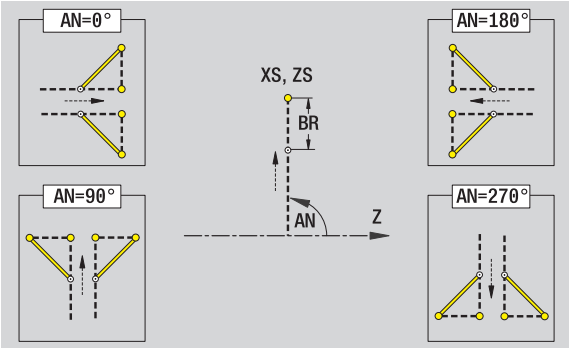


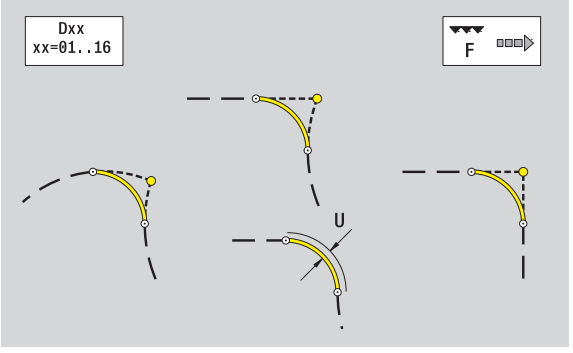
Select rounding arc.

Enter the chamfer width BR or the **rounding radius BR**.

Chamfer/rounding arc as first element: Enter **element position AN**.







Parameters

BR Chamfer width/rounding radius
AN Element position
U, F, D, FP: See machining attributes Page 361

Chamfers/rounding arcs are defined on contour corners. A "contour corner" is the point of intersection between the approaching and departing contour elements. A chamfer or rounding cannot be calculated until the departing contour element is known.

ICP integrates the chamfer/rounding arc in smart.Turn on the basic element G1, G2 or G3.

Contour begins with a chamfer or a rounding arc: Enter the position of the intended corner as starting point. Then, in the form element menu, select chamfer or rounding arc. Since the introducing contour element is missing, you enter the **element position AN** to clearly define the position of the chamfer or rounding arc.

Example of an outside chamfer at start of contour: If you program "element position AN=90°," the imaginary approaching reference element is a transverse element in the positive **+X axis direction** (see figure).

ICP converts a chamfer or rounding arc at the start of the contour to a linear or circular element.

Thread undercut DIN 76

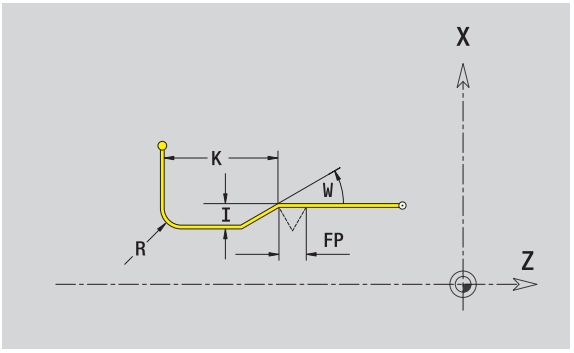


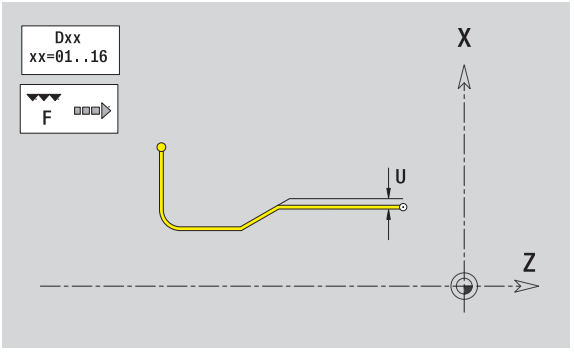
Select the form elements.



Select Undercut DIN 76.

Enter the undercut parameters.





Parameters

FP	Thread pitch (default: value from standard table)
I	Undercut depth (radius) (default: value from standard table)
K	Undercut length (default: value from standard table)
R	Undercut radius (default: value from standard table)
W	Undercut angle (default: value from standard table)

U, F, D, FP: See machining attributes Page 361

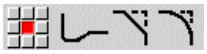
In smart.Turn, ICP generates a G25.

Parameters that are not defined are automatically calculated by the CNC PILOT from the standard table (see "DIN 76—undercut parameters" on page 579):

- The "thread pitch FP" is determined from the diameter.
- The parameters I, K, W, and R are calculated from the thread pitch FP.

- If you are programming an internal thread, it is advisable to preset the **thread pitch FP**, since the diameter of the longitudinal element is not the thread diameter. If you have the CNC PILOT calculate the thread pitch automatically, slight deviations may occur.
- Undercuts can be programmed only between two linear elements. One of the two linear elements must be parallel to the X axis.

Undercut DIN 509 E

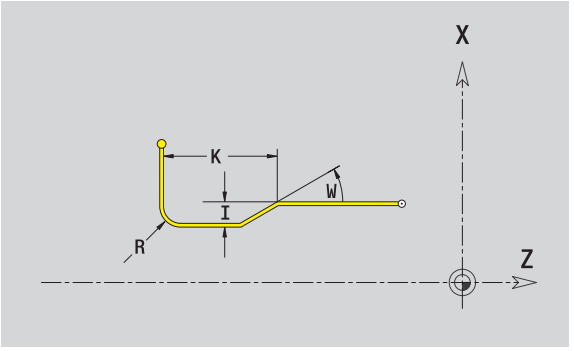


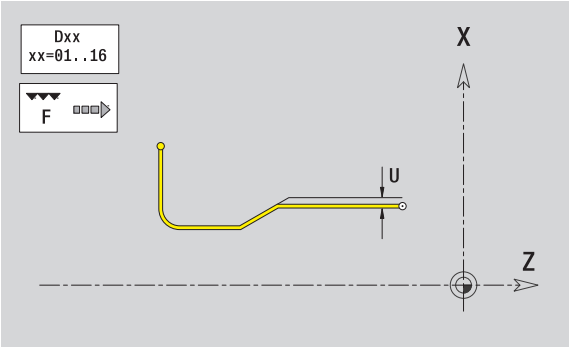
Select the form elements.



Select undercut DIN 509 E.

Enter the undercut parameters.





Parameters

I Undercut depth (radius) (default: value from standard table)

K Undercut length (default: value from standard table)

R Undercut radius (default: value from standard table)

W Undercut angle (default: value from standard table)

U, F, D, FP: See machining attributes Page 361

In smart.Turn, ICP generates a G25.

The CNC PILOT calculates unentered parameters automatically from the diameter in the standard table (see "DIN 509 E – undercut parameters" on page 581).

Undercuts can be programmed only between two linear elements. One of the two linear elements must be parallel to the X axis.

Undercut DIN 509 F

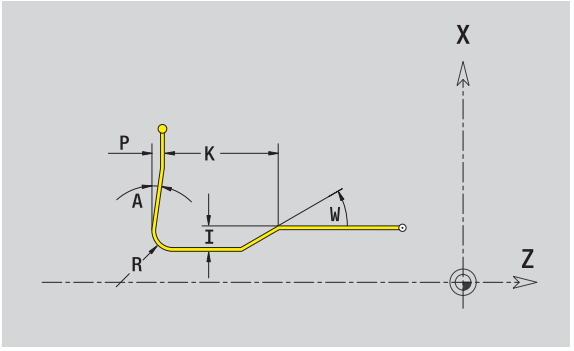


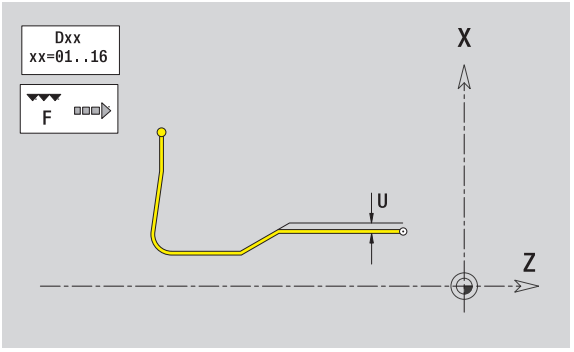
Select the form elements.



Select undercut DIN 509 F.

Enter the undercut parameters.





Parameters

I	Undercut depth (radius) (default: value from standard table)
K	Undercut length (default: value from standard table)
R	Undercut radius (default: value from standard table)
W	Undercut angle (default: value from standard table)
P	Face depth (default: value from standard table)
A	Face angle (default: value from standard table)

U, F, D, FP: See machining attributes Page 361

In smart.Turn, ICP generates a G25.

The CNC PILOT calculates unentered parameters automatically from the diameter in the standard table (see "DIN 509 F – undercut parameters" on page 581).



Undercuts can be programmed only between two linear elements. One of the two linear elements must be parallel to the X axis.

Undercut type U

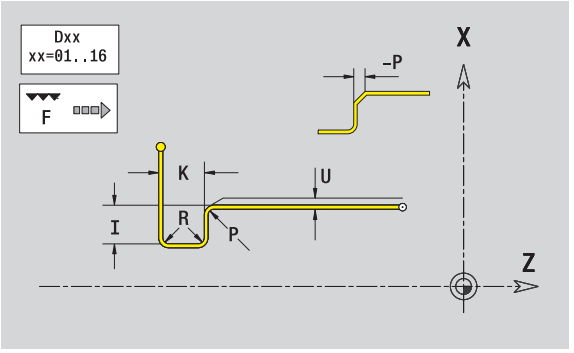


Select the form elements.



Select the undercut type U.

Enter the undercut parameters.



Parameters

I Undercut depth (radius)
K Undercut length
R Undercut radius
P Chamfer/rounding
U, F, D, FP: See machining attributes Page 361

In smart.Turn, ICP generates a G25.

Undercuts can be programmed only between two linear elements. One of the two linear elements must be parallel to the X axis.

Undercut type H

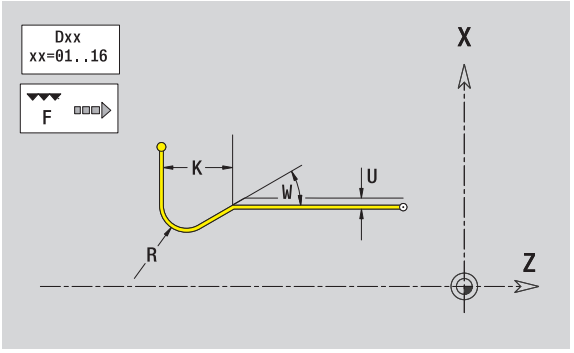


Select the form elements.



Select the undercut type H.

Enter the undercut parameters.



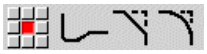
Parameters

K Undercut length
R Undercut radius
W Plunging angle
U, F, D, FP: See machining attributes Page 361

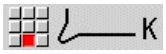
In smart.Turn, ICP generates a G25.

Undercuts can be programmed only between two linear elements. One of the two linear elements must be parallel to the X axis.

Undercut type K

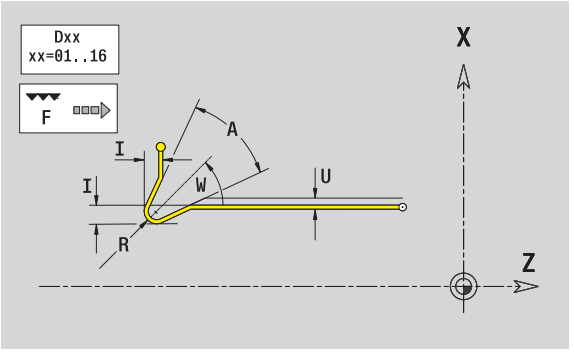


Select the form elements.



Select the undercut type K.

Enter the undercut parameters.



Parameters

I	Undercut depth
R	Undercut radius
W	Angular length
A	Plunging angle

U, F, D, FP: See machining attributes Page 361

In smart.Turn, ICP generates a G25.

Undercuts can be programmed only between two linear elements. One of the two linear elements must be parallel to the X axis.

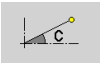
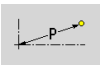
5.9 Contour elements on face

With the "contour elements of a face" you can create complex milling contours.

- Cycle mode: Contour for axial ICP milling cycles
- smart.Turn: Contour for machining with the C axis

Enter the dimensions of the front face contour elements in Cartesian or polar values. You can switch between them by pressing a soft key (see table). You can mix Cartesian coordinates and polar coordinates to define a point.

Soft keys for polar coordinates

	Switches the field to entering the angle **C**.
	Switches the field to entering the radius **P**.

Starting point of face contour

Enter the coordinates for the starting point and target point in the first contour element of the contour. Entering the starting point is only possible in the first contour element. In subsequent contour elements, the starting point results from the previous contour element in each case.

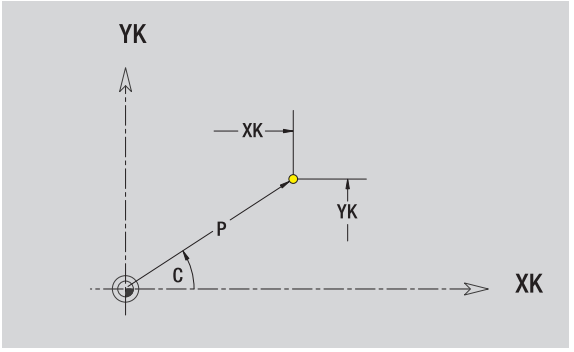


Press the **Contour** menu key.



Press the **Insert element** soft key.

Specify the starting point.



Parameters for defining the starting point

XKS, YKS	Starting point of contour
C	Polar starting point of contour (angle)
P	Polar starting point of contour (radius)
HC	Milling/drilling attribute: ■ 1: Contour milling ■ 2: Pocket milling ■ 3: Area milling ■ 4: Deburring ■ 5: Engraving ■ 6: Contour milling and deburring ■ 7: Pocket milling and deburring ■ 14: Do not machine
QF	Milling location: ■ 0: On the contour ■ 1: Inside/left ■ 2: Outside/right

HF Direction:
- 0: Up-cut milling
- 1: Climb milling

DF Cutter diameter
WF Angle of the chamfer
BR Chamfer width
RB Return plane

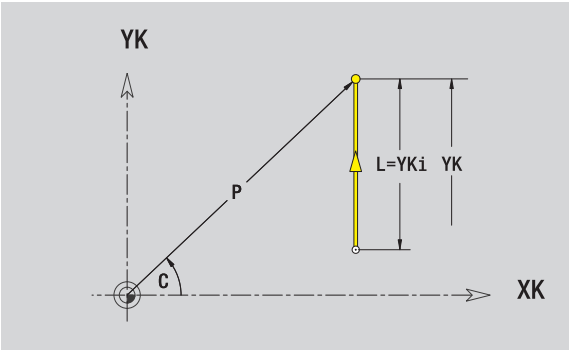
In smart.Turn, ICP generates a G100.

Vertical lines on face



Select the line direction.

Enter the line dimensions and define the transition to the next contour element.

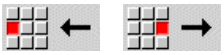


Parameters

YK Cartesian target point
YKi Incremental target point (distance from starting point to target point)
C Polar target point (angle)
P Polar target point
L Length of line
F: See machining attributes Page 361

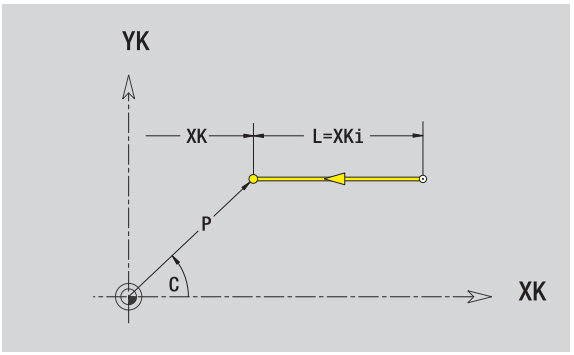
In smart.Turn, ICP generates a G101.

Horizontal lines on face



Select the line direction.

Enter the line dimensions and define the transition to the next contour element.



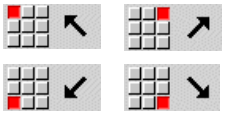
Parameters

XK	Cartesian target point
XKi	Incremental target point (distance from starting point to target point)
C	Polar target point (angle)
P	Polar target point
L	Length of line

F: See machining attributes Page 361

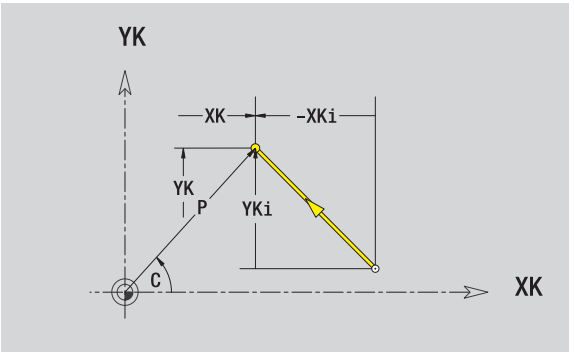
In smart.Turn, ICP generates a G101.

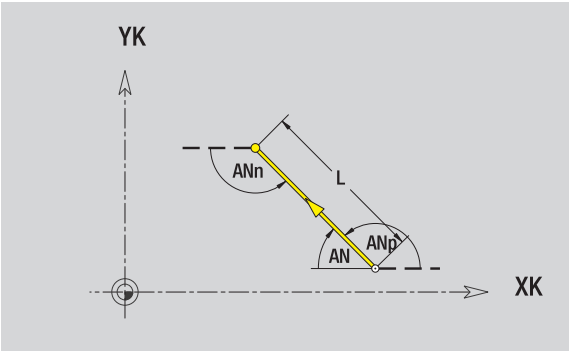
Line at angle on face



Select the line direction.

Enter the line dimensions and define the transition to the next contour element.





Parameters

XK, YK	Cartesian target point
XKi, YKi	Incremental target point (distance from starting point to target point)
C	Polar target point (angle)
P	Polar target point
AN	Angle to XK axis (for direction of angle, see help graphic)
L	Length of line
ANn	Angle to the subsequent element
ANp	Angle to the previous element

F: See machining attributes Page 361

In smart.Turn, ICP generates a G101.

Circular arc on face



Select the arc's direction of rotation.

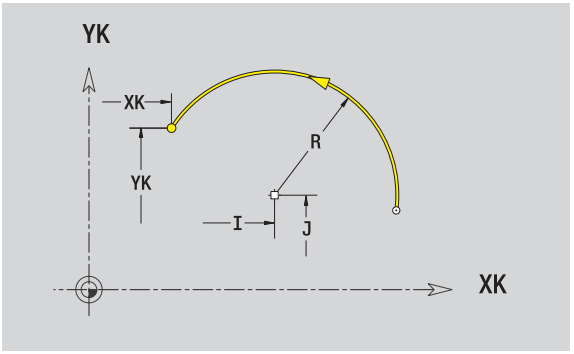
Enter the arc dimensions and define the transition to the next element.

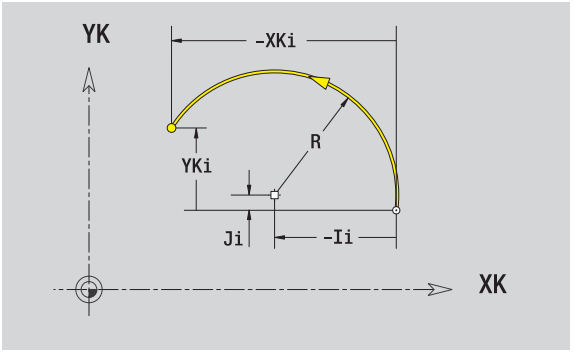
Parameters

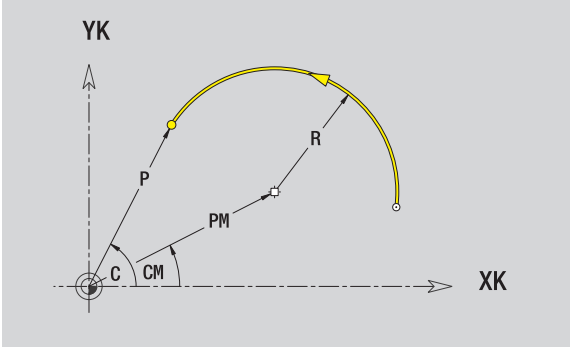
XK, YK	Target point (end point of the circular arc)
XKi, YKi	Incremental target point (distance from starting point to target point)
P	Polar target point (radius)
Pi	Incremental polar target point (distance from starting point to target point)
C	Polar target point (angle)
Ci	Incremental polar target point and angle (relative to the starting point)
I, J	Center of arc
Ii, Ji:	Incremental center of arc (distance from starting point to center in X, Z)
PM	Polar center of arc
PMi	Incremental polar center (distance from starting point to center)
CM	Polar center of arc (angle)
CMi	Incremental polar center and angle (relative to the starting point)
R	Radius
ANs	Tangential angle in the starting point
ANe	Tangential angle in the target point
ANp	Angle to the previous element
ANn	Angle to the subsequent element

F: See machining attributes Page 361

ICP generates a G102 or G103 in smart.Turn.







Chamfer/rounding arc on the face



Select the form elements.



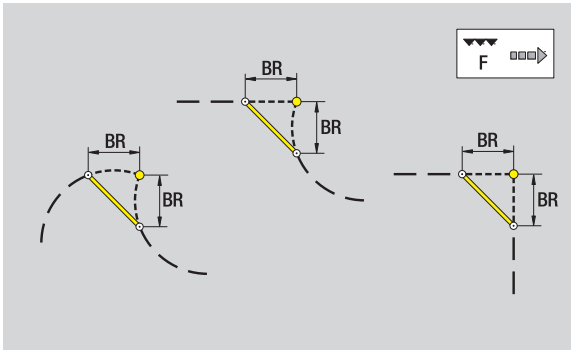
Select a chamfer.

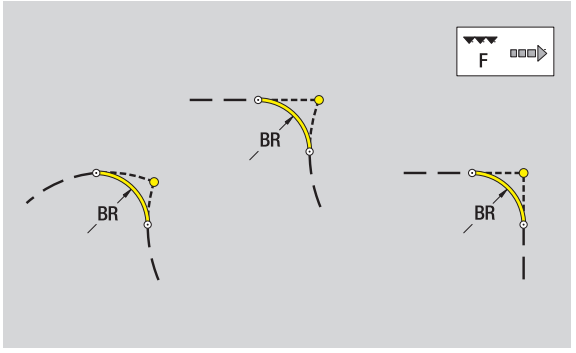


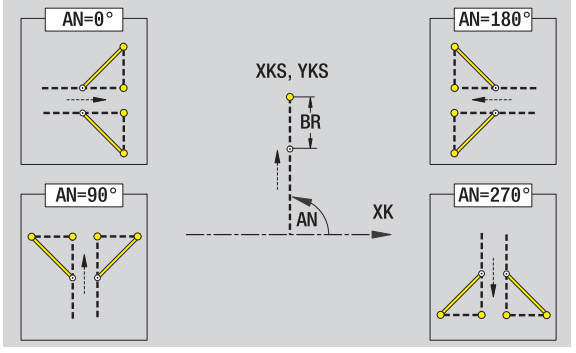
Select rounding arc.

Enter the chamfer width BR or the **rounding radius BR**.

Chamfer/rounding arc as first element: Enter **element position AN**.







Parameters

BR Chamfer width/rounding radius

AN Element position

F: See machining attributes Page 361

Chamfers/rounding arcs are defined on contour corners. A "contour corner" is the point of intersection between the approaching and departing contour elements. A chamfer or rounding cannot be calculated until the departing contour element is known.

ICP integrates the chamfer/rounding arc in smart.Turn on the basic element G101, G102 or G103.

Contour begins with a chamfer or a rounding arc: Enter the position of the intended corner as starting point. Then, in the form element menu, select chamfer or rounding arc. Since the introducing contour element is missing, you enter the **element position AN** to clearly define the position of the chamfer or rounding arc.

ICP converts a chamfer or rounding arc at the start of the contour to a linear or circular element.

5.10 Contour elements on lateral surface

With the "contour elements of a lateral surface" you can create complex milling contours.

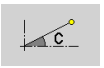
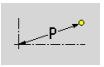
- Cycle mode: Contour for radial ICP milling cycles
- smart.Turn: Contour for machining with the C axis

Enter the dimensions of the lateral surface contour elements in Cartesian or polar values. You can use the linear dimension as an alternative to the angular dimension. You can switch between them by pressing a soft key (see table).

The **linear dimension** is essentially the unrolled surface at the reference diameter.

- For lateral surface contours, the reference diameter is specified in the cycle. The linear dimensions of all subsequent contour elements are referenced to this diameter.
- When called from smart.Turn, the reference diameter in the reference data is requested.

Soft keys for polar coordinates

	Switches the field from linear dimensions to entry of the angle **C**.
	Switches the field to entering the polar dimension **P**.

Starting point of lateral surface contour

Enter the coordinates for the starting point and target point in the first contour element of the contour. Entering the starting point is only possible in the first contour element. In subsequent contour elements, the starting point results from the previous contour element in each case.

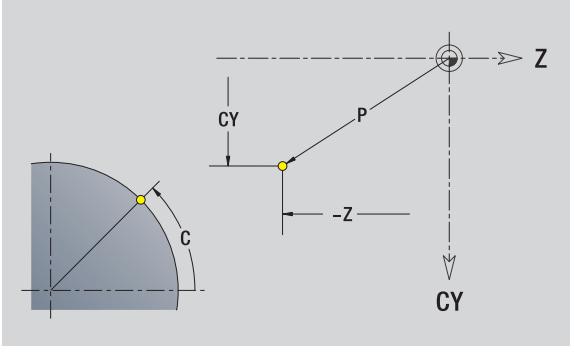


Press the **Contour** menu key.

Insert element

Press the **Insert element** soft key.

Specify the starting point.



Parameters for defining the starting point

ZS	Starting point of contour
CYS	Starting point of the contour as linear dimension (reference: diameter XS)
P	Polar starting point of contour
C	Polar starting point of contour (angle)

HC Milling/drilling attribute:

- 1: Contour milling
- 2: Pocket milling
- 3: Area milling
- 4: Deburring
- 5: Engraving
- 6: Contour milling and deburring
- 7: Pocket milling and deburring
- 14: Do not machine

QF Milling location:

- 0: On the contour
- 1: Inside/left
- 2: Outside/right

HF Direction:

- 0: Up-cut milling
- 1: Climb milling

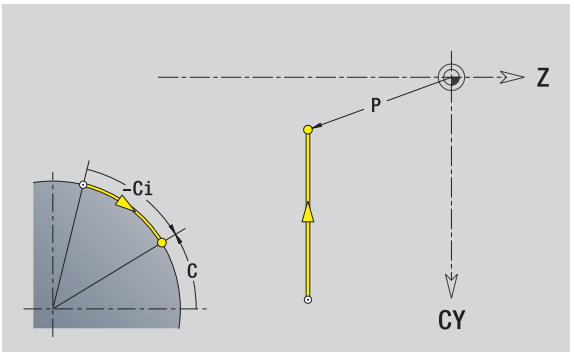
DF Cutter diameter
WF Angle of the chamfer
BR Chamfer width
RB Return plane

In smart.Turn, ICP generates a G110.

Vertical lines on lateral surface



Select the line direction.



Enter the line dimensions and define the transition to the next contour element.

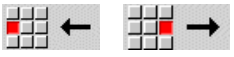
Parameters

CY	Target point as linear dimension (reference: diameter XS)
CYi	Incremental target point as linear dimension (reference: diameter XS)
P	Target point as polar radius
C	Polar target point (angle)
Ci	Incremental polar target point – angle
L	Length of line

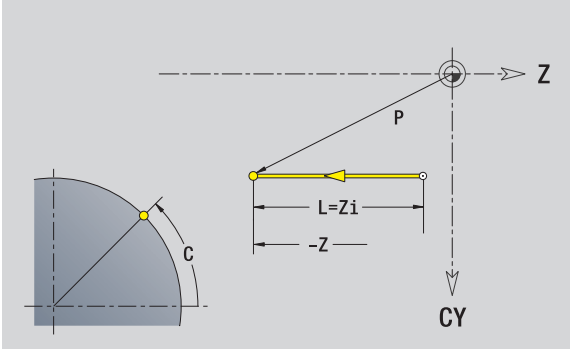
F: See machining attributes Page 361

In smart.Turn, ICP generates a G111.

Horizontal lines on lateral surface



Select the line direction.



Enter the line dimensions and define the transition to the next contour element.

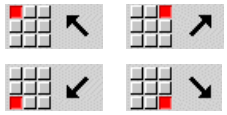
Parameters

Z	Target point
Zi	Incremental target point
P	Target point as polar radius
L	Length of line

F: See machining attributes Page 361

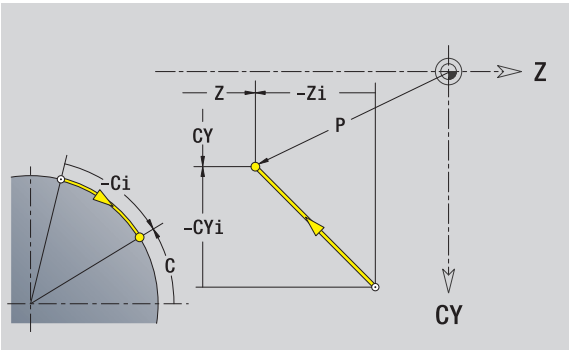
In smart.Turn, ICP generates a G111.

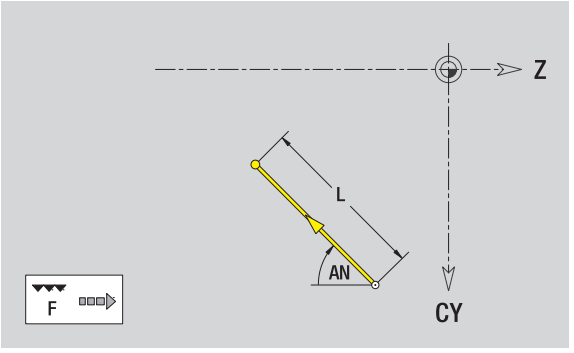
Line at angle on lateral surface



Direction of the line

Enter the line dimensions and define the transition to the next contour element.





Parameters

Z	Target point
Zi	Incremental target point
CY	Target point as linear dimension (reference: diameter XS)
CYi	Incremental target point as linear dimension (reference: diameter XS)
P	Target point as polar radius
C	Polar target point (angle)
Ci	Incremental polar target point – angle
AN	Angle to Z axis (for direction of angle, see help graphic)
ANn	Angle to the subsequent element
ANp	Angle to the previous element
L	Length of line

F: See machining attributes Page 361

In smart.Turn, ICP generates a G111.

Circular arc on lateral surface



Select the arc's direction of rotation.

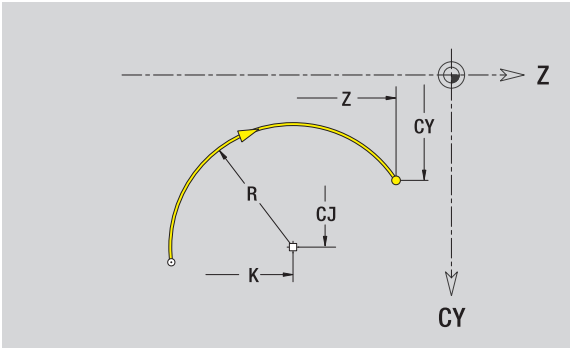
Enter the arc dimensions and define the transition to the next element.

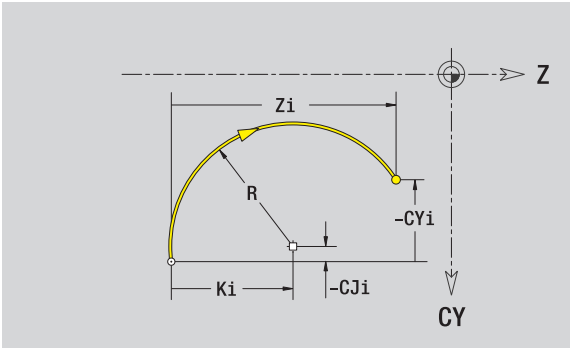
Parameters

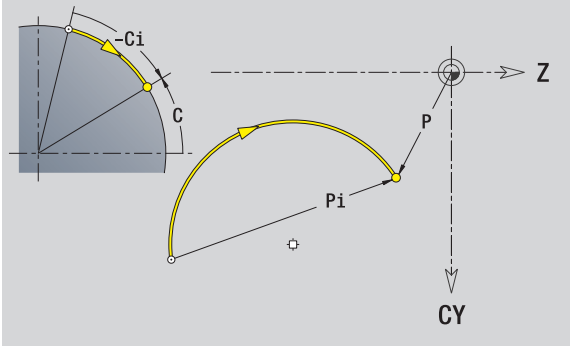
Z	Target point
Zi	Incremental target point
CY	Target point as linear dimension (reference: diameter XS)
CYi	Incremental target point as linear dimension (reference: diameter XS)
P	Target point as polar radius
C	Polar target point (angle)
Pi	Incremental polar target point (distance from starting point to target point)
Ci	Incremental polar target point and angle (relative to the starting point)
K	Center point in Z
Ki	Incremental center point in Z
CJ	Center as linear dimension (reference: diameter XS)
CJi	Incremental center point as linear dimension (reference: diameter XS)
PM	Polar center of arc
PMi	Incremental polar center (distance from starting point to center)
WM	Polar center of arc (angle)
WMi	Incremental polar center and angle (relative to the starting point)
R	Radius
ANs	Tangential angle in the starting point
ANe	Tangential angle in the target point
ANn	Angle to the subsequent element
ANp	Angle to the previous element
L	Length of line

F: See machining attributes Page 361

ICP generates a G112 or G113 in smart.Turn.







Chamfers/rounding arcs on a lateral surface



Select the form elements.



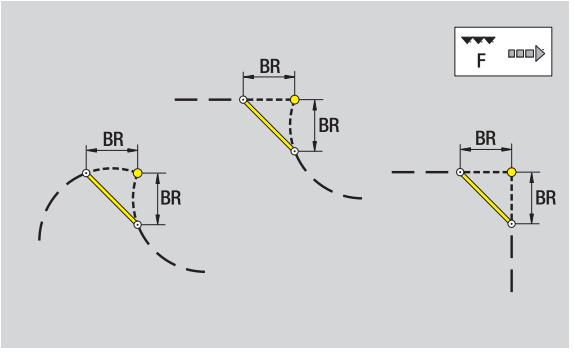
Select a chamfer.

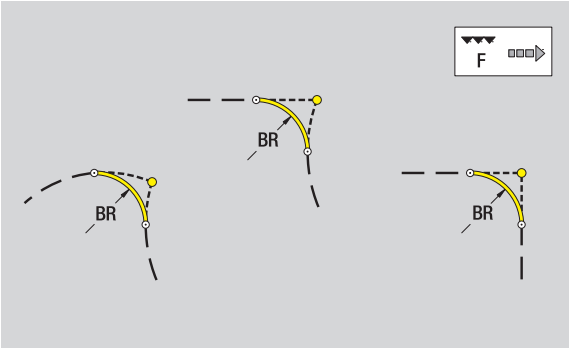


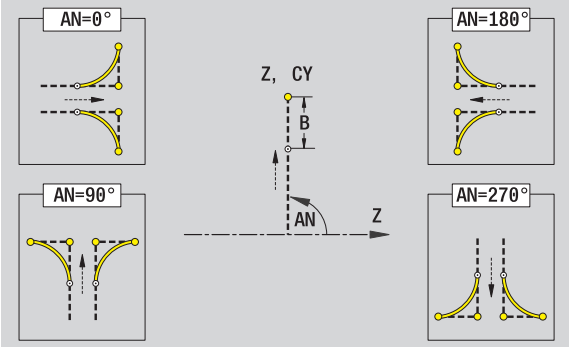
Select rounding arc.

Enter the chamfer width BR or the **rounding radius BR**.

Chamfer/rounding arc as first element: Enter **element position AN**.







Parameters

BR Chamfer width/rounding radius

AN Element position

F: See machining attributes Page 361

Chamfers/rounding arcs are defined on contour corners. A "contour corner" is the point of intersection between the approaching and departing contour elements. A chamfer or rounding cannot be calculated until the departing contour element is known.

ICP integrates the chamfer/rounding arc in smart.Turn on the basic element G111, G112 or G113.

Contour begins with a chamfer or a rounding arc: Enter the position of the intended corner as starting point. Then, in the form element menu, select chamfer or rounding arc. Since the introducing contour element is missing, you enter the **element position AN** to clearly define the position of the chamfer or rounding arc.

ICP converts a chamfer or rounding arc at the start of the contour to a linear or circular element.

5.11 C and Y axis machining in smart.Turn

In smart.Turn, ICP supports the definition of milling contours and holes as well as the creation of milling and drilling patterns that are machined with the aid of the C or Y axis.

Before you use ICP to describe a milling contour or a hole, select the plane:

- C axis
  - Face (XC plane)
  - Lateral surface (ZC plane)
- Y axis
  - Y face (XY plane)
  - Y lateral surface (YZ plane)

A **hole** can contain the following elements:

- Centering
- Core hole
- Countersink
- Thread

The parameters are evaluated for drilling or tapping.

You can arrange holes in linear or circular patterns.

Milling contours: The CNC PILOT is familiar with standard figures (full circle, polygon, slot, etc.). You can define these figures with a few parameters. Complex contours are described with lines and circular arcs.

You can arrange standard figures in linear or circular patterns.

Reference data, nested contours

When describing a milling contour or hole you specify the **reference plane**. The reference plane is the position on which the milling contour or the hole is created.

- Face (C axis): The Z position (reference dimension)
- Lateral surface (C axis): The X position (reference diameter)
- XY plane (Y axis): The Z position (reference dimension)
- YZ plane (Y axis): The X position (reference diameter)

It is also possible to **nest** milling contours and holes. Example: Defining a slot in a rectangular pocket. Holes are drilled inside this slot. You use the reference plane to specify the positions of these elements.

ICP supports the selection of the reference plane. The following reference data are loaded during selection of a reference plane.

- **Face:** Reference dimension
- **Lateral surface:** Reference diameter
- **XY plane:** Reference dimension, spindle angle, limit diameter
- **YZ plane:** Reference dimension, spindle angle

Selecting a reference plane

Select contour, figure, hole, pattern, single surface or polygon.

Select reference plane

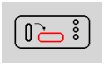
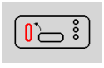
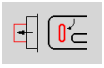
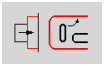
Press the **Select reference plane** soft key. ICP displays the finished part and, if available, the contours already defined.

Use the soft keys (see table at right) to select the reference dimension, reference diameter or existing milling contour as reference plane.

✓

Confirm the reference plane. ICP loads the values of the reference plane as reference data.

Complete the reference data and describe the contour, figure, hole, pattern, single surface or polygon.

Soft keys and nested contours	
	Switches to the next contour of the same reference plane.
	Switches to the previous contour of the same reference plane.
	Switches to the next nested contour.
	Switches to the previous nested contour.

Representation of the ICP elements in the smart.Turn program

Each ICP dialog in **smart.Turn programs** is followed by a section code of further G commands. A hole or milling contour (standard figure and complex contour) contains the following commands:

- Section code (with reference data of this section):
  - FRONT (XC plane)
  - SURFACE (ZC plane)
  - FACE_Y (XY plane)
  - SURFACE_Y (ZY plane)
- G308 (with parameters) as "beginning of the reference plane"
- G function of the figure or hole; sequence of commands with patterns or complex contours
- G309 as "end of a reference plane"

In nested contours a reference plane begins with G308, the next reference plane with the next G308, etc. When the deepest level is finally reached, this reference plane is concluded with G309. Then the next reference plane is concluded with G309 etc.

Note the following points if you describe milling contours or holes with G commands and then machine them with ICP:

- Some parameters are redundant in the DIN contour description. For example, the milling depth can be programmed in the G308 and/or in the G function of the figure. This redundancy does not appear in ICP.
- In DIN programming of figures you have the choice between a Cartesian or polar center dimensioning. The center of the figures is entered in ICP in Cartesian coordinates.

Example: In the DIN contour description, the milling depth is programmed in G308 and in the figure definition. If this figure is changed with ICP, it overwrites the milling depth from G308 with the milling depth from the figure. When saving, ICP saves the milling depth in G308. The G function of the figure is saved without milling depth.



- If you use ICP to edit contour descriptions that were created with G functions, redundant parameters are lost.
- If you load a figure with a center in polar dimensions, the center is converted to Cartesian coordinates.

Example: "Rectangle on the face"

```
. . .
FRONT Z0
N 100 G308 ID"FACE_1" P-5
N 101   G305 XK40 YK10 A0 K30 B15
N 102 G309
```

Example: "Nested figures"

```
. . .
FRONT Z0
N 100 G308 ID"FACE_2" P-5
N 101   G307 XK-40 YK-40 Q5 A0 K-50
N 102   G308 ID"FACE_12" P-3
N 103      G301 XK-35 YK-40 A30 K40 B20
N 104   G309
N 105 G309
```




5.12 Face contours in smart.Turn

In smart.Turn, ICP provides the following contours for machining with the C axis:

- Complex contours defined with individual contour elements
- Figures
- Holes
- Pattern of figures or holes

Reference data for complex face contours

The reference data is followed by the contour definition with individual contour elements: See "Contour elements on face" on page 398.

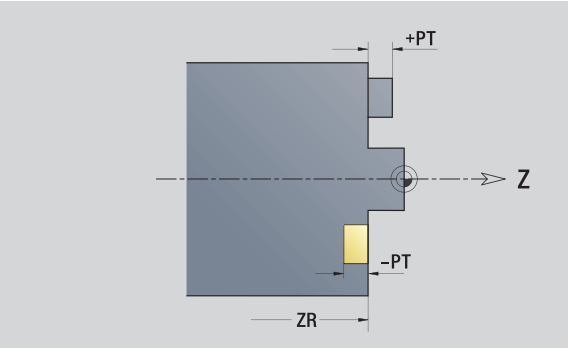
Reference data of face

ID	Contour name
PT	Milling depth
ZR	Reference dimension

You can find the **reference dimension ZR** with the "select reference plane" function (see page 411).

ICP generates:

- The FRONT section code with the reference dimension parameter. In nested contours, ICP generates only one section code.
- A G308 with the parameters contour name and milling depth.
- A G309 at the end of the contour description.



TURN PLUS attributes

In the TURN PLUS attributes you can define settings for the automatic program generation (AWG).

Parameters for defining the starting point

HC Milling/drilling attribute:

- 1: Contour milling
- 2: Pocket milling
- 3: Area milling
- 4: Deburring
- 5: Engraving
- 6: Contour milling and deburring
- 7: Pocket milling and deburring
- 14: Do not machine

QF Milling location:

- 0: On the contour
- 1: Inside/left
- 2: Outside/right

HF Direction:

- 0: Up-cut milling
- 1: Climb milling

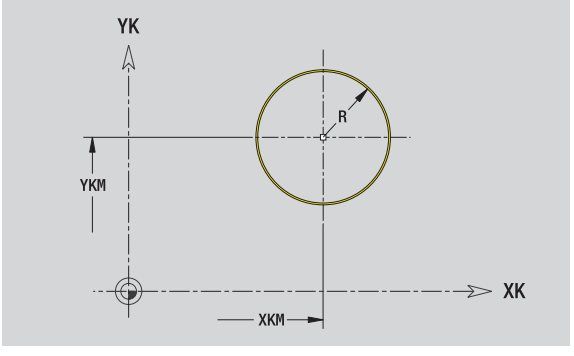
DF Cutter diameter

WF Angle of the chamfer

BR Chamfer width

RB Return plane

Circle on face



Reference data of face

ID Contour name

PT Milling depth

ZR Reference dimension

Parameters of figure

XKM, YKM Center of figure (Cartesian coordinates)

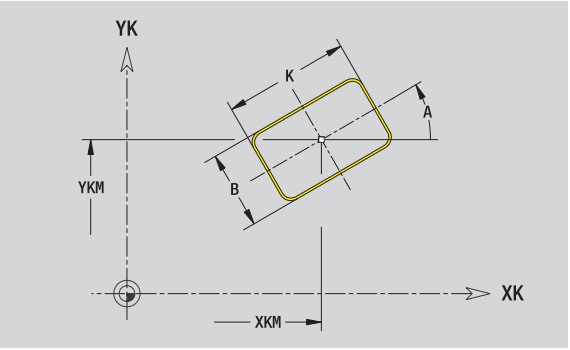
R Radius

You can find the **reference dimension ZR** with the "select reference plane" function (see page 411).

ICP generates:

- The FRONT section code with the reference dimension parameter. In nested contours, ICP generates only one section code.
- A G308 with the parameters contour name and milling depth.
- A G304 with the parameters of the figure.
- A G309.

Rectangle on face



Reference data of face

ID	Contour name
PT	Milling depth
ZR	Reference dimension

Parameters of figure

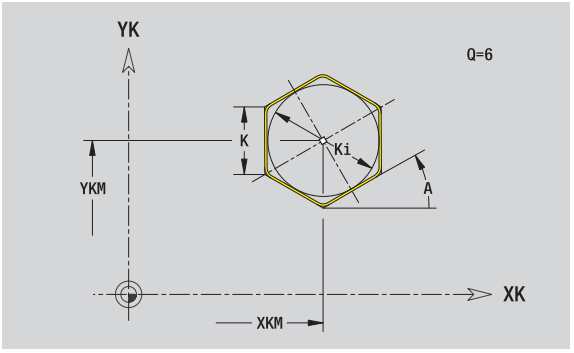
XKM, YKM	Center of figure (Cartesian coordinates)
A	Position angle (reference: XK axis)
K	Length
B	Width
BR	Rounding arc

You can find the **`reference dimension ZR`** with the "select reference plane" function (see page 411).

ICP generates:

- The FRONT section code with the reference dimension parameter. In nested contours, ICP generates only one section code.
- A G308 with the parameters contour name and milling depth.
- A G305 with the parameters of the figure.
- A G309.

Polygon on face



Reference data of face

ID	Contour name
PT	Milling depth
ZR	Reference dimension

Parameters of figure

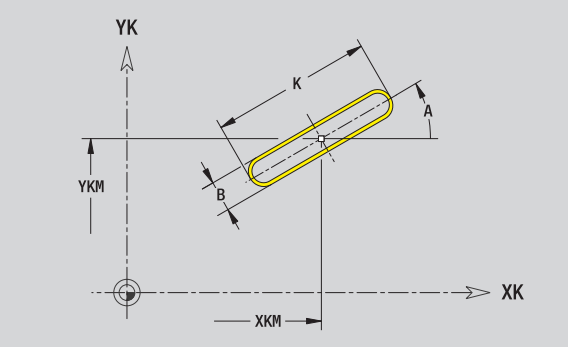
XKM, YKM	Center of figure (Cartesian coordinates)
A	Position angle (reference: XK axis)
Q	Number of corners
K	Length of edge
Ki	Width across flats (inscribed circle diameter)
BR	Rounding arc

You can find the **`reference dimension ZR`** with the "select reference plane" function (see page 411).

ICP generates:

- The FRONT section code with the reference dimension parameter. In nested contours, ICP generates only one section code.
- A G308 with the parameters contour name and milling depth.
- A G307 with the parameters of the figure.
- A G309.

Linear slot on face



Reference data of face

ID	Contour name
PT	Milling depth
ZR	Reference dimension

Parameters of figure

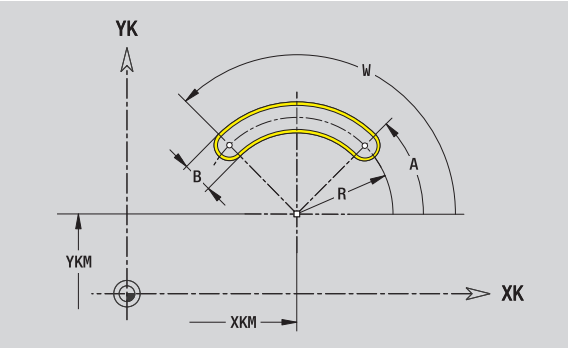
XKM, YKM	Center of figure (Cartesian coordinates)
A	Position angle (reference: XK axis)
K	Length
B	Width

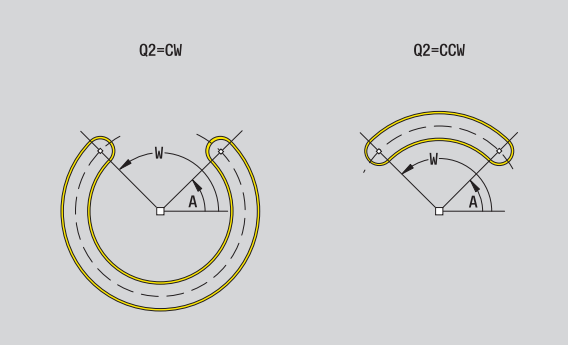
You can find the **reference dimension ZR** with the "select reference plane" function (see page 411).

ICP generates:

- The FRONT section code with the reference dimension parameter. In nested contours, ICP generates only one section code.
- A G308 with the parameters contour name and milling depth.
- A G301 with the parameters of the figure.
- A G309.

Circular slot on face





Reference data of face

ID	Contour name
PT	Milling depth
ZR	Reference dimension

Parameters of figure

XKM, YKM	Center of figure (Cartesian coordinates)
A	Starting angle (reference: XK axis)
W	Position angle (reference: XK axis)
R	Curvature radius (reference: center point path of the slot)
Q2	Direction of rotation ■ CW ■ CCW
B	Width

You can find the **reference dimension ZR** with the "select reference plane" function (see page 411).

ICP generates:

- The FRONT section code with the reference dimension parameter. In nested contours, ICP generates only one section code.
- A G308 with the parameters contour name and milling depth.
- A G302 or G303 with the parameters of the figure.
- A G309.

Hole on face

This function defines a single hole that can contain the following elements:

- Centering
- Core hole
- Countersink
- Thread

Reference data of hole

ID	Contour name
ZR	Reference dimension

Parameters of the hole

XKM, YKM	Center of hole (Cartesian coordinates)

Centering

O	Diameter

Hole

B	Diameter
BT	Depth (without algebraic sign)
W	Angle

Countersink

R	Diameter
U	Depth
E	Sinking angle

Thread

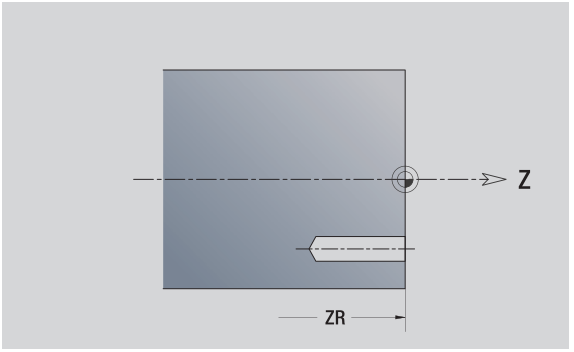
GD	Diameter
GT	Depth
K	Run-out length
F	Thread pitch
GA	Thread type (right-hand/left-hand)

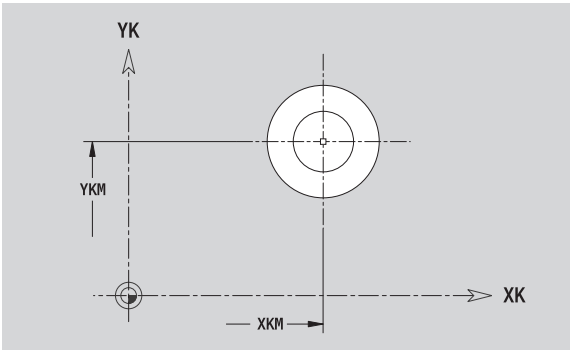
- 0: Right-hand thread
- 1: Left-hand thread

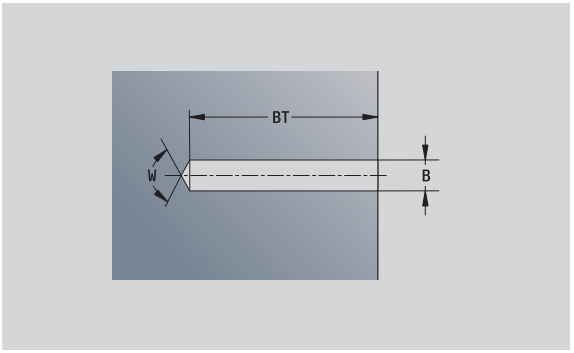
You can find the **`reference dimension ZR`** with the "select reference plane" function (see page 411).

ICP generates:

- The FRONT section code with the reference dimension parameter. In nested contours, ICP generates only one section code.
- A G308 with the parameters contour name and milling depth (–1*BT).
- A G300 with the parameters of the hole.
- A G309.







Linear pattern on face

Reference data of face

ID	Contour name
PT	Milling depth
ZR	Reference dimension

Parameters of pattern

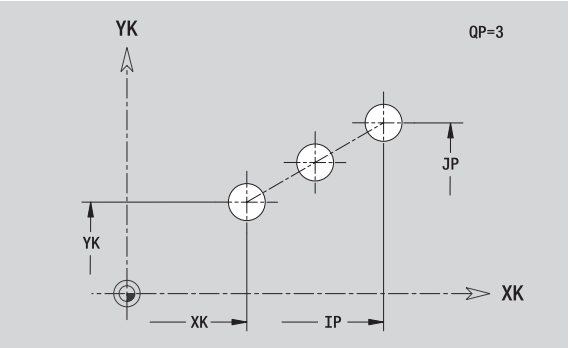
XK, YK	1st point of pattern (Cartesian coordinates)
QP	Number of points in pattern
IP, JP	End point of pattern (Cartesian coordinates)
IPi, JPi	Distance between two pattern points (in XK, YK direction)
AP	Position angle
RP	Total length of pattern
RPi	Distance between two pattern points

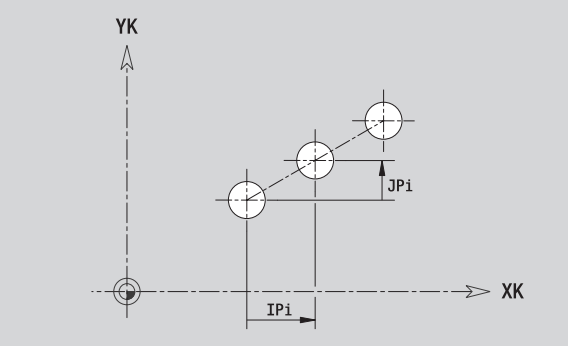
Parameters of the selected figure/hole

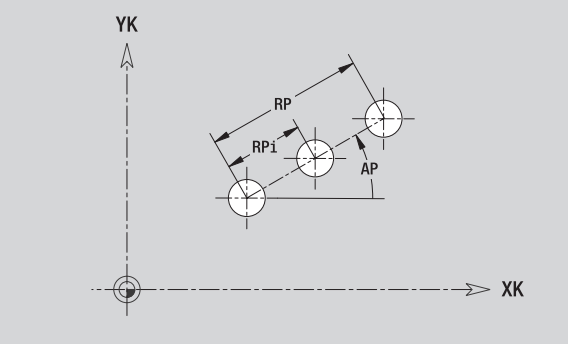
You can find the **`reference dimension ZR`** with the "select reference plane" function (see page 411).

ICP generates:

- The FRONT section code with the reference dimension parameter. In nested contours, ICP generates only one section code.
- A G308 with the parameters contour name and milling depth (–1*BT).
- A G401 with the parameters of the pattern.
- The G function and the parameters of the figure/hole.
- A G309.







Circular pattern on face

Reference data of face

ID	Contour name
PT	Milling depth
ZR	Reference dimension

Parameters of pattern

XK, YK	Center of pattern (Cartesian coordinates)
QP	Number of points in pattern
DR	Direction of rotation (default: 0)

- DR=0, without EP: Figures are arranged on a full circle
- DR=0, with EP: Figures are arranged on the longer circular arc
- DR=0, with EPi: The algebraic sign of EPi defines the direction (EPi<0: clockwise)
- DR=1, with EP: clockwise
- DR=1, with EPi: clockwise (algebraic sign of EPi has no effect)
- DR=2, with EP: counterclockwise
- DR=2, with EPi: counterclockwise (algebraic sign of EPi has no effect)

DP	Pattern diameter
AP	Starting angle (default: 0°)
EP	End angle (no entry: the pattern elements are equally divided into 360°)
EPi	Angle between two figures
H	Element position

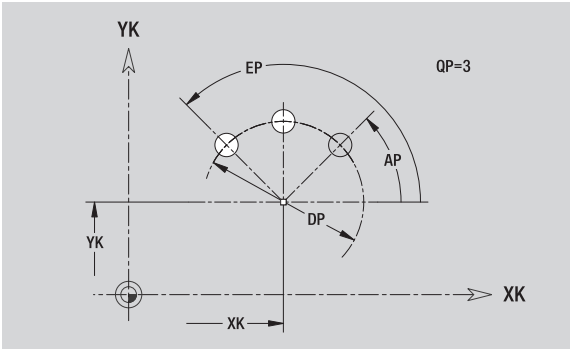
- 0: Normal – the figures are rotated about the circle center (rotation)
- 1: Original position – the position of the figures relative to the coordinate system remains unchanged (translation)

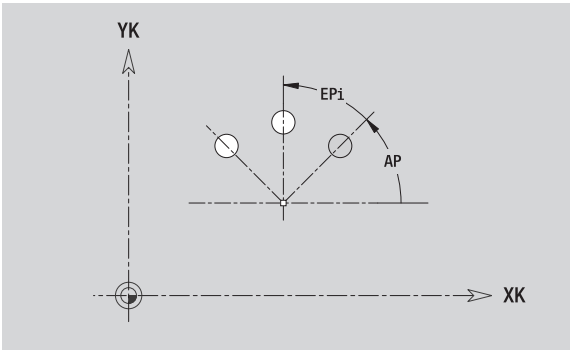
Parameters of the selected figure/hole

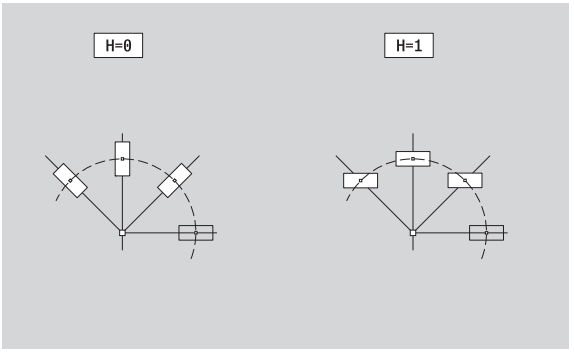
You can find the **reference dimension ZR** with the "select reference plane" function (see page 411).

ICP generates:

- The FRONT section code with the reference dimension parameter. In nested contours, ICP generates only one section code.
- A G308 with the parameters contour name and milling depth (–1*BT).
- A G402 with the parameters of the pattern.
- The G function and the parameters of the figure/hole.
- A G309.







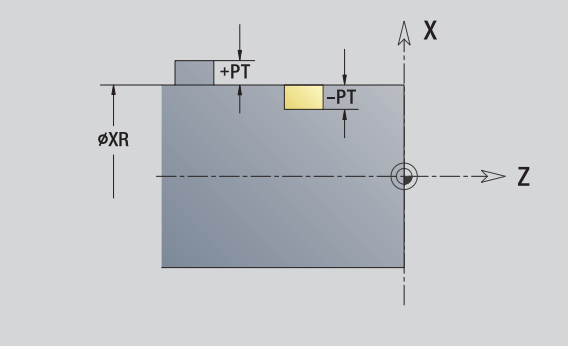
5.13 Lateral surface contours in smart.Turn

In smart.Turn, ICP provides the following contours for machining with the C axis:

- Complex contours defined with individual contour elements
- Figures
- Holes
- Pattern of figures or holes

Reference data of lateral surface

The reference data is followed by the contour definition with individual contour elements: See "Contour elements on lateral surface" on page 404.



Parameters of milling operations

ID	Contour name
PT	Milling depth
XR	Reference diameter

You can find the **reference diameter XR** with the "select reference plane" function (see page 411). The reference diameter is used for converting angle dimensions into linear dimensions.

ICP generates:

- The SURFACE section code with the reference dimension parameter. In nested contours, ICP generates only one section code.
- A G308 with the parameters contour name and milling depth.
- A G309 at the end of the contour description or after the figure.

TURN PLUS attributes

In the TURN PLUS attributes you can define settings for the automatic program generation (AWG).

Parameters for defining the starting point

HC Milling/drilling attribute:

- 1: Contour milling
- 2: Pocket milling
- 3: Area milling
- 4: Deburring
- 5: Engraving
- 6: Contour milling and deburring
- 7: Pocket milling and deburring
- 14: Do not machine

QF Milling location:

- 0: On the contour
- 1: Inside/left
- 2: Outside/right

HF Direction:

- 0: Up-cut milling
- 1: Climb milling

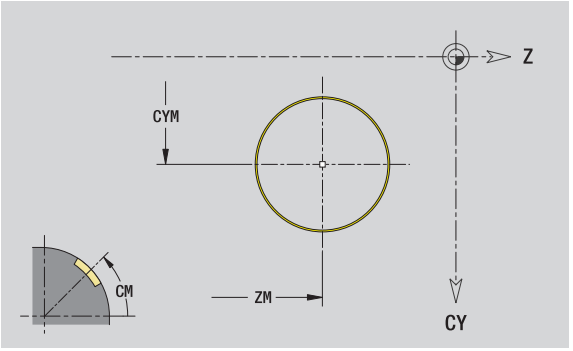
DF Cutter diameter

WF Angle of the chamfer

BR Chamfer width

RB Return plane

Circle on lateral surface



Reference data of lateral surface

ID Contour name
PT Milling depth
XR Reference diameter

Parameters of figure

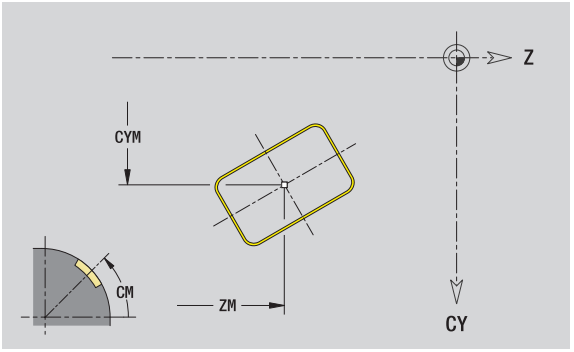
Z Figure center
CYM Center of figure as linear dimension (reference: diameter XR)
CM Center of figure (angle)
R Radius

You can find the **reference diameter XR** with the "select reference plane" function (see page 411).

ICP generates:

- The SURFACE section code with the reference dimension parameter. In nested contours, ICP generates only one section code.
- A G308 with the parameters contour name and milling depth.
- A G314 with the parameters of the figure.
- A G309.

Rectangle on lateral surface



Reference data of lateral surface

ID	Contour name
PT	Milling depth
XR	Reference diameter

Parameters of figure

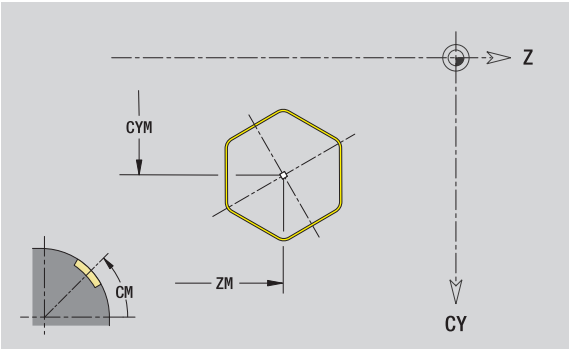
Z	Figure center
CYM	Center of figure as linear dimension (reference: diameter XR)
CM	Center of figure (angle)
A	Position angle
K	Length
B	Width
BR	Rounding arc

You can find the **`reference diameter XR`** with the "select reference plane" function (see page 411).

ICP generates:

- The SURFACE section code with the reference dimension parameter. In nested contours, ICP generates only one section code.
- A G308 with the parameters contour name and milling depth.
- A G315 with the parameters of the figure.
- A G309.

Polygon on lateral surface



Reference data of lateral surface

ID	Contour name
PT	Milling depth
XR	Reference diameter

Parameters of figure

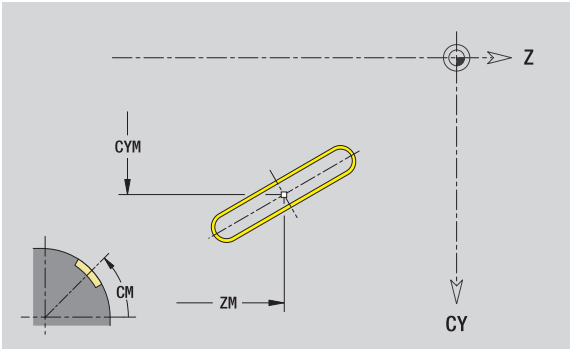
Z	Figure center
CYM	Center of figure as linear dimension (reference: diameter XR)
CM	Center of figure (angle)
A	Position angle
Q	Number of corners
K	Length of edge
Ki	Width across flats (inscribed circle diameter)
BR	Rounding arc

You can find the **`reference diameter XR`** with the "select reference plane" function (see page 411).

ICP generates:

- The SURFACE section code with the reference dimension parameter. In nested contours, ICP generates only one section code.
- A G308 with the parameters contour name and milling depth.
- A G317 with the parameters of the figure.
- A G309.

Linear slot on lateral surface



Reference data of lateral surface

ID Contour name
PT Milling depth
XR Reference diameter

Parameters of figure

Z Figure center
CYM Center of figure as linear dimension (reference: diameter XR)
CM Center of figure (angle)
A Position angle
K Length
B Width

You can find the **`reference diameter XR`** with the "select reference plane" function (see page 411).

ICP generates:

- The SURFACE section code with the reference dimension parameter. In nested contours, ICP generates only one section code.
- A G308 with the parameters contour name and milling depth.
- A G311 with the parameters of the figure.
- A G309.

Circular slot on lateral surface

Reference data of lateral surface

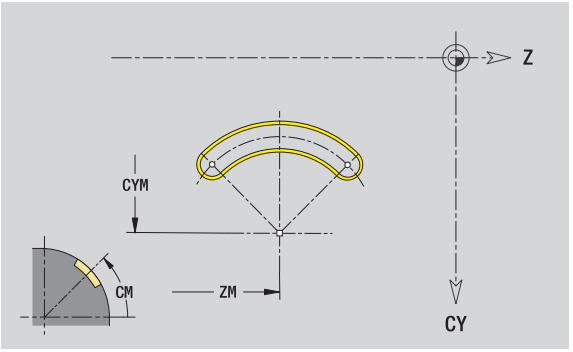
ID Contour name
PT Milling depth
XR Reference diameter

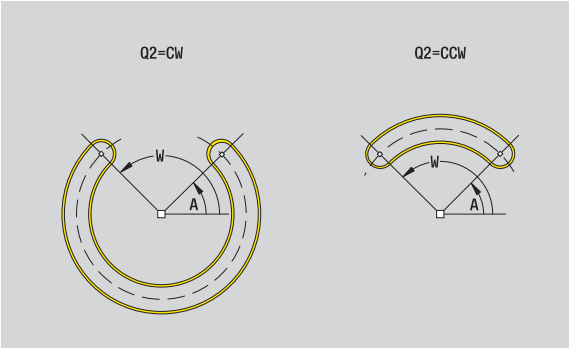
Parameters of figure

Z Figure center
CYM Center of figure as linear dimension (reference: diameter XR)
CM Center of figure (angle)
A Starting angle
W End angle
R Radius
Q2 Direction of rotation

- CW
- CCW

B Width





You can find the **`reference diameter XR`** with the "select reference plane" function (see page 411).

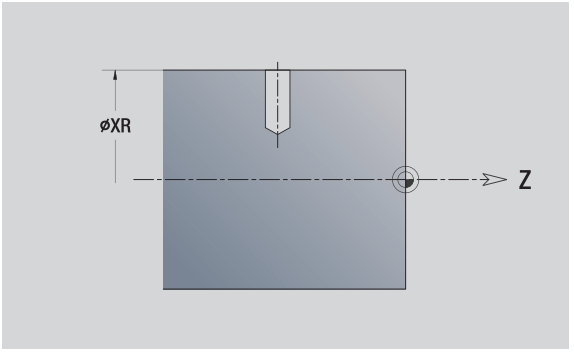
ICP generates:

- The SURFACE section code with the reference dimension parameter. In nested contours, ICP generates only one section code.
- A G308 with the parameters contour name and milling depth.
- A G312 or G313 with the parameters of the figure.
- A G309.

Hole on lateral surface

This function defines a single hole that can contain the following elements:

- Centering
- Core hole
- Countersink
- Thread



Reference data of hole

ID Contour name
XR Reference diameter

Parameters of the hole

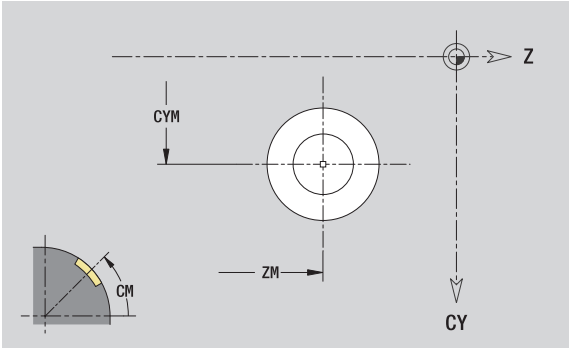
Z Center of hole
CYM Center of figure as linear dimension (reference: diameter XR)
CM Center of figure (angle)

Centering

O Diameter

Hole

B Diameter
BT Depth
W Angle



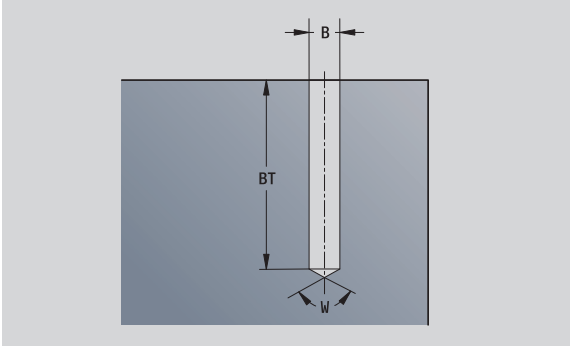
Countersink

R Diameter
U Depth
E Sinking angle

Thread

GD Diameter
GT Depth
K Run-out length
F Thread pitch
GA Thread type (right-hand/left-hand)

- 0: Right-hand thread
- 1: Left-hand thread

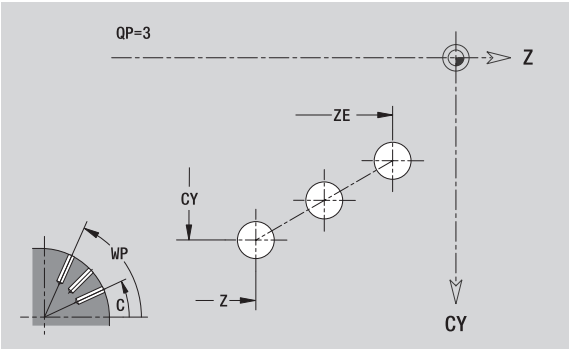


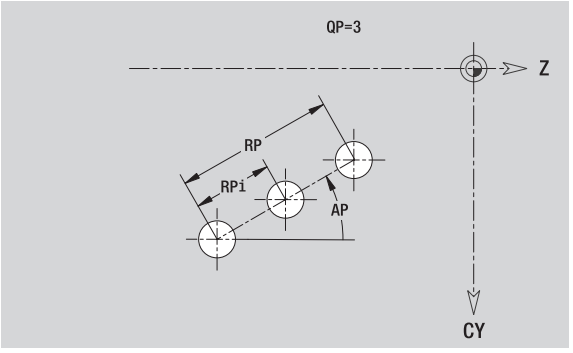
You can find the **`reference diameter XR`** with the "select reference plane" function (see page 411).

ICP generates:

- The SURFACE section code with the reference dimension parameter. In nested contours, ICP generates only one section code.
- A G308 with the parameters contour name and milling depth (–1*BT).
- A G310 with the parameters of the figure.
- A G309.

Linear pattern on lateral surface





Reference data of lateral surface

ID	Contour name
PT	Milling depth
XR	Reference diameter

Parameters of pattern

Z	1st point of pattern
CY	1st pattern point as linear dimension (reference: diameter XR)
C	1st point of pattern (angle)
QP	Number of points in pattern
ZE	End point of pattern
ZEi	Distance between two pattern points (in Z direction)
WP	End point of pattern (angle)
WPi	Distance between two pattern points (angle)
AP	Position angle
RP	Total length of pattern
RPi	Distance between two pattern points

Parameters of the selected figure/hole

You can find the **reference diameter XR** with the "select reference plane" function (see page 411).

ICP generates:

- The SURFACE section code with the reference dimension parameter. In nested contours, ICP generates only one section code.
- A G308 with the parameters contour name and milling depth (–1*BT).
- A G411 with the parameters of the pattern.
- The G function and the parameters of the figure/hole.
- A G309.

Circular pattern on lateral surface

Reference data: (see "Reference data of lateral surface" on page 421)

Reference data of lateral surface

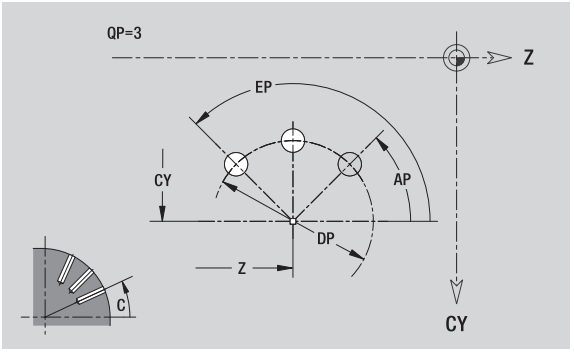
ID Contour name
PT Milling depth
XR Reference diameter

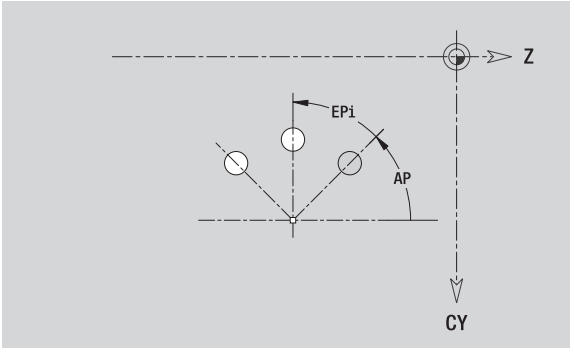
Parameters of pattern

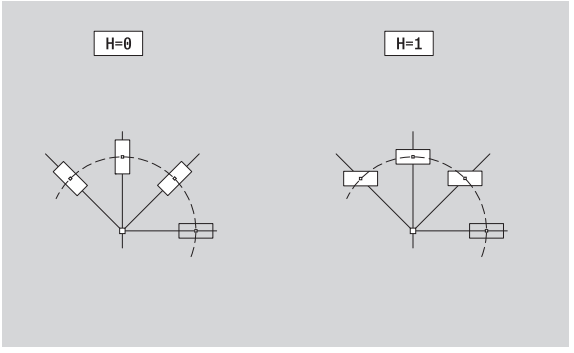
Z Center of pattern
CY Center of pattern as linear dimension (reference: diameter XR)
C Center of pattern (angle)
QP Number of points in pattern
DR Direction of rotation (default: 0)

- DR=0, without EP: Figures are arranged on a full circle
- DR=0, with EP: Figures are arranged on the longer circular arc
- DR=0, with EPi: The algebraic sign of EPi defines the direction (EPi<0: clockwise)
- DR=1, with EP: clockwise
- DR=1, with EPi: clockwise (algebraic sign of EPi has no effect)
- DR=2, with EP: counterclockwise
- DR=2, with EPi: counterclockwise (algebraic sign of EPi has no effect)

DP Pattern diameter
AP Starting angle (default: 0°)







EP End angle (no entry: the pattern elements are equally divided into 360°)

EPi Angle between two figures

H Element position

- 0: Normal – the figures are rotated about the circle center (rotation)
- 1: Original position – the position of the figures relative to the coordinate system remains unchanged (translation)

Parameters of the selected figure/hole

You can find the **reference diameter XR** with the "select reference plane" function (see page 411).

ICP generates:

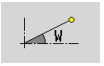
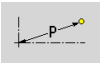
- The SURFACE section code with the reference dimension parameter. In nested contours, ICP generates only one section code.
- A G308 with the parameters contour name and milling depth (-1*BT).
- A G412 with the parameters of the pattern.
- The G function and the parameters of the figure/hole.
- A G309.

5.14 Contours in the XY plane

In smart.Turn, ICP provides the following contours for machining with the Y axis:

- Complex contours defined with individual contour elements
- Figures
- Holes
- Pattern of figures or holes
- Single surface
- Polygon

Enter the dimensions of the XY plane contour elements in Cartesian or polar values. You can switch between them by pressing a soft key (see table). You can mix Cartesian coordinates and polar coordinates to define a point.

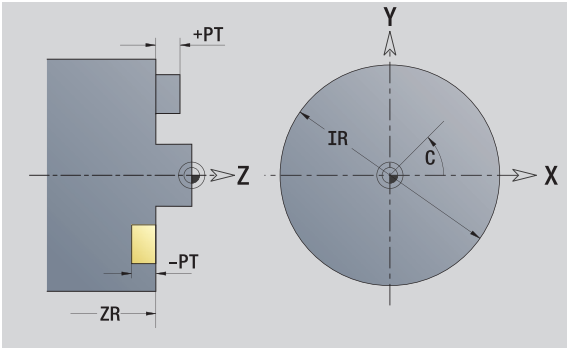
Soft keys for polar coordinates	
	Switches the field to entering the angle **W**.
	Switches the field to entering the radius **P**.

Reference data in XY plane

The reference data is followed by the contour definition with individual contour elements.

Reference data of milling operations

ID	Contour name
PT	Milling depth
C	Spindle angle
IR	Limit diameter
ZR	Reference dimension



You can find the **reference dimension ZR** and the **limit diameter IR** with the "select reference plane" function (see page 411).

ICP generates:

- The section code FRONT_Y with the parameters reference dimension, spindle angle, and limit diameter. The section code is omitted for nested contours.
- A G308 with the parameters contour name and milling depth.
- A G309 at the end of the contour description.

Starting point of contour in XY plane

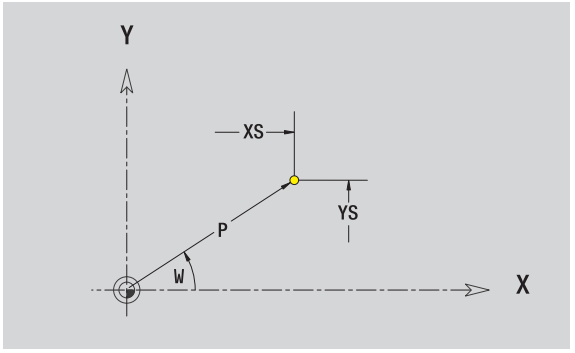
Enter the coordinates for the starting point and target point in the first contour element of the contour. Entering the starting point is only possible in the first contour element. In subsequent contour elements, the starting point results from the previous contour element in each case.



Press the **Contour** menu key.

Press the **Insert element** soft key.

Specify the starting point.



Parameters for defining the starting point

XS, YS	Starting point of contour
W	Polar starting point of contour (angle)
P	Polar starting point of contour (radius)

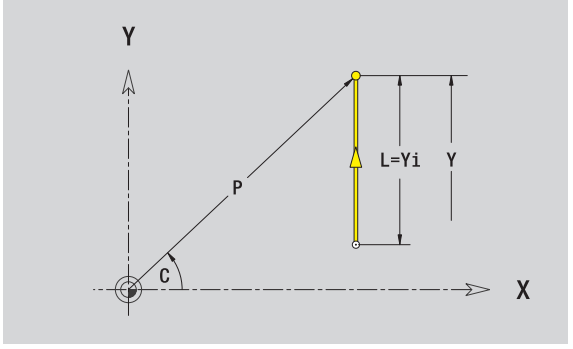
In smart.Turn, ICP generates a G170.

Vertical lines in XY plane



Select the line direction.

Enter the line dimensions and define the transition to the next contour element.



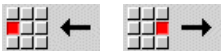
Parameters

Y	Target point
Yi	Incremental target point (distance from starting point to target point)
W	Polar target point (angle)
P	Polar target point
L	Length of line

F: See machining attributes Page 361

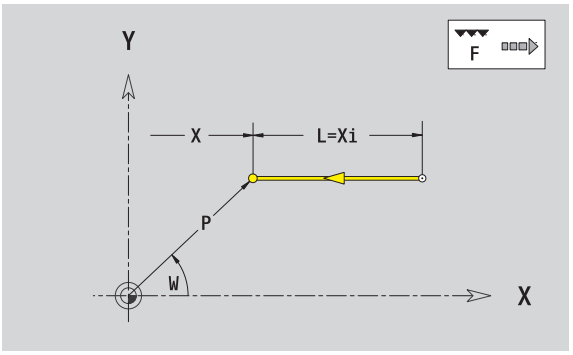
In smart.Turn, ICP generates a G171.

Horizontal lines in XY plane



Select the line direction.

Enter the line dimensions and define the transition to the next contour element.



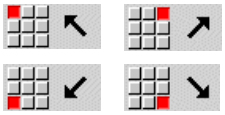
Parameters

X	Target point
Xi	Incremental target point (distance from starting point to target point)
W	Polar target point (angle)
P	Polar target point
L	Length of line

F: See machining attributes Page 361

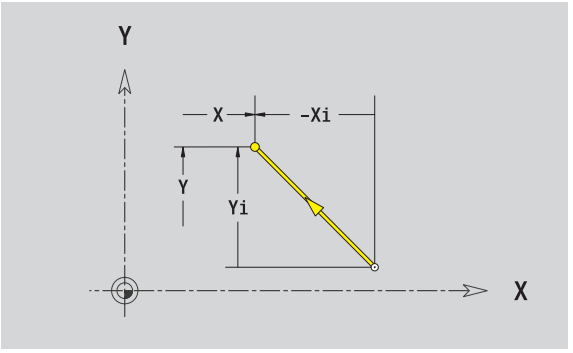
In smart.Turn, ICP generates a G171.

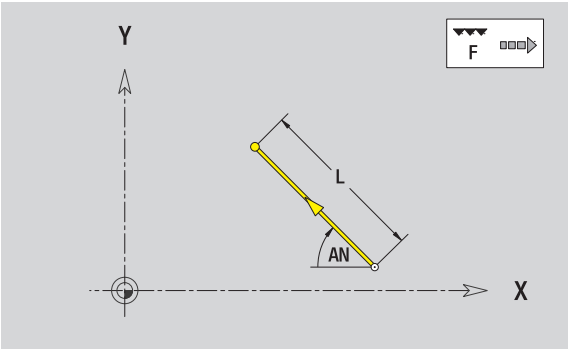
Line at angle in XY plane



Select the line direction.

Enter the line dimensions and define the transition to the next contour element.





Parameters

X, Y	Target point
Xi, Yi	Incremental target point (distance from starting point to target point)
W	Polar target point (angle)
P	Polar target point
AN	Angle to X axis (for direction of angle, see help graphic)
L	Length of line
ANn	Angle to the subsequent element
ANp	Angle to the previous element

F: See machining attributes Page 361

In smart.Turn, ICP generates a G171.

Circular arc in XY plane



Select the arc's direction of rotation.

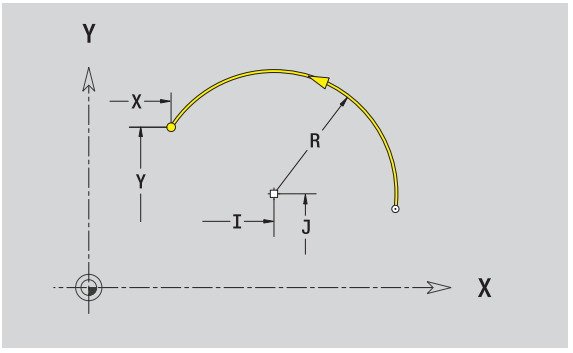
Enter the arc dimensions and define the transition to the next contour element.

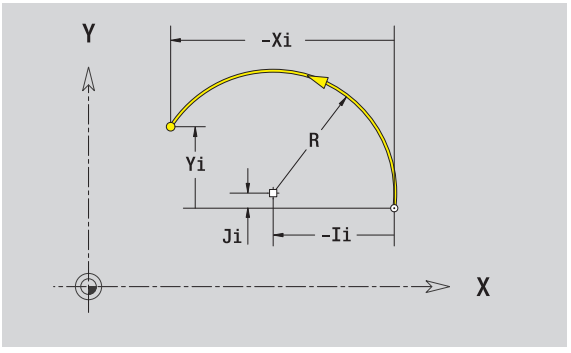
Parameters

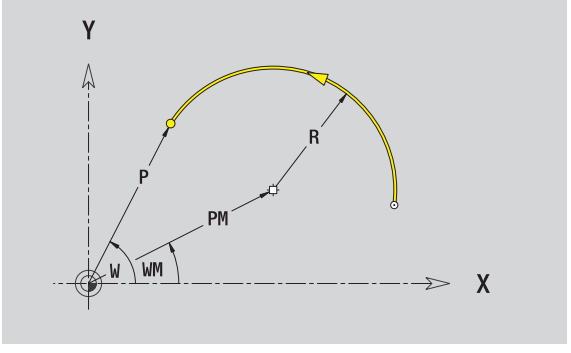
X, Y	Target point (end point of the circular arc)
Xi, Yi	Incremental target point (distance from starting point to target point)
P	Polar target point (radius)
Pi	Incremental polar target point (distance from starting point to target point)
W	Polar target point (angle)
Wi	Incremental polar target point and angle (relative to the starting point)
I, J	Center of arc
Ii, Ji:	Incremental center of arc (distance from starting point to center in X, Z)
PM	Polar center of arc
PMi	Incremental polar center (distance from starting point to center)
WM	Polar center of arc (angle)
WMi	Incremental polar center and angle (relative to the starting point)
R	Radius
ANs	Tangential angle in the starting point
ANe	Tangential angle in the target point
ANp	Angle to the previous element
ANn	Angle to the subsequent element

F: See machining attributes Page 361

ICP generates a G172 or G173 in smart.Turn.







Chamfer/rounding arc in XY plane



Select the form elements.



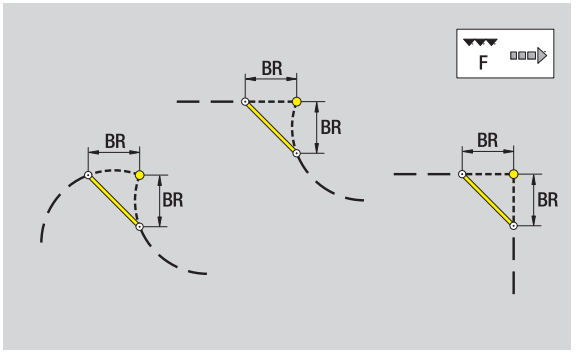
Select a chamfer.

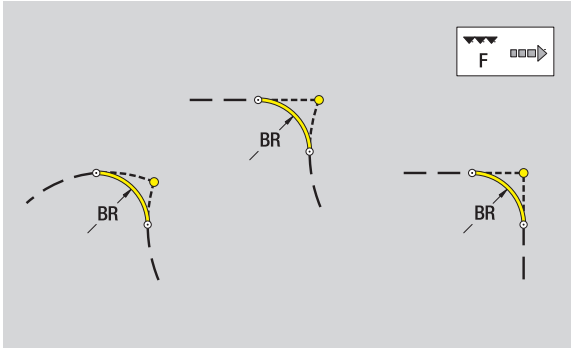


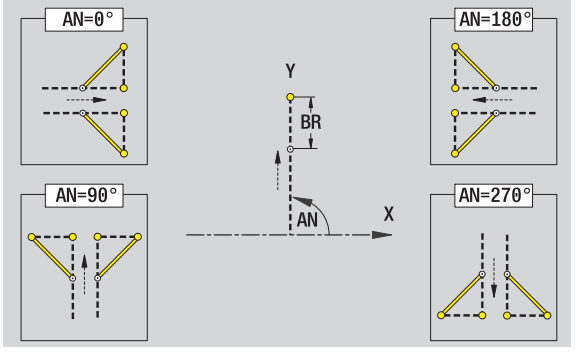
Select rounding arc.

Enter the chamfer width BR or the **rounding radius BR**.

Chamfer/rounding arc as first element: Enter **element position AN**.







Parameters

BR Chamfer width/rounding radius

AN Element position

F: See machining attributes Page 361

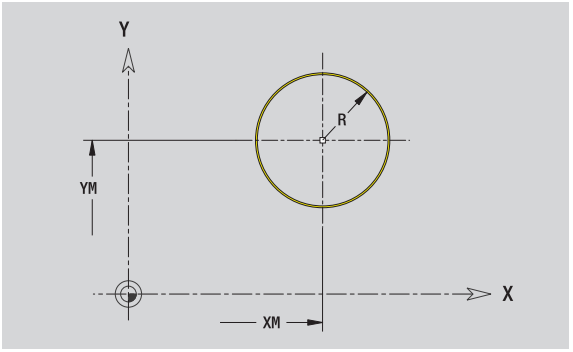
Chamfers/rounding arcs are defined on contour corners. A "contour corner" is the point of intersection between the approaching and departing contour elements. A chamfer or rounding cannot be calculated until the departing contour element is known.

ICP integrates the chamfer/rounding arc in smart.Turn on the basic element G171, G172 or G173.

Contour begins with a chamfer or a rounding arc: Enter the position of the intended corner as starting point. Then, in the form element menu, select chamfer or rounding arc. Since the introducing contour element is missing, you enter the **element position AN** to clearly define the position of the chamfer or rounding arc.

ICP converts a chamfer or rounding arc at the start of the contour to a linear or circular element.

Circle in XY plane



Reference data in XY plane

ID	Contour name
PT	Milling depth
C	Spindle angle
IR	Limit diameter
ZR	Reference dimension

Parameters of figure

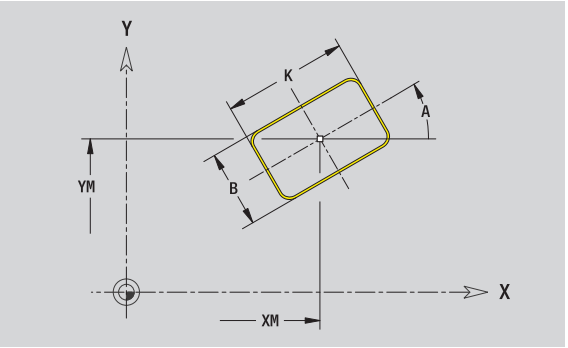
XM, YM	Figure center
R	Radius

You can find the **reference dimension ZR** and the **limit diameter IR** with the "select reference plane" function (see page 411).

ICP generates:

- The section code FRONT_Y with the parameters limit diameter, reference dimension and spindle angle. The section code is omitted for nested contours.
- A G308 with the parameters contour name and milling depth.
- A G374 with the parameters of the figure.
- A G309.

Rectangle in XY plane



Reference data in XY plane

ID	Contour name
PT	Milling depth
C	Spindle angle
IR	Limit diameter
ZR	Reference dimension

Parameters of figure

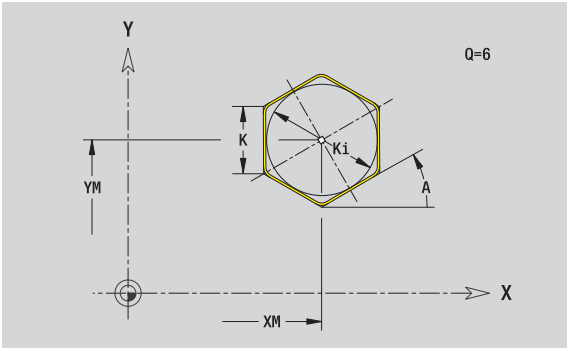
XM, YM	Figure center
A	Position angle (reference: X axis)
K	Length
B	Width
BR	Rounding arc

You can find the **reference dimension ZR** and the **limit diameter IR** with the "select reference plane" function (see page 411).

ICP generates:

- The section code FRONT_Y with the parameters limit diameter, reference dimension and spindle angle. The section code is omitted for nested contours.
- A G308 with the parameters contour name and milling depth.
- A G375 with the parameters of the figure.
- A G309.

Polygon in XY plane



Reference data in XY plane

ID	Contour name
PT	Milling depth
C	Spindle angle
IR	Limit diameter
ZR	Reference dimension

Parameters of figure

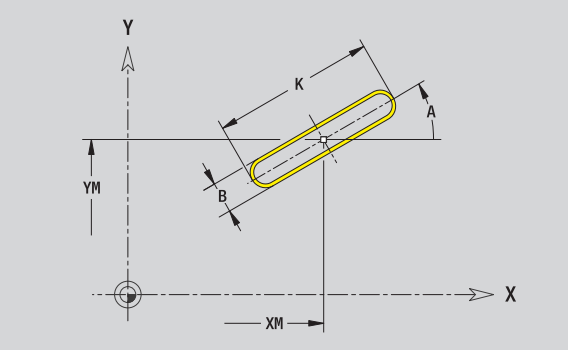
XM, YM	Figure center
A	Position angle (reference: X axis)
Q	Number of corners
K	Length of edge
Ki	Width across flats (inscribed circle diameter)
BR	Rounding arc

You can find the **`reference dimension ZR`** and the **`limit diameter IR`** with the "select reference plane" function (see page 411).

ICP generates:

- The section code FRONT_Y with the parameters limit diameter, reference dimension and spindle angle. The section code is omitted for nested contours.
- A G308 with the parameters contour name and milling depth.
- A G377 with the parameters of the figure.
- A G309.

Linear slot in XY plane



Reference data in XY plane

ID	Contour name
PT	Milling depth
C	Spindle angle
IR	Limit diameter
ZR	Reference dimension

Parameters of figure

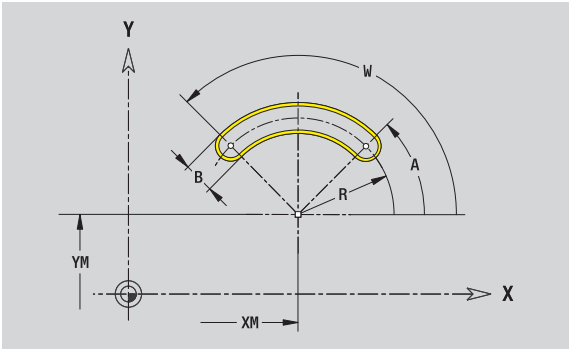
XM, YM	Figure center
A	Position angle (reference: X axis)
K	Length
B	Width

You can find the **reference dimension ZR** and the **limit diameter IR** with the "select reference plane" function (see page 411).

ICP generates:

- The section code FRONT_Y with the parameters limit diameter, reference dimension and spindle angle. The section code is omitted for nested contours.
- A G308 with the parameters contour name and milling depth.
- A G371 with the parameters of the figure.
- A G309.

Circular slot in XY plane

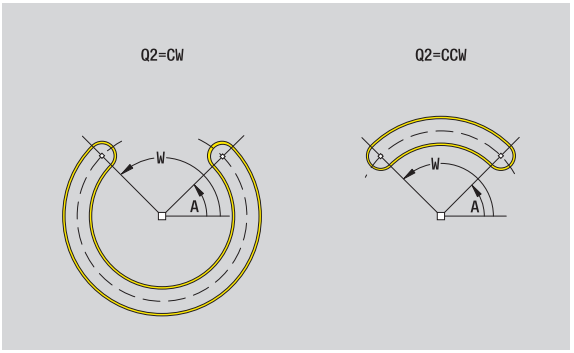


Reference data in XY plane

ID	Contour name
PT	Milling depth
C	Spindle angle
IR	Limit diameter
ZR	Reference dimension

Parameters of figure

XM, YM	Figure center
A	Starting angle (reference: X axis)
W	End angle (reference: X axis)
R	Curvature radius (reference: center point path of the slot)
Q2	Direction of rotation ■ CW ■ CCW
B	Width



You can find the **reference dimension ZR** and the **limit diameter IR** with the "select reference plane" function (see page 411).

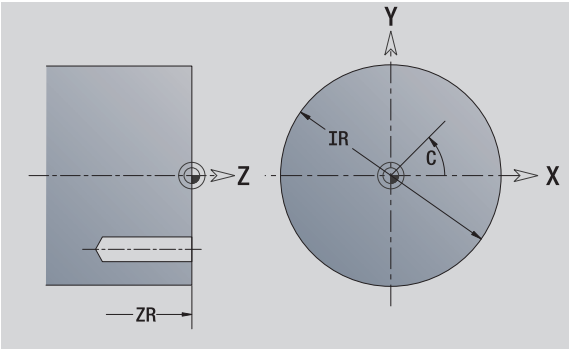
ICP generates:

- The section code FRONT_Y with the parameters limit diameter, reference dimension and spindle angle. The section code is omitted for nested contours.
- A G308 with the parameters contour name and milling depth.
- A G372 or G373 with the parameters of the figure.
- A G309.

Hole in XY plane

This hole defines a single hole that can contain the following elements:

- Centering
- Core hole
- Countersink
- Thread



Reference data of hole

ID	Contour name
C	Spindle angle
IR	Limit diameter
ZR	Reference dimension

Parameters of the hole

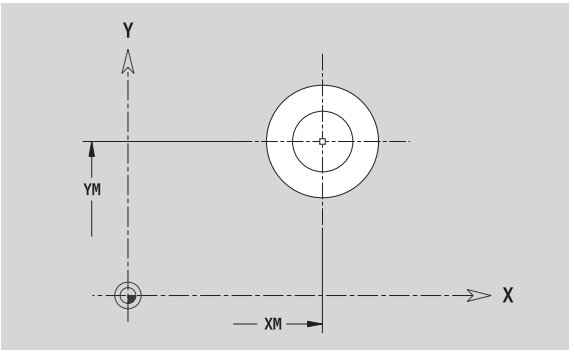
XM, YM	Center of hole

Centering

O	Diameter

Hole

B	Diameter
BT	Depth
W	Angle



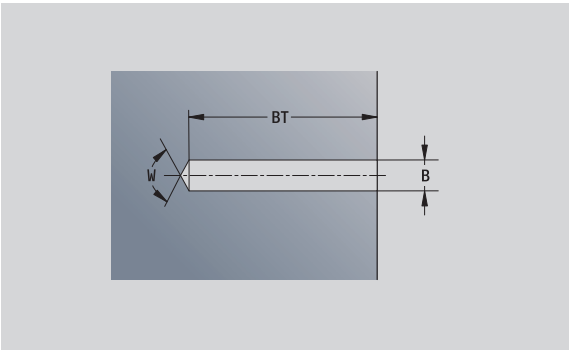
Countersink

R	Diameter
U	Depth
E	Sinking angle

Thread

GD	Diameter
GT	Depth
K	Run-out length
F	Thread pitch
GA	Thread type (right-hand/left-hand)

- 0: Right-hand thread
- 1: Left-hand thread



You can find the **reference dimension ZR** and the **limit diameter IR** with the "select reference plane" function (see page 411).

ICP generates:

- The section code FRONT_Y with the parameters reference dimension, spindle angle, and limit diameter. The section code is omitted for nested contours.
- A G308 with the parameters contour name and milling depth (–1*BT).
- A G370 with the parameters of the figure.
- A G309.

Linear pattern in XY plane

Reference data in XY plane

ID	Contour name
PT	Milling depth
C	Spindle angle
IR	Limit diameter
ZR	Reference dimension

Parameters of pattern

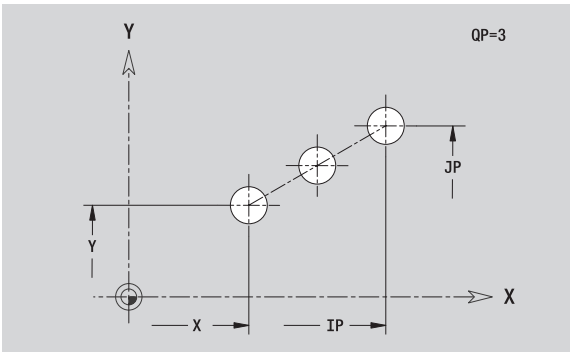
X, Y	1st point of pattern
QP	Number of points in pattern
IP, JP	End point of pattern (Cartesian coordinates)
IPi, JPi	Distance between two pattern points (in X or Y direction)
AP	Position angle
RP	Total length of pattern
RPi	Distance between two pattern points

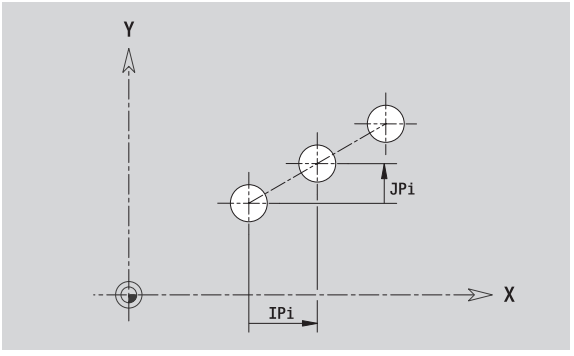
Parameters of the selected figure/hole

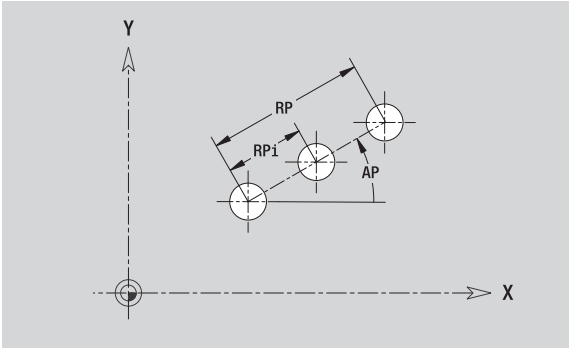
You can find the **reference dimension ZR** and the **limit diameter IR** with the "select reference plane" function (see page 411).

ICP generates:

- The section code FRONT_Y with the parameters limit diameter, reference dimension and spindle angle. The section code is omitted for nested contours.
- A G308 with the parameters contour name and milling depth (–1*BT).
- A G471 with the parameters of the pattern.
- The G function and the parameters of the figure/hole.
- A G309.

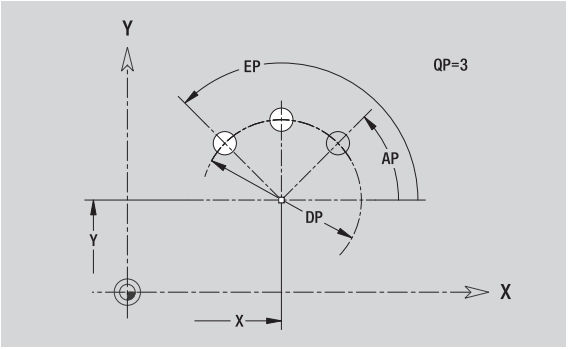


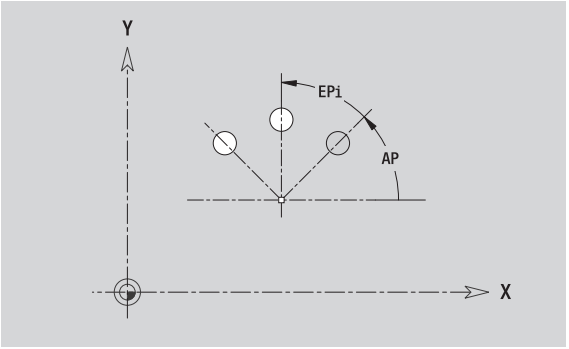


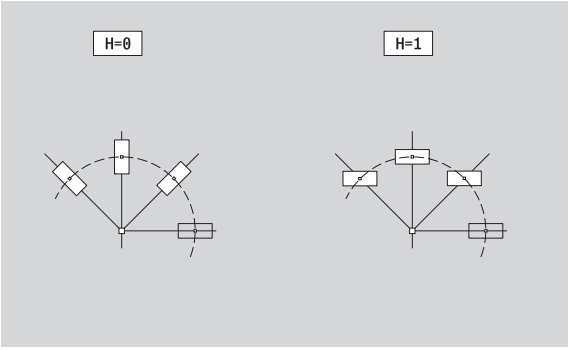


Circular pattern in XY plane

Reference data: (see "Reference data in XY plane" on page 432)







Reference data in XY plane

ID	Contour name
PT	Milling depth
C	Spindle angle
IR	Limit diameter
ZR	Reference dimension

Parameters of pattern

X, Y	Center of pattern
QP	Number of points in pattern
DR	Direction of rotation (default: 0)

- DR=0, without EP: Figures are arranged on a full circle
- DR=0, with EP: Figures are arranged on the longer circular arc
- DR=0, with EPi: The algebraic sign of EPi defines the direction (EPi<0: clockwise)
- DR=1, with EP: clockwise
- DR=1, with EPi: clockwise (algebraic sign of EPi has no effect)
- DR=2, with EP: counterclockwise
- DR=2, with EPi: counterclockwise (algebraic sign of EPi has no effect)

DP	Pattern diameter
AP	Starting angle (default: 0°)
EP	End angle (no entry: the pattern elements are equally divided into 360°)
EPi	Angle between two figures
H	Element position

- 0: Normal – the figures are rotated about the circle center (rotation)
- 1: Original position – the position of the figures relative to the coordinate system remains unchanged (translation)

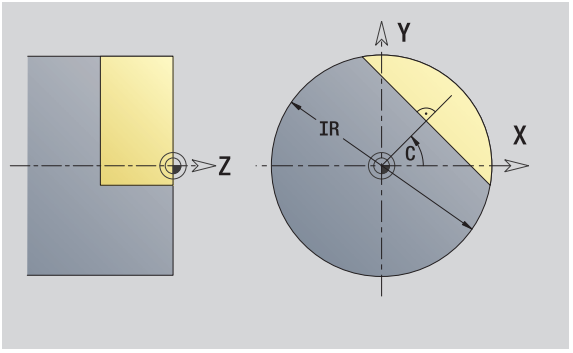
You can find the **reference dimension ZR** and the **limit diameter IR** with the "select reference plane" function (see page 411).

ICP generates:

- The section code FRONT_Y with the parameters limit diameter, reference dimension and spindle angle. The section code is omitted for nested contours.
- A G308 with the parameters contour name and milling depth (–1*BT).
- A G472 with the parameters of the pattern.
- The G function and the parameters of the figure/hole.
- A G309.

Single surface in XY plane

This function defines a surface in the XY plane.



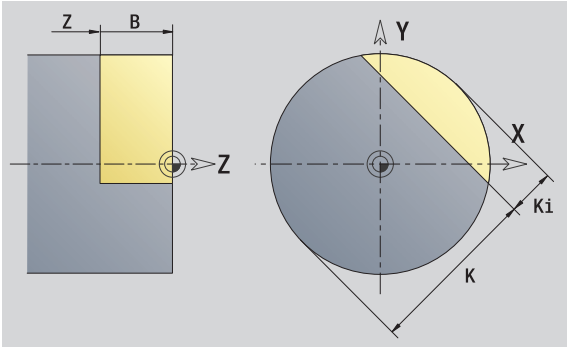
Reference data of the single hole

ID	Contour name
C	Spindle angle (position angle of the surface normal)
IR	Limit diameter

Parameters of the single surface

Z	Reference edge
Ki	Depth
K	Residual depth
B	Width (reference: reference dimension ZR)

- B<0: Surface in negative Z direction
- B>0: Surface in positive Z direction



You can switch between depth (Ki) and residual depth (K) by pressing a soft key (see table at right).

You can find the **reference dimension ZR** and the **limit diameter IR** with the "select reference plane" function (see page 411).

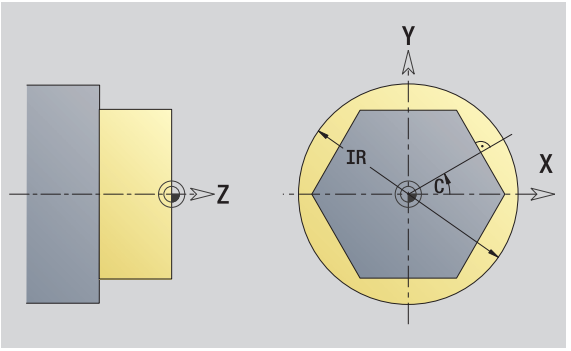
ICP generates:

- The section code FRONT_Y with the parameters limit diameter, reference dimension and spindle angle. The section code is omitted for nested contours.
- A G308 with the parameter contour name.
- A G376 with the parameters of the single surface.
- A G309.

Soft key	
Residual depth	Switches the field to entering the residual depth K.

Centric polygon in XY plane

This function defines polygonal surfaces in the XY plane.



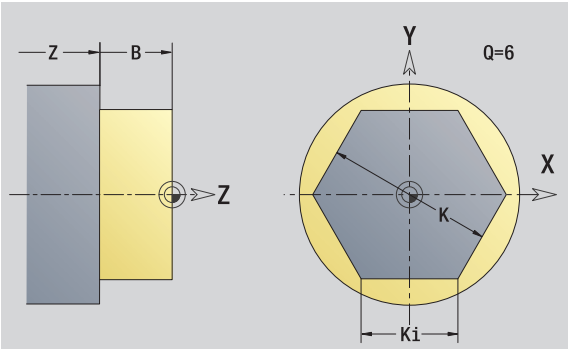
Reference data of the polygon

ID	Contour name
C	Spindle angle (position angle of the surface normal)
IR	Limit diameter

Parameters of the polygon

Z	Reference edge
Q	Number of sides (Q>=2)
K	Width across flats
Ki	Length of side
B	Width (reference: reference dimension ZR)

- B<0: Surface in negative Z direction
- B>0: Surface in positive Z direction



You can switch between edge length (Ki) and width across flats (K) by pressing a soft key (see table at right).

You can find the **reference dimension ZR** and the **limit diameter IR** with the "select reference plane" function (see page 411).

ICP generates:

- The section code FRONT_Y with the parameters limit diameter, reference dimension and spindle angle. The section code is omitted for nested contours.
- A G308 with the parameter contour name.
- A G477 with the parameters of the polygon.
- A G309.

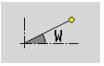
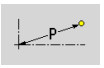
Soft key	
	Switches the field to entering the width across flats K.

5.15 Contours in the YZ plane

In smart.Turn, ICP provides the following contours for machining with the Y axis:

- Complex contours defined with individual contour elements
- Figures
- Holes
- Pattern of figures or holes
- Single surface
- Polygon

Enter the dimensions of the YZ plane contour elements in Cartesian or polar values. You can switch between them by pressing a soft key (see table). You can mix Cartesian coordinates and polar coordinates to define a point.

Soft keys for polar coordinates	
	Switches the field to entering the angle **W**.
	Switches the field to entering the radius **P**.

Reference data in YZ plane

The reference data is followed by the contour definition with individual contour elements.

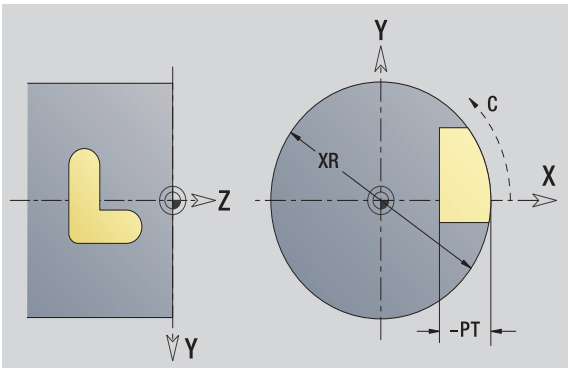
Reference data of milling operations

ID	Contour name
PT	Milling depth
C	Spindle angle
XR	Reference diameter

You can find the **reference diameter XR** with the "select reference plane" function (see page 411).

ICP generates:

- The section code SURFACE_Y with the reference diameter and spindle angle. The section code is omitted for nested contours.
- A G308 with the parameters contour name and milling depth.
- A G309 at the end of the contour description.



TURN PLUS attributes

In the TURN PLUS attributes you can define settings for the automatic program generation (AWG).

Parameters for defining the starting point

HC Milling/drilling attribute:

- 1: Contour milling
- 2: Pocket milling
- 3: Area milling
- 4: Deburring
- 5: Engraving
- 6: Contour milling and deburring
- 7: Pocket milling and deburring
- 14: Do not machine

QF Milling location:

- 0: On the contour
- 1: Inside/left
- 2: Outside/right

HF Direction:

- 0: Up-cut milling
- 1: Climb milling

DF Cutter diameter

WF Angle of the chamfer

BR Chamfer width

RB Return plane

Starting point of contour in YZ plane

Enter the coordinates for the starting point and target point in the first contour element of the contour. Entering the starting point is only possible in the first contour element. In subsequent contour elements, the starting point results from the previous contour element in each case.

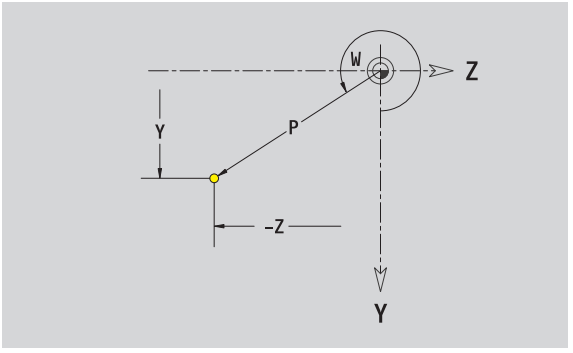
Contour

Press the **Contour** menu key.

Insert element

Press the **Insert element** soft key.

Specify the starting point.



Parameters for defining the starting point

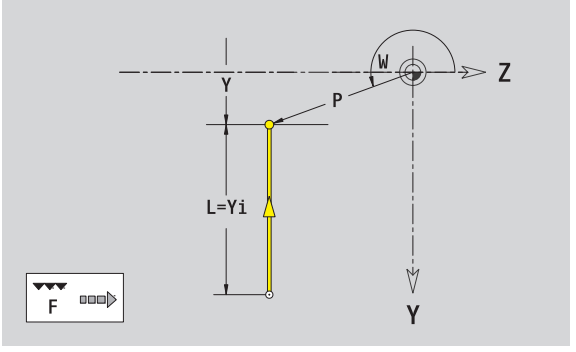
YS, ZS	Starting point of contour
W	Polar starting point of contour (angle)
P	Polar starting point of contour (radius)

In smart.Turn, ICP generates a G180.

Vertical lines in YZ plane

Select the line direction.

Enter the line dimensions and define the transition to the next contour element.



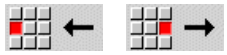
Parameters

Y	Target point
Yi	Incremental target point (distance from starting point to target point)
W	Polar target point (angle)
P	Polar target point
L	Length of line

F: See machining attributes Page 361

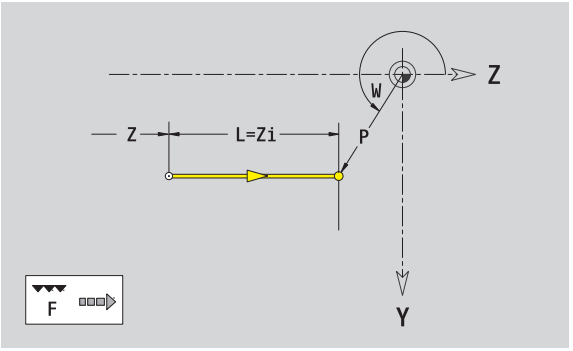
In smart.Turn, ICP generates a G181.

Horizontal lines in YZ plane



Select the line direction.

Enter the line dimensions and define the transition to the next contour element.



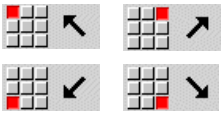
Parameters

Z	Target point
Zi	Incremental target point (distance from starting point to target point)
W	Polar target point (angle)
P	Polar target point
L	Length of line

F: See machining attributes Page 361

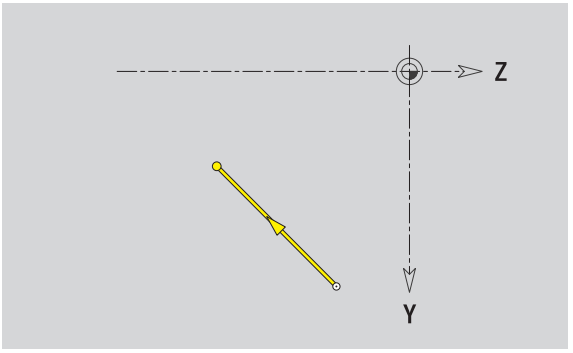
In smart.Turn, ICP generates a G181.

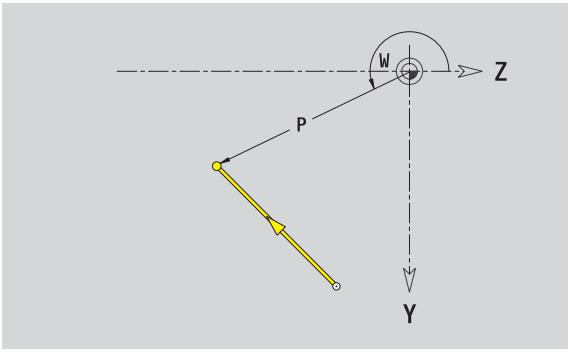
Line at angle in YZ plane



Select the line direction.

Enter the line dimensions and define the transition to the next contour element.





Parameters

Y, Z	Target point
Yi, Zi	Incremental target point (distance from starting point to target point)
W	Polar target point (angle)
P	Polar target point
AN	Angle to Z axis (for direction of angle, see help graphic)
L	Length of line
ANn	Angle to the subsequent element
ANp	Angle to the previous element

F: See machining attributes Page 361

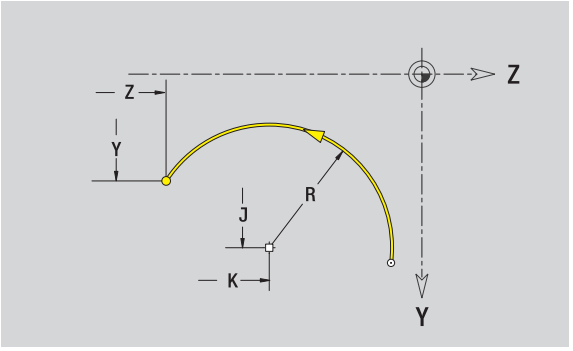
In smart.Turn, ICP generates a G181.

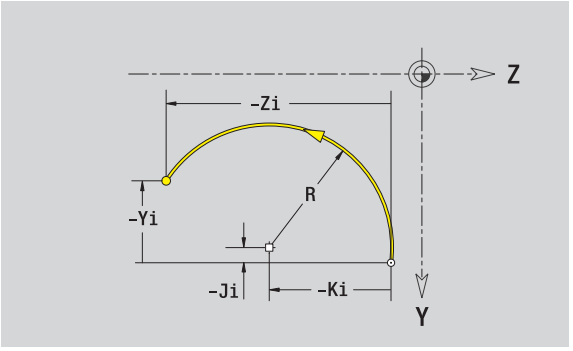
Circular arc in YZ plane

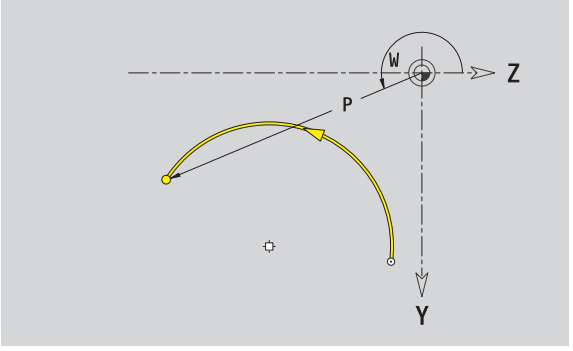


Select the arc's direction of rotation.

Enter the arc dimensions and define the transition to the next contour element.







Parameters

Y, Z	Target point (end point of the circular arc)
Yi, Zi	Incremental target point (distance from starting point to target point)
P	Polar target point (radius)
Pi	Incremental polar target point (distance from starting point to target point)
W	Polar target point (angle)
Wi	Incremental polar target point and angle (relative to the starting point)
J, K	Center of arc
Ji, Ki	Incremental center of arc (distance from starting point to center in X, Z)
PM	Polar center of arc
PMi	Incremental polar center (distance from starting point to center)
WM	Polar center of arc (angle)
WMi	Incremental polar center and angle (relative to the starting point)
R	Radius
ANs	Tangential angle in the starting point
ANe	Tangential angle in the target point
ANp	Angle to the previous element
ANn	Angle to the subsequent element

F: See machining attributes Page 361

ICP generates a G182 or G183 in smart.Turn.

Chamfer/rounding arc in YZ plane



Select the form elements.



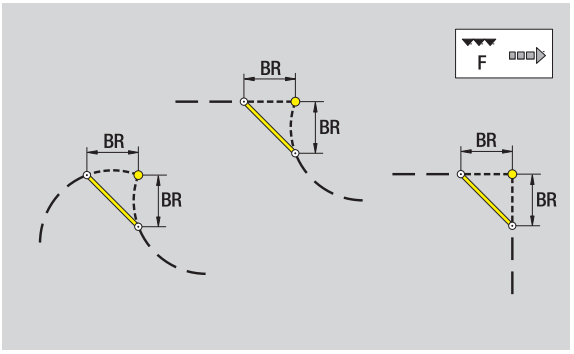
Select a chamfer.

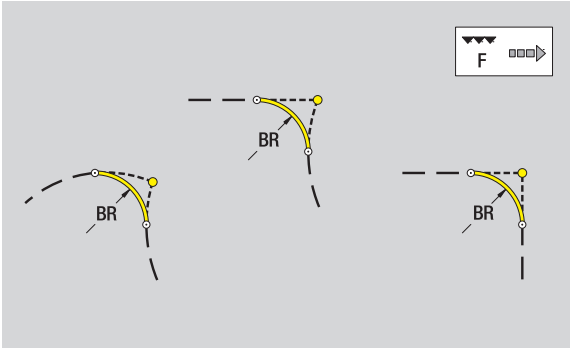


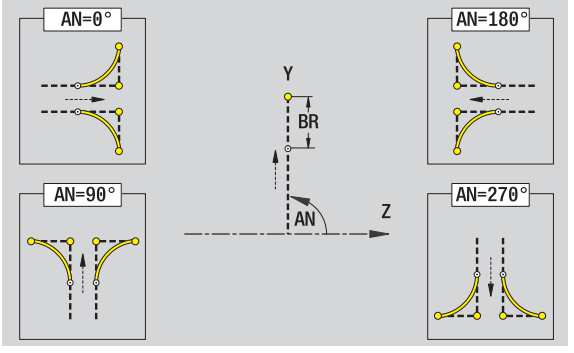
Select rounding arc.

Enter the chamfer width BR or the **rounding radius BR**.

Chamfer/rounding arc as first element: Enter **element position AN**.







Parameters

BR Chamfer width/rounding radius

AN Element position

F: See machining attributes Page 361

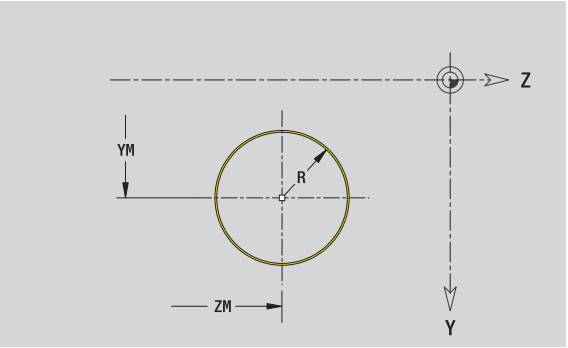
Chamfers/rounding arcs are defined on contour corners. A "contour corner" is the point of intersection between the approaching and departing contour elements. A chamfer or rounding cannot be calculated until the departing contour element is known.

ICP integrates the chamfer/rounding arc in smart.Turn on the basic element G181, G182 or G183.

Contour begins with a chamfer or a rounding arc: Enter the position of the intended corner as starting point. Then, in the form element menu, select chamfer or rounding arc. Since the introducing contour element is missing, you enter the **element position AN** to clearly define the position of the chamfer or rounding arc.

ICP converts a chamfer or rounding arc at the start of the contour to a linear or circular element.

Circle in YZ plane



Reference data in YZ plane

ID	Contour name
PT	Milling depth
C	Spindle angle
XR	Reference diameter

Parameters of figure

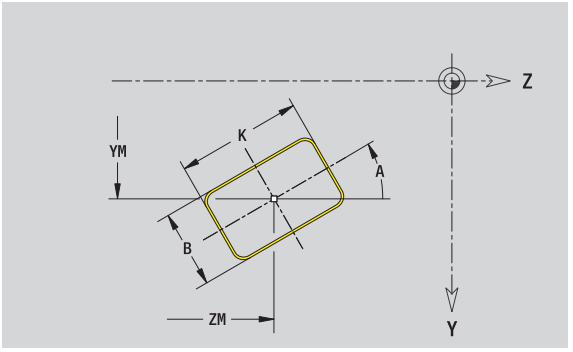
YM, ZM	Figure center
R	Radius

You can find the **reference diameter XR** with the "select reference plane" function (see page 411).

ICP generates:

- The section code SURFACE_Y with the reference diameter and spindle angle. The section code is omitted for nested contours.
- A G308 with the parameters contour name and milling depth.
- A G384 with the parameters of the figure.
- A G309.

Rectangle in YZ plane



Reference data in YZ plane

ID	Contour name
PT	Milling depth
C	Spindle angle
XR	Reference diameter

Parameters of figure

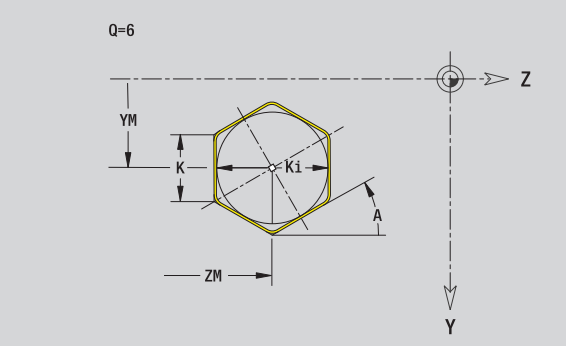
YM, ZM	Figure center
A	Position angle (reference: X axis)
K	Length
B	Width
BR	Rounding arc

You can find the **reference diameter XR** with the "select reference plane" function (see page 411).

ICP generates:

- The section code SURFACE_Y with the reference diameter and spindle angle. The section code is omitted for nested contours.
- A G308 with the parameters contour name and milling depth.
- A G385 with the parameters of the figure.
- A G309.

Polygon in YZ plane



Reference data in YZ plane

ID	Contour name
PT	Milling depth
C	Spindle angle
XR	Reference diameter

Parameters of figure

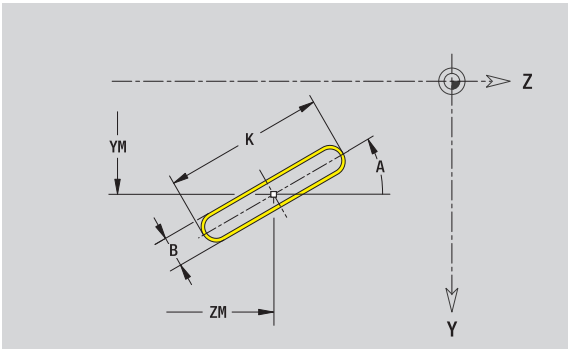
YM, ZM	Figure center
A	Position angle (reference: X axis)
Q	Number of corners
K	Length of edge
Ki	Width across flats (inscribed circle diameter)
BR	Rounding arc

You can find the **`reference diameter XR`** with the "select reference plane" function (see page 411).

ICP generates:

- The section code SURFACE_Y with the reference diameter and spindle angle. The section code is omitted for nested contours.
- A G308 with the parameters contour name and milling depth.
- A G387 with the parameters of the figure.
- A G309.

Linear slot in YZ plane



Reference data in YZ plane

ID	Contour name
PT	Milling depth
C	Spindle angle
XR	Reference diameter

Parameters of figure

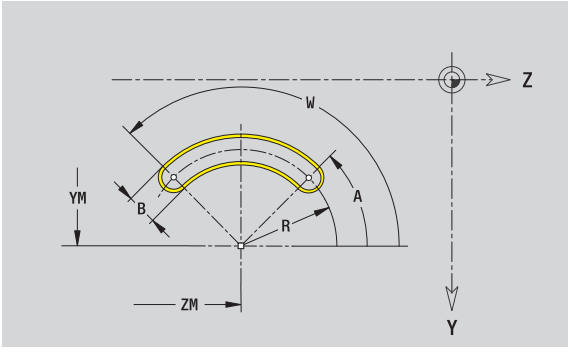
YM, ZM	Figure center
A	Position angle (reference: X axis)
K	Length
B	Width

You can find the **reference diameter XR** with the "select reference plane" function (see page 411).

ICP generates:

- The section code SURFACE_Y with the reference diameter and spindle angle. The section code is omitted for nested contours.
- A G308 with the parameters contour name and milling depth.
- A G381 with the parameters of the figure.
- A G309.

Circular slot in YZ plane

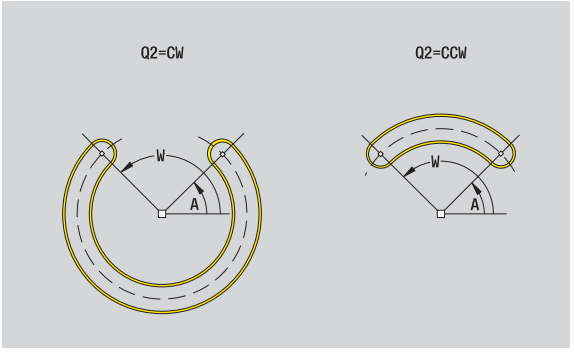


Reference data in YZ plane

ID	Contour name
PT	Milling depth
C	Spindle angle
XR	Reference diameter

Parameters of figure

YM, ZM	Figure center
A	Starting angle (reference: X axis)
W	End angle (reference: X axis)
R	Curvature radius (reference: center point path of the slot)
Q2	Direction of rotation ■ CW ■ CCW
B	Width



You can find the **reference diameter XR** with the "select reference plane" function (see page 411).

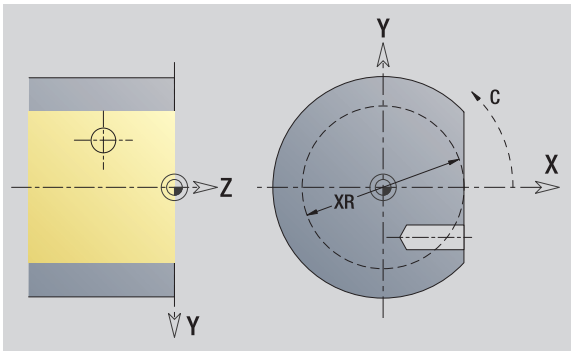
ICP generates:

- The section code SURFACE_Y with the reference diameter and spindle angle. The section code is omitted for nested contours.
- A G308 with the parameters contour name and milling depth.
- A G382 or G383 with the parameters of the figure.
- A G309.

Hole in YZ plane

This hole defines a single hole that can contain the following elements:

- Centering
- Core hole
- Countersink
- Thread



Reference data of hole

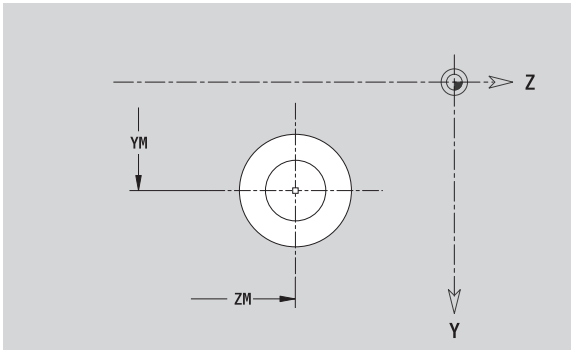
ID	Contour name
C	Spindle angle
XR	Reference diameter

Parameters of the hole

YM, ZM	Center of hole

Centering

O	Diameter



Hole

B	Diameter
BT	Depth
W	Angle

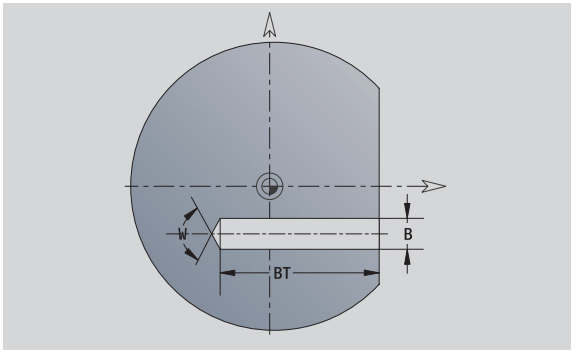
Countersink

R	Diameter
U	Depth
E	Sinking angle

Thread

GD	Diameter
GT	Depth
K	Run-out length
F	Thread pitch
GA	Thread type (right-hand/left-hand)

- 0: Right-hand thread
- 1: Left-hand thread



You can find the **`reference diameter XR`** with the "select reference plane" function (see page 411).

ICP generates:

- The section code SURFACE_Y with the reference diameter and spindle angle. The section code is omitted for nested contours.
- A G308 with the parameters contour name and milling depth (-1*BT).
- A G380 with the parameters of the figure.
- A G309.

Linear pattern in YZ plane

Reference data in YZ plane

ID	Contour name
PT	Milling depth
C	Spindle angle
XR	Reference diameter

Parameters of pattern

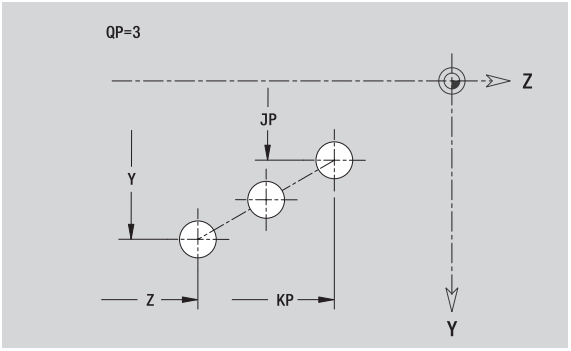
Y, Z	1st point of pattern
QP	Number of points in pattern
JP, KP	End point of pattern (Cartesian coordinates)
JPi, KPi	Distance between two pattern points (in Y or Z direction)
AP	Position angle
RP	Total length of pattern
RPi	Distance between two pattern points

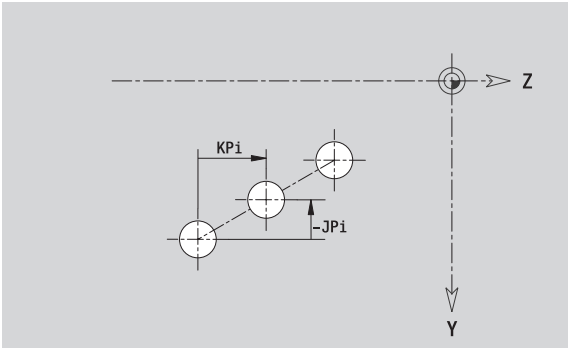
Parameters of the selected figure/hole

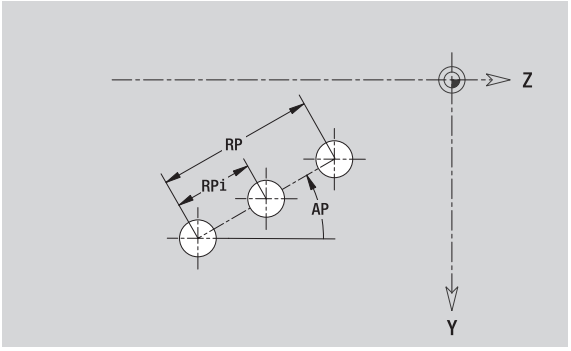
You can find the **`reference diameter XR`** with the "select reference plane" function (see page 411).

ICP generates:

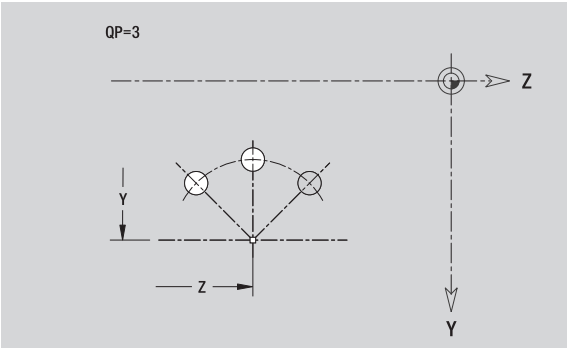
- The section code SURFACE_Y with the reference diameter and spindle angle. The section code is omitted for nested contours.
- A G308 with the parameters contour name and milling depth (-1*BT).
- A G481 with the parameters of the pattern.
- The G function and the parameters of the figure/hole.
- A G309.

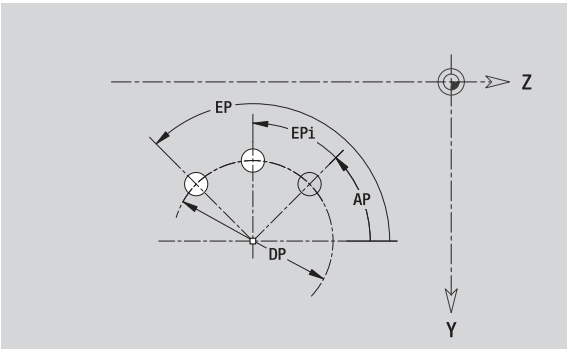


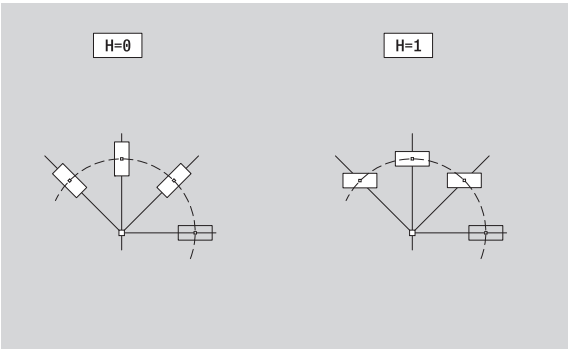




Circular pattern in YZ plane







Reference data in YZ plane

ID	Contour name
PT	Milling depth
C	Spindle angle
XR	Reference diameter

Parameters of pattern

Y, Z	Center of pattern
QP	Number of points in pattern
DR	Direction of rotation (default: 0)

- DR=0, without EP: Figures are arranged on a full circle
- DR=0, with EP: Figures are arranged on the longer circular arc
- DR=0, with EPi: The algebraic sign of EPi defines the direction (EPi<0: clockwise)
- DR=1, with EP: clockwise
- DR=1, with EPi: clockwise (algebraic sign of EPi has no effect)
- DR=2, with EP: counterclockwise
- DR=2, with EPi: counterclockwise (algebraic sign of EPi has no effect)

DP	Pattern diameter
AP	Starting angle (default: 0°)
EP	End angle (no entry: the pattern elements are equally divided into 360°)
EPi	Angle between two figures
H	Element position

- 0: Normal – the figures are rotated about the circle center (rotation)
- 1: Original position – the position of the figures relative to the coordinate system remains unchanged (translation)

Parameters of the selected figure/hole

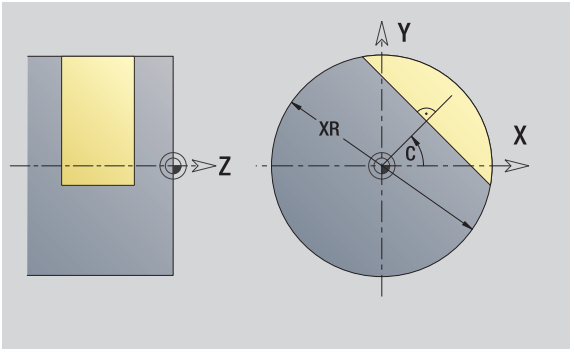
You can find the **reference diameter XR** with the "select reference plane" function (see page 411).

ICP generates:

- The section code SURFACE_Y with the reference diameter and spindle angle. The section code is omitted for nested contours.
- A G308 with the parameters contour name and milling depth (-1*BT).
- A G482 with the parameters of the pattern.
- The G function and the parameters of the figure/hole.
- A G309.

Single surface in YZ plane

This function defines a surface in the YZ plane.



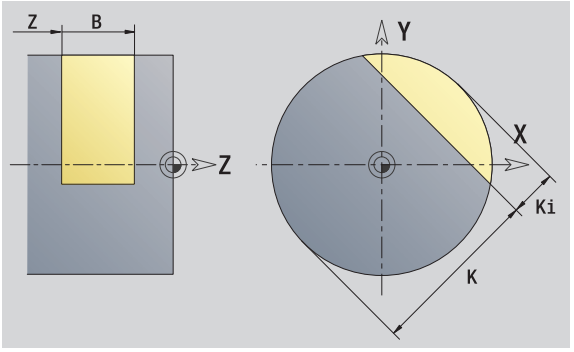
Reference data of the single hole

ID	Contour name
C	Spindle angle (position angle of the surface normal)
XR	Reference diameter

Parameters of the single surface

Z	Reference edge
Ki	Depth
K	Residual depth
B	Width (reference: reference dimension ZR)

- B<0: Surface in negative Z direction
- B>0: Surface in positive Z direction



You can switch between depth (Ki) and residual depth (K) by pressing a soft key (see table at right).

You can find the **reference diameter XR** with the "select reference plane" function (see page 411).

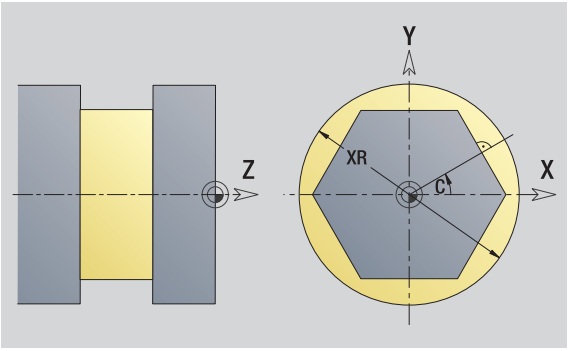
ICP generates:

- The section code SURFACE_Y with the reference diameter and spindle angle. The section code is omitted for nested contours.
- A G308 with the parameter contour name.
- A G386 with the parameters of the single surface.
- A G309.

Soft key	
Residual depth	Switches the field to entering the residual depth K.

Centric polygons in YZ plane

This function defines centric polygons in the YZ plane.



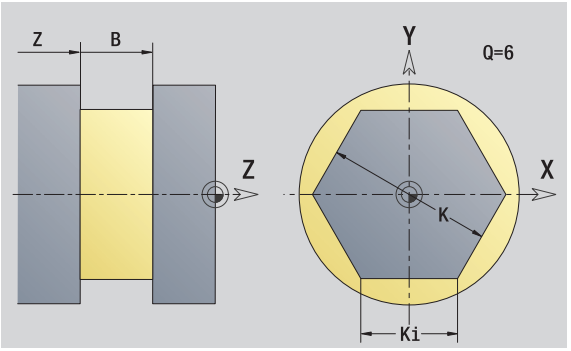
Reference data of the polygon

ID	Contour name
C	Spindle angle (position angle of the surface normal)
XR	Reference diameter

Parameters of the polygon

Z	Reference edge
Q	Number of sides (Q>=2)
K	Width across flats
Ki	Length of side
B	Width (reference: reference dimension ZR)

- B<0: Surface in negative Z direction
- B>0: Surface in positive Z direction



You can switch between edge length (Ki) and width across flats (K) by pressing a soft key (see table at right).

You can find the **reference diameter XR** with the "select reference plane" function (see page 411).

ICP generates:

- The section code SURFACE_Y with the reference diameter and spindle angle. The section code is omitted for nested contours.
- A G308 with the parameter contour name.
- A G487 with the parameters of the polygon.
- A G309.

Soft key	
	Switches the field to entering the width across flats K.

5.16 Loading existing contours

Integrating cycle contours in smart.Turn

ICP contours that you have created **for cycle programs** can be loaded in smart.Turn. ICP converts the contours into G commands and integrates them in the smart.Turn program. The contour is now part of the smart.Turn program.

The ICP editor considers the type of contour. For example, you can load a contour defined for the face only if you have selected the face (C axis) in smart.Turn.

Activate the ICP editor.

Contour list

Press the **Contour list** soft key. The ICP editor opens the window "Selection of ICP contours."

Next file type

Press the **Next file type** soft key until the cycle contours are displayed (see file extension table at right).

Select the file.

Open

Load the selected file.

- **Blank or finished part contour:** Supplement or adapt the contour, if necessary.
- **C-axis contour:** Complete the reference data.

Extension	Group
*.gmi	Turning contours
*.gmr	Contours of workpiece blanks
*.gms	Milling contours on face
*.gmm	Milling contours on lateral surface

DXF contours (option)

Contours that exist in DXF format are imported with the ICP editor. You can use DXF contours both for cycle operation and for smart.Turn.

Requirements of a DXF contour:

- Only two-dimensional elements
- The contour must be in a separate layer (without dimension lines, without wraparound edges, etc.).
- Depending on the setup of the lathe, contours for turning operations must be either in front of or behind the workpiece.
- No complete circles, no splines, no DXF blocks (macros), etc.

Contour preparation during the DXF format: Since the DXF format is fundamentally different from the ICP format, the contour is converted from DXF to ICP format during the import. The process makes the following changes:

- Polylines are transformed into linear elements:
- Gaps between contour elements that are < 0.01 mm are closed.
- Open contours are described "from right to left" (starting point: right).
- The starting point on closed contours is specified according to internal rules.
- The rotational direction for closed contours is counterclockwise.

Activate the ICP editor.

Contour list

Press the **Contour list** soft key. The ICP editor opens the window "Selection of ICP contours."

Next file type

Press the **Next file type** soft key until the DXF contours are displayed (see file extension "*.DXF").

Select the file.

Open

Open the selected file.

Next contour

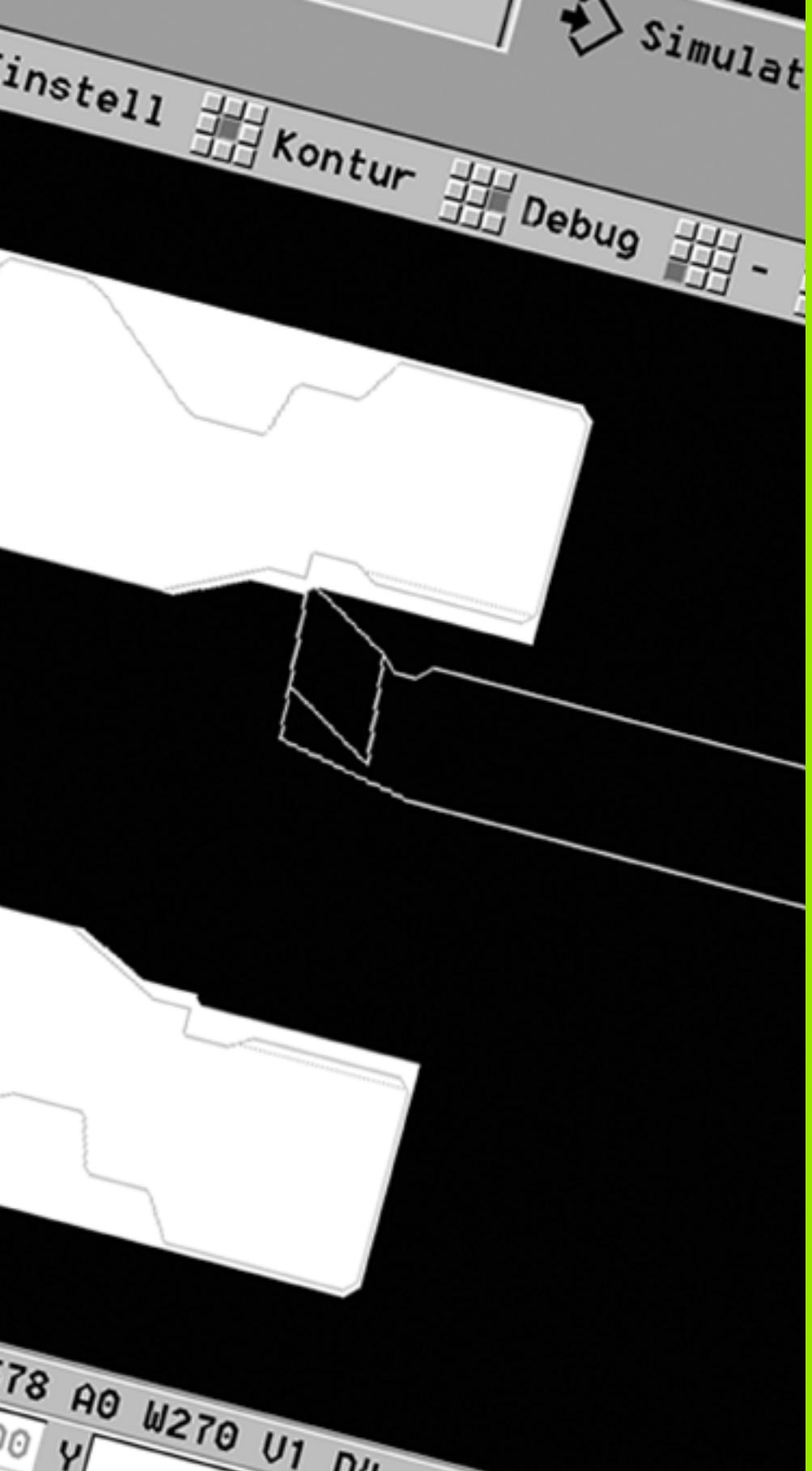
Previous contour

Select the DXF layer.

✓

Load the selected contour.

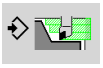
- **Blank or finished part contour:** Supplement or adapt the contour, if necessary.
- **C-axis or Y-axis contour:** Complete the reference data.



6

Graphic simulation

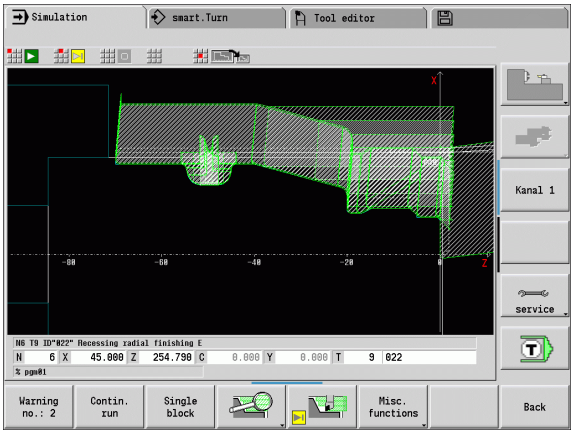
6.1 Simulation mode of operation

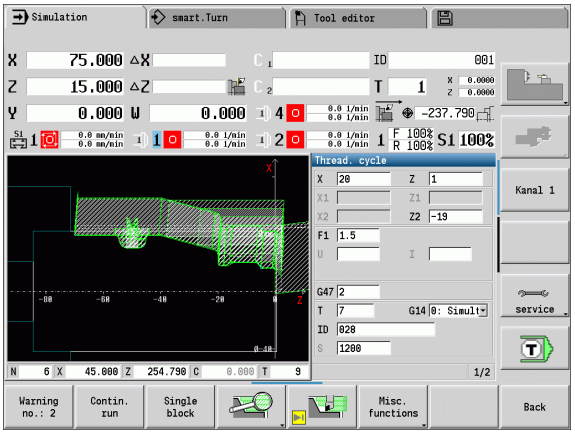


Press this soft key to start a graphic simulation from the following operating modes:

- smart.Turn
- Program Run
- Teach-in
- Manual Operation (cycles)

When called from the smart.Turn mode, the graphic simulation opens the **large** simulation window and loads the selected program. When you call the simulation from the Machine modes of operation, either the **small** simulation window or the last window you used is opened.





Large simulation window

- **Menu line** for controlling the simulation through the numeric keypad
- **Simulation window:** Displays the workpieces and the tool movements. The simulation supports the simultaneous display or multiple views in the simulation window. In the "window selection," select the following views:
  - XZ view (turning view)
  - XC view (face view)
  - ZC view (lateral surface)
  - YZ view (for operations with the Y axis)
- **Displays:**
  - NC program blocks
  - NC block number, position values and tool information
  - Name of the NC program

Small simulation window:

- The machine display fields and the cycle dialog are not hidden and remain visible during the simulation of cycle programs.
- In the smart.Turn operating mode, the machine display is not hidden.
- You can set the following views by soft key:
  - XZ view (turning view)
  - XC view (face view)
  - ZC view (unrolled lateral surface)

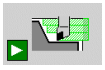
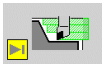
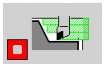


In the Program Run, Teach-in and Manual modes of operation, the simulation is started automatically for the current program. In smart.Turn, the program is only loaded. Press the soft key to start the simulation.

Using the graphic simulation

The simulation is controlled by soft keys in all operating states. In addition, you can always use the menu keys (numeric keys) to control the simulation, even in the small simulation window, when the menu bar is **not visible**.

Starting and stopping with soft keys

	Starts the simulation from the beginning. The soft key switches to the stop symbol; you can now use the soft key to stop and resume the simulation, depending on the simulation status.
	Resumes a stopped simulation (Single Block mode).
	The key indicates that a simulation is running at present. Pressing the key stops the simulation.

Starting and stopping with menu keys

	Starts the simulation from the beginning.
	Resumes a stopped simulation (Single Block mode).
	The key indicates that a simulation is running at present. Pressing the key stops the simulation.

Large and small simulation window

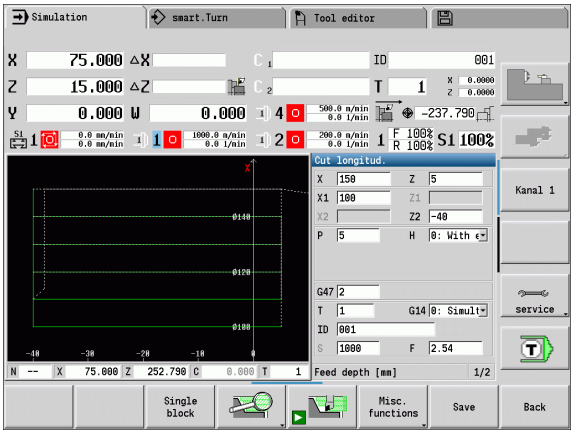


▶ This menu item switches between the small and large simulation window, even if the **menu bar is not visible**.

With the other menu items and the soft keys listed in the table, you define the sequence of the simulation, activate the magnify function, or use the miscellaneous functions to make settings for the simulation.

- You can operate the simulation with the numeric keys even if the menu bar is **not visible**.
- In the Machine modes of operation, use the numeric key **[5]** to switch between the small and large simulation windows.

- In the Machine modes of operation, an activation of the **Single Block** soft key is also effective in automatic mode.
- In the Machine modes of operation, you can start automatic program run directly from the simulation by pressing **Cycle on**.



Soft keys with active simulation window

Warning no.: 3	For calling warnings. If the interpreter issues warnings during simulation (e.g. "Material remains unmachined"), the soft key is activated and the number of warnings is shown. Each time the soft key is pressed, it switches to the next warning.
Contin. run	In the "Continuous Run" mode, all cycles of the program are simulated in the program run mode without stopping.
Single block	In the Single Block mode, the simulation stops after each movement (basic block).
	Opens the soft-key menu for the zoom functions and displays the zoom frame (see "Adjusting the visible section" on page 476).
Misc. functions	Switches the menu **and** the soft-key row over to the miscellaneous functions.

The miscellaneous functions

You use the **miscellaneous functions** to select the simulation window, to determine how the tool path is depicted, or to call the time calculation.

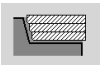
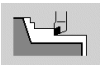
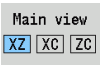
The tables provide you with an overview of the menu and soft-key functions.

Overview of the "miscellaneous functions" menu

	Selects the simulation window (see "Simulation window" on page 471).
	Activates a startup-block search (see "Simulation with mid-program startup" on page 478).
	Selects 3-D view (see "Simulation with mid-program startup" on page 478).
	Calls the time calculation (see "Time calculation" on page 480).
	Switches between the large and the small simulation window (see "Using the graphic simulation" on page 469).
	Switches between the single and multiple window display (see "Multiple window view" on page 472).
	Saves the contour (see "Saving the contour" on page 481).



Soft keys for the miscellaneous functions

	Switches between wire-frame graphics and cutting-path graphics.
	Switches between light-dot and cutting-edge view.
	Activates the 2-D material-removal graphic
	Selects the view.
	Switches the focus to the next window.

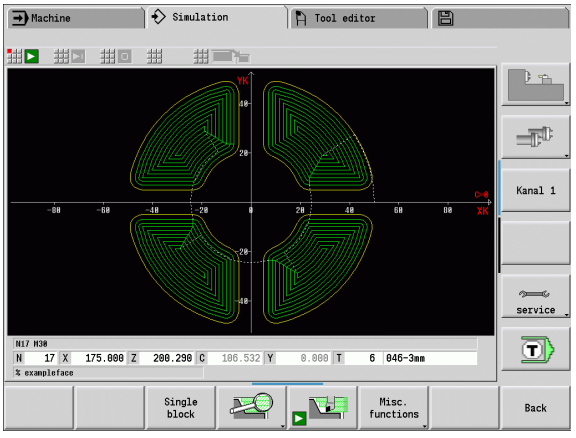
6.2 Simulation window

Setting up the views

With the simulation windows described in the following you check not only the turning work but also the drilling and milling operations.

- **XZ view (turning view):** The turning contour is depicted in the XZ coordinate system. The configured coordinate system is taken into account (tool carrier in front of/behind the turning center, vertical lathes).
- **XC view (face view):** The displayed coordinate system is displayed as a Cartesian system with the axis designations **XK** (horizontal) and **YK** (vertical). The angle position C=0° is on the XK axis, the positive direction of rotation is counterclockwise.
- **ZC view (lateral surface view):** The contour display and traverse-path display are oriented to the position on the unrolled lateral surface and the Z coordinates. The upper and lower lines of this workpiece correspond to the angular positions C=–180°/+180°, respectively. All drilling and milling operations are within the range –180° to +180°.
  - **Cycle program or DIN program with workpiece blank definition:** The basis for the unrolled workpiece surface is the dimensions of the programmed workpiece.
  - **Cycle program or DIN program without workpiece blank definition:** The basis for the unrolled workpiece surface is the dimensions of the "standard blank" (user parameter: "simulation > definition of the standard workpiece blank size").
  - **Single cycle or Teach-In:** The basis of the unrolled workpiece surface is the section defined by the respective cycle (expansion in Z and limiting diameter X).
- **YZ view (side view):** The contour and traverse path are shown in the YZ plane. The side view depicts only the Y and Z coordinates—not the spindle position.





Front face and surface windows operate with a fixed spindle position. Whereas the machine turns the workpiece, the graphic simulation moves the tool.

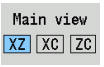
Single-window view

Single-window view

Only one view is shown in the small simulation window. You switch the view with the **Main view** soft key. You can also use this soft key when only one view is set in the large simulation window.

With cycle programs, you can activate the face or lateral surface view only when the C axis is used in the program.

Soft key for selecting the view



Select the view:

- Turning view XZ
- Face view XC
- Lateral surface view ZC

Multiple window view

Activating the multiple window view (possible only in the large simulation window):



▶ Switch the menu bar to "Miscellaneous functions."



▶ Select the "Window" menu item (in the large simulation window).

▶ Set the desired window combination.

▶ Set the path display in the auxiliary windows.

The path display in the supplementary windows: The front face window, surface window, and YZ view are considered supplementary windows. The following setting specifies whether the simulation depicts traverse paths in the windows.

- **Automatic:** The simulation displays traverse paths if the C axis has been oriented or a G17 or G19 has been executed. G18 or a C axis out of orientation stops output of the traverse paths.
- **Always:** The simulation shows the traverse path in all simulation windows.

In the multiple window view, a window is indicated with a green frame. This window has the "focus," which means that magnification settings and other functions apply to this window.

Switching the focus



▶ Press the soft key (or the GOTO key) several times until the focus is on the desired window.

Switching between the single and multiple window display:



▶ Select the menu item (or the decimal point key) to switch from the multiple-window to single-window view. The window with the green frame is shown as a single window view.



▶ Pressing the menu item again (or the decimal key) switches back to the multiple window view.

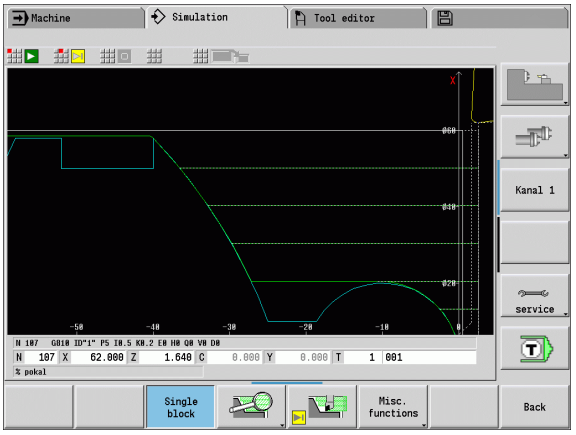
6.3 Views

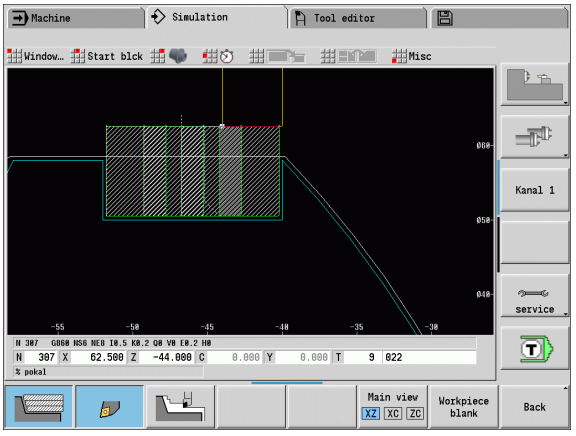
Traverse path display

Rapid traverse paths are shown as a broken white line.

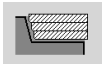
Feed paths are displayed either as a line or as a cutting trace, depending on the soft-key setting:

- **Line display:** A solid line describes the path of the theoretical tool tip (wire-frame graphics) The wire frame display is particularly convenient if you only need a quick overview of the proportioning of cuts. The path of the theoretical tool tip, however, is not identical with the contour of the workpiece. This view is therefore not as suitable if you wish to run a thorough check of the machined contour. In the CNC, this "falsification" is compensated by the cutting radius compensation.
- **Cutting path display:** The simulation uses hatch marking to depict the surface covered by the cutting area of the tool. The cutting path graphic accounts for the exact geometry of the tool tip (cutting radius, cutting width, tool-tip position, etc.). You can check in the simulation whether the contour is machined completely or needs to be reworked, whether the contour is damaged by the tool or overlaps are too large. The cutting path graphics is especially useful for recessing or drilling operations as well as for machining slopes where the tool shape has an essential influence on the accuracy of the resulting workpiece.





Activating the cutting path graphics:



▶ With the soft key activated, the cutting paths of traverse are shown.

The "Simulation/General settings/Traverse delay" user parameters are used to influence the simulation speed.

Tool depiction

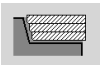
You adjust by soft key whether the tool cutting edge or the light dot is shown (see table at right).

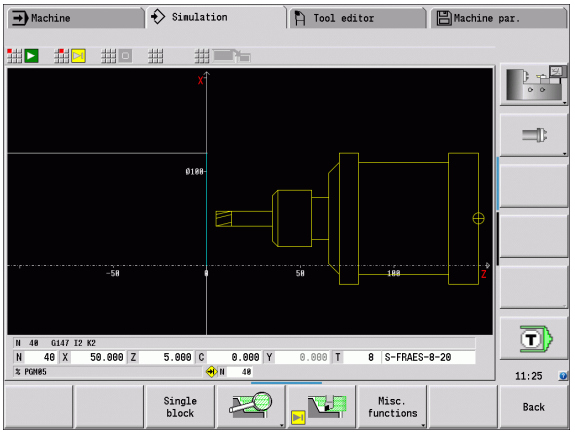
- The **tool cutting edge** is shown with the correct angles and cutting radius, as defined in the tool database.
- **Light-dot view:** A white square (light dot) is shown at the currently programmed position. The light dot represents the position of the imaginary cutting edge.

Depicting the tool holder during machining simulation

The control can depict the associated tool holder with the corresponding dimensions in addition to the tool's cutting edge. The requirements for this are:

- Creating a new tool holder in the holder editor or selecting an existing holder
- Describing the tool holder with the required parameters (type, dimensions and position)
- The appropriate tool holder must be assigned to the tool (HID)

Soft keys for miscellaneous functions	
	Switches between wire-frame graphics and cutting-path graphics.
	Switches between light-dot and cutting-edge view.

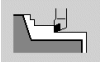


Material-removal graphic

The material-removal graphics shows the workpiece blank as a "filled surface." When the tool tip passes through the workpiece blank, the workpiece area covered by the tool is erased in the graphic.

The material-removal graphic mode shows all paths of traverse according to the programmed speed. The 2-D material-removal graphic mode is only available in side view (XZ). You activate this type of simulation by soft key (see table at right).

You can change the speed of simulation in the metal-removal graphic by using the keys in the table at right.

Soft keys for miscellaneous functions	
	Activates the 2-D material-removal graphic

Menu for the material-removal graphic	
	Slows the material-removal graphic.
	Material removal graphic at the programmed feed rate
	Accelerates the material-removal graphic.

3-D view

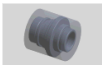
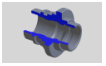
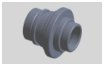


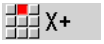
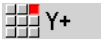
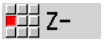
▶ The "3-D View" menu item switches to a perspective view.

With the 3-D view, you can display the workpiece, auxiliary contour and finished part in a solid-model view. If there are two or more auxiliary contours in the program, they are displayed by pressing the "auxiliary contour" soft key repeatedly. The "workpiece" display shows the defined workpiece blank or the workpiece after respective stages of machining.

With the menu functions you can rotate the graphic around the principal axes X, Y and Z. The "perspective view" soft key resets the graphic to its initial condition.

The workpiece, auxiliary contour, and finished-part soft keys are displayed regardless of the content of the program.

Soft keys for miscellaneous functions	
	Displays a transparent workpiece.
Workpiece	Displays the workpiece.
Finished part	Displays the finished part.
Auxiliary contour	Displays an auxiliary contour.
	Displays a section view.
	Selects the front view.
	Selects the perspective view.

Menu assignment for 3-D view	
	Rotate graphics in positive direction around the X axis.
	Rotate graphics in positive direction around the Y axis.
	Rotate graphics in positive direction around the Z axis.
	Rotate graphics in negative direction around the X axis.
	Rotate graphics in negative direction around the Y axis.
	Rotate graphics in negative direction around the Z axis.

6.4 The zoom function

Adjusting the visible section



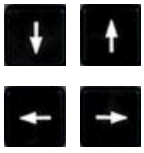
Press this soft key to activate the zoom function. With the zoom menu, you can modify the section displayed in the simulation window. As an alternative to the soft keys, you can use the **cursor keys** and the **PG DN** and **PG UP keys** to change the visible section.

For cycle programs, and when a program is simulated for the first time, CNC PILOT automatically selects the displayed section. When you simulate the same smart.Turn program again, the last selected section is used.

In the multiple window view, the zoom function applies to the window with a green frame.

Modifying the section by using keys

- You can use the following keys to change the visible section without opening the zoom function:

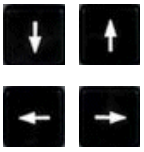
Keys for modifying the displayed section	
	The arrow keys move the workpiece in the indicated direction.
PG DN	Reduces the displayed workpiece (zoom –).
PG UP	Magnifies the displayed workpiece (zoom +).



Soft keys in the zoom menu	
Delete paths	■ Deletes all traverse paths drawn so far. ■ The workpiece blank regeneration is active, the workpiece blank is updated and redrawn. ■ Closes the zoom menu.
Extend view	Extends the displayed section directly (zoom –).
Zoom off	Switches back to the standard screen section and closes the zoom menu.
Last zoom	Switches to the last section selected.
Take over	Applies the area selected by the red frame as the new section and closes the zoom menu.
Back	Closes the zoom menu without changing the selected section.

Modifying the section with the zoom menu

- When you activate the zoom menu, a red frame is shown in the simulation window. This red frame indicates the zoom area, which you can select using the **Take over** soft key or the **Enter** key. Use the following keys to change the frame size and position:

Keys for modifying the red frame	
	The arrow keys move the red frame in the indicated direction.
	Reduces the size of the red frame.
	Increases the size of the red frame.

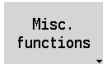
6.5 Simulation with mid-program startup

Startup block with smart.Turn programs

smart.Turn programs are always simulated from the first block, regardless of which block the cursor is in. If you use the mid-program startup, the simulation suppresses the display up to the startup block. If there is a workpiece blank, the simulation scans the blocks up to this position, updates the blank and redraws it.

The simulation then draws the traverse paths starting from the startup block.

Activating a mid-program startup:



▶ Switch the menu bar to "Miscellaneous functions."



▶ Select the "Start block" menu item.



▶ Enter the number of the startup block and confirm.



▶ Go back to the main menu of the simulation.



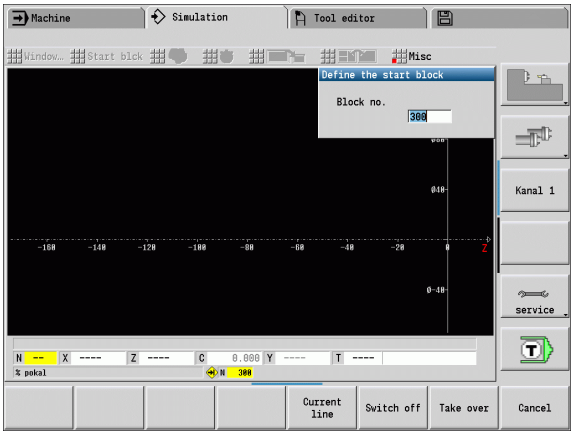
▶ Start the simulation. The CNC PILOT scans the NC program up to the startup block, regenerates the blank and stops at this position.

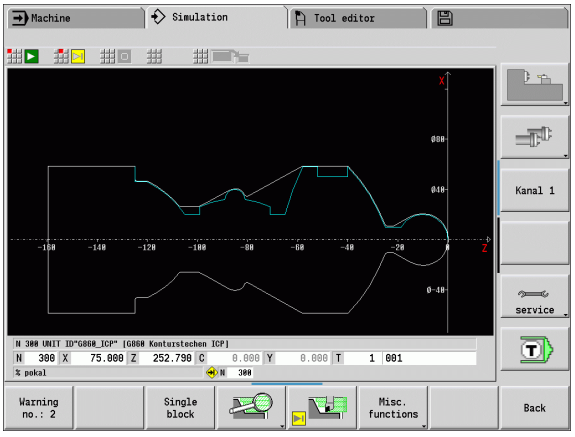


▶ Continue the simulation.

The block number for startup is shown in the lowest line of the display. The field for the startup block and the block number in the display have a yellow background as long as the simulation is conducting the mid-program startup.

Mid-program startup remains on even if you cancel the simulation. If you restart the simulation after cancelation, it stops at the section code MACHINING. This enables you to change settings before continuing the simulation.





Soft keys for mid-program startup

Current line	Loads the NC block number in the display as the startup block.
Switch off	Switches off the mid-program startup.
Take over	Loads the defined starting block and activate the mid-program startup.
Cancel	Cancels the mid-program startup.

Mid-program startup in cycle programs

For cycle programs, you first place the cursor on a cycle and then call the simulation. The simulation begins with this cycle. All previous cycles are ignored.

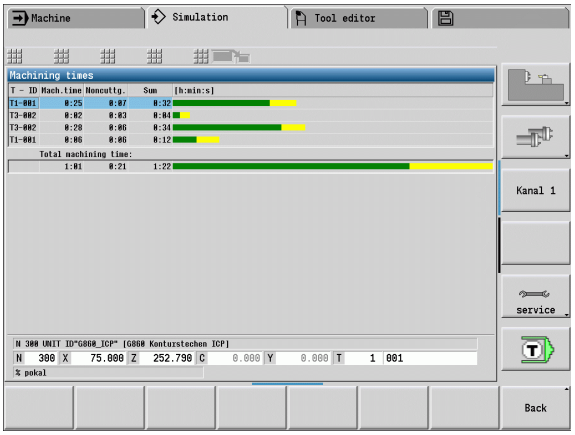
The **Start block** menu item is deactivated for cycle programs.

6.6 Time calculation

Showing the machining times

During simulation, the machining and idle-machine times are calculated. The machining times, idle times and total times are shown in the "Time calculation" table (green: machining times; yellow: idle times). If you are working with cycle programs, each cycle is shown in a separate line. In DIN programs, each line represents the use of a new tool (for each tool call with T).

If there are more table entries than fit on a screen page, you can call further time data with the **cursor keys** and **PG UP/PG DN**.



Calling the machining times:

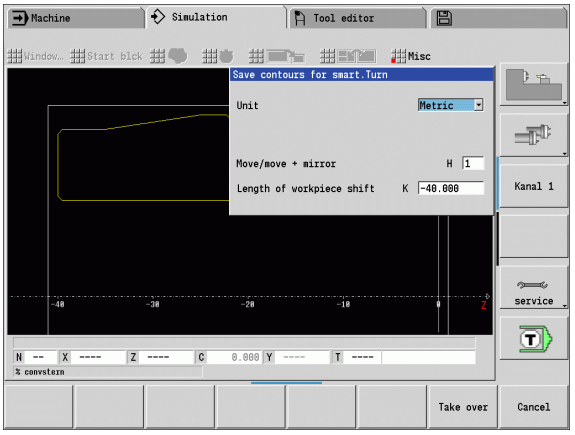
Misc. functions

▶ Switch the menu bar to "Miscellaneous functions."

▶ Call the time calculation.

6.7 Saving the contour

Saving the generated contour in the simulation



You can save a contour generated in the simulation and read it into smart.Turn. In smart.Turn, you insert into the program the workpiece blank and finished part contour that you generated during simulation. Select the "Insert contour" function on the "ICP" menu.

Example: You describe the blank form and finished part and simulate machining of the first setup. Then you save the machined contour and use it for the second setup.

During contour generation, the simulation saves:

- BLANK: The simulated contour's state of progress
- FINISHED PART: The programmed finished part

The simulation takes into account a shift of the workpiece zero point and/or a mirror image.

Saving the contour:

Misc. functions

▶ Select "miscellaneous functions" soft key

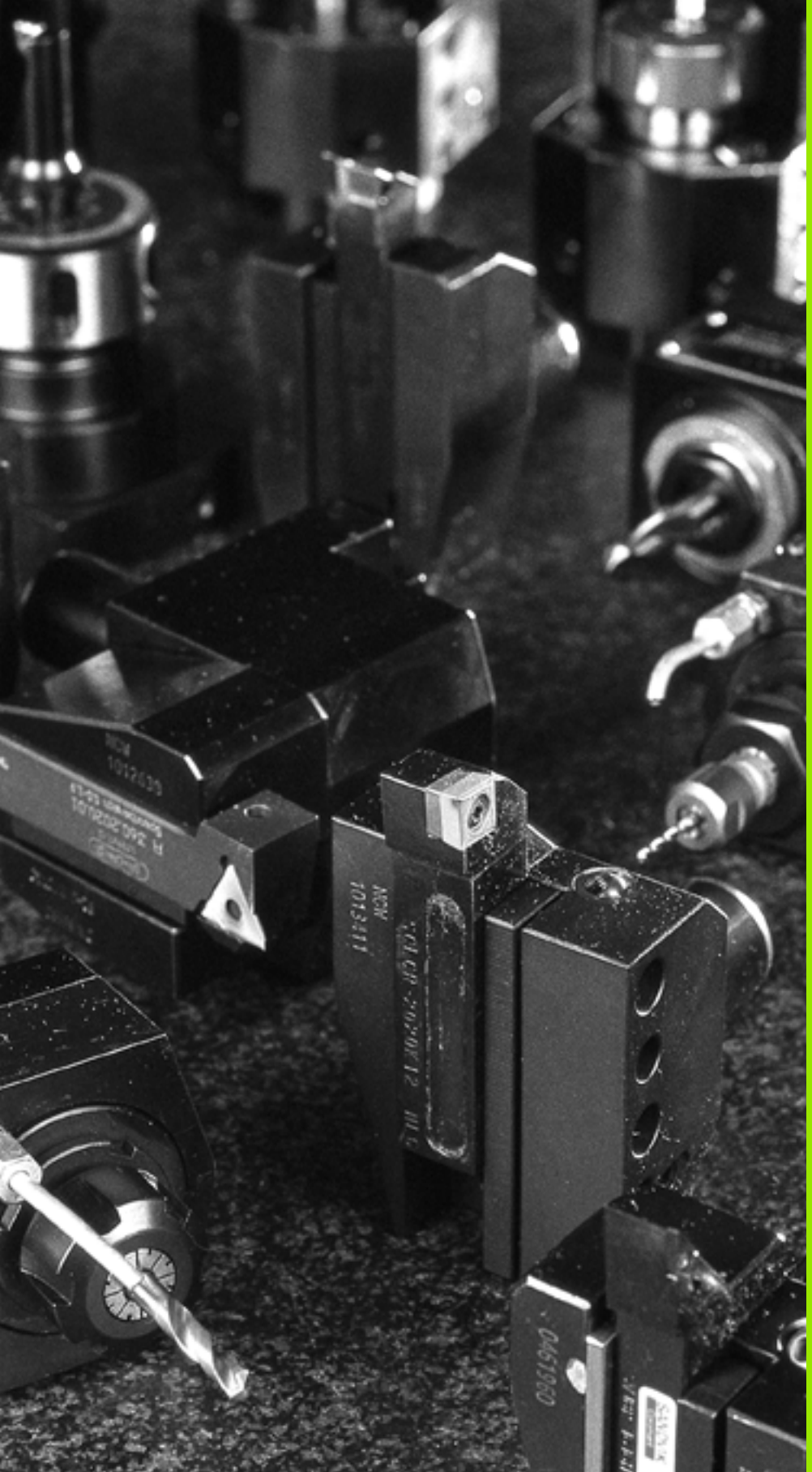


▶ Select the "Misc." menu



▶ Select the "Save contour" menu

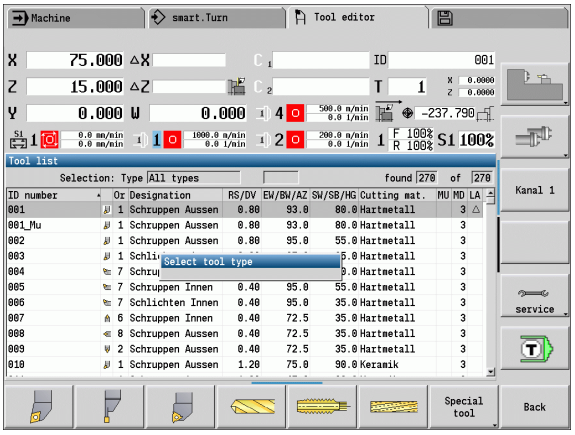
▶ The control opens a dialog box enabling you to define the following input fields:
- Unit: Contour description metric or in inches
- Shift: Shifting the workpiece zero point
- Mirroring: Mirror/do not mirror the contours



7

Tool and technology database

7.1 Tool database



You usually program the coordinates for the contour by taking the dimensions from the drawing. To enable the CNC PILOT to calculate the slide path, compensate the cutting radius and determine the number of cutting passes, you need to enter the tool length, cutting radius, tool angle, etc.

The CNC PILOT can save tool data for up to 250 tools (optionally 999), whereby each tool is identified with a number (ID code). For each tool, you can enter an additional tool description which makes it easier to find the tool data again when needed.

The Machine mode has functions for determining the tool length dimensions (see "Tool measurement" on page 99).

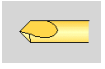
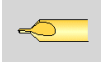
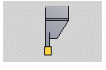
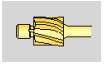
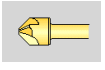
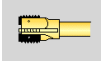
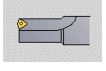
Wear compensation is managed separately. This allows you to enter new compensation values at any time, even during program run.

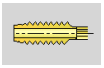
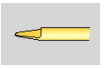
You can also assign a **cutting material** to the tools, which gives you direct access to the terminology database (feed rate, cutting speed). This saves you a lot of time since you only need to determine and enter the cutting data once.

Tool types

Tools for drilling, recessing, finishing, etc., have very different shapes. Therefore, the reference points for determining the tool length and other tool data also vary.

The following table provides an overview of the tool types.

Tool types		Tool types	
	Standard turning tools (Page 500) ■ Roughing tools ■ Finishing tools		■ NC center drills (Page 504)
	■ Button tools (Page 500)		■ Centering tools (Page 505)
	Recessing tools (Page 501) ■ Recessing tools ■ Parting tools ■ Recess-turning tools		■ Counterbores (Page 506)
			■ Countersinks (Page 507)
	■ Thread-cutting tools (Page 502)		■ Standard milling tools (Page 509)
	■ Twist drills (Page 503)		■ Thread mills (Page 510)
	■ Indexable-insert drills (Page 503)		■ Angle cutters (Page 511)

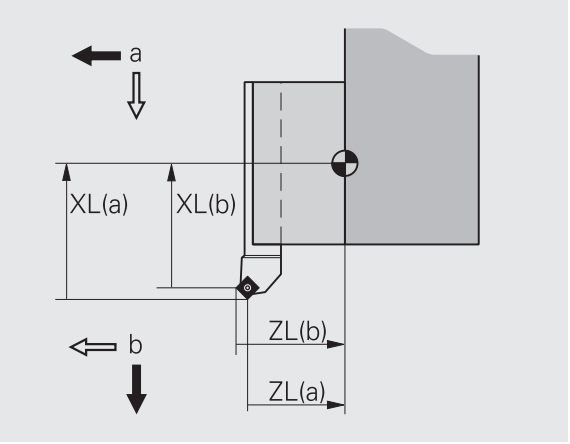
Tool types	Tool types
 ■ Taps (Page 508)	 ■ Milling pins (Page 512)
 ■ Touch probes (Page 513)	

Multipoint tools

A multipoint tool is a tool with multiple cutting edges or multiple reference points. A data record is created for every cutting edge or every reference mark. Then all data records of the multipoint tool are linked together.

For every data record of a multipoint tool, the position within the multipoint tool chain is shown in the tool list in the column "MU." Counting begins with byte “0.”

The figure at right shows a tool with two reference points.



Tool life management

The control can "count" either the machining time of a tool (i.e. the time a tool is traversed at the programmed feed rate) or the number of parts that were produced with that tool. These two options are used for tool life management.

As soon as the tool life expires or the programmed quantity is reached, the system interrupts machining and asks you to replace the tool or cutting edge. The machining operation, however, is not interrupted until the workpiece that is currently being produced is finished.

7.2 Tool editor

Tool list

In the tool list, the CNC PILOT displays important parameters and the tool descriptions. You can recognize the tool type and the tool orientation from the provided sketch of the tool point.

Sorting the tool list

Sort by ID / Type

▶ The tool list switches between sorting by ID number and by tool type (and orientation)“.

Reverse sorting

▶ The tool list switches between ascending and descending sorting.

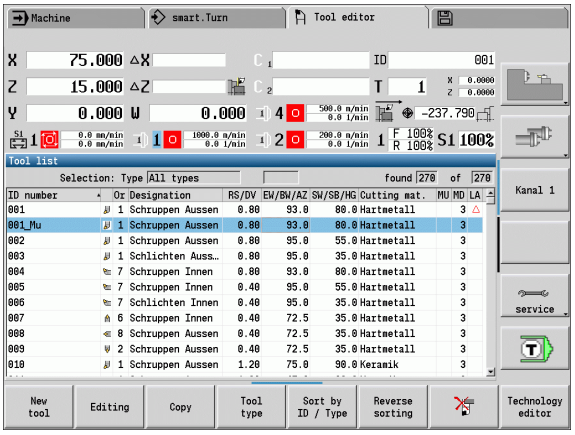
Displaying the entries of only one tool type.

Tool type

▶ Press the soft key and select the tool type in the following soft-key rows.

▶ The CNC PILOT creates a list containing only tools of the selected type.

You can navigate within the tool list with the **arrow keys** and **PG UP/PG DN** to check the entries.



Soft keys for tool organization

Tool type	Opens the **soft-key menu** to select the tool type.
Sort by ID / Type	Sorts the tool list as desired by tool type or ID number.
Reverse sorting	Switches between ascending and descending sorting.

Editing the tool data

Adding a new tool

New tool

- Press the soft key
- Select the tool type (see soft-key table at right).
- The CNC PILOT opens the input window.
- First assign the ID number (1 to 16 places, alphanumeric) and specify the tool orientation.
- Enter further parameters.
- Assign a tool text (see Page 488).

The CNC PILOT does not show the help graphics for individual parameters until the tool orientation is known.

Creating new tools by copying

- Place the cursor on the desired entry.

Copy

- Press the soft key. The CNC PILOT opens the input window with the tool data.
- Enter a new **ID number**. Check and edit the other tool data.

Save

- Press the soft key. The new tool is added to the database.

Editing the tool data

- Place the cursor on the desired entry.

Editing

- Press the soft key. The tool parameters are provided for editing.

Deleting an entry

- Place the cursor on the entry to be deleted.



- Press the soft key and answer the security prompt with **Yes**.

Soft keys for tool organization

New tool	Opens the following type selection for adding a new tool.
	
Special tools:	
	
	Type selection for special drilling tools:
	
	Type selection for special milling tools:
	
	Type selection for touch probes:
	
Editing	Opens the dialog box for the selected tool.
Copy	Copies the selected tool and creates a new tool with the copied data.
	**Deletes** the selected tool after you confirm a dialog prompt from the database.
Technology editor	**Opens** the technology editor (see Page 514).

Tool texts

Tool texts are assigned to the tools and displayed in the tool list. The CNC PILOT manages the tool texts in a separate list.

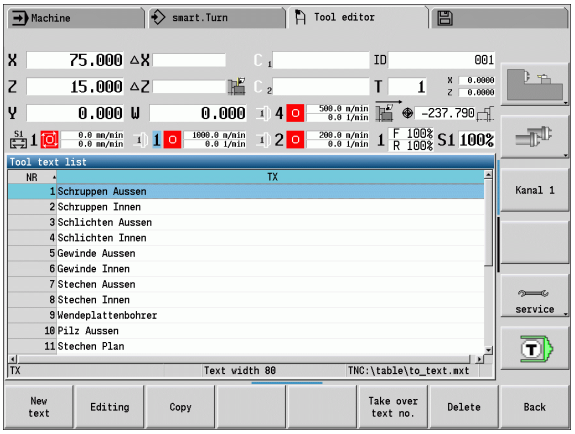
Connections:

- The descriptions are managed in the **tool text** list. Each entry is preceded by a "QT number."
- The parameter "Tool text QT" contains the reference number for the "tool text" list. The text indicated by QT is then displayed in the tool list.

When the tool dialog box is open, CNC PILOT lets you enter tool texts. Press the **Tool texts** soft key.

You can define up to 999 tool texts of max. 80 characters length.

- New texts are inserted in the next free line after the cursor position.
- When deleting and editing a tool text, please remember that the text might be used for more than one tool.



Soft keys in tool list

New text	Generates a new line in the text list and opens it for text input.
Editing	Opens the selected tool text for editing. Press ENTER to confirm.
Copy	Copies the currently selected tool text to a new text line. This creates a new tool text.
Take over text no.	Transfers the text number to the tool dialog box as a reference and exits the tool text editor.
Delete	Deletes the selected tool text after a confirmation prompt.
Back	Closes the tool text editor and returns to the tool dialog box without applying any changes to the text references.

Editing multipoint tools

Creating multipoint tools

For each cutting edge, or each reference point, make a separate data record with the tool description.

Place the cursor on a first cutting edge.

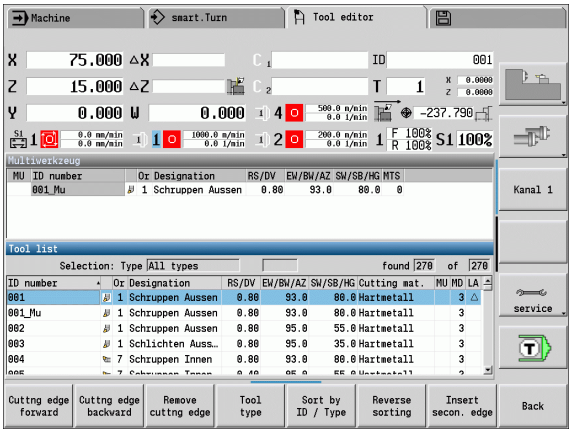
Editing	Press the soft key.
Multipoint tool	Press the soft key. The tool editor considers this cutting edge to be the main cutting edge (MU=0).

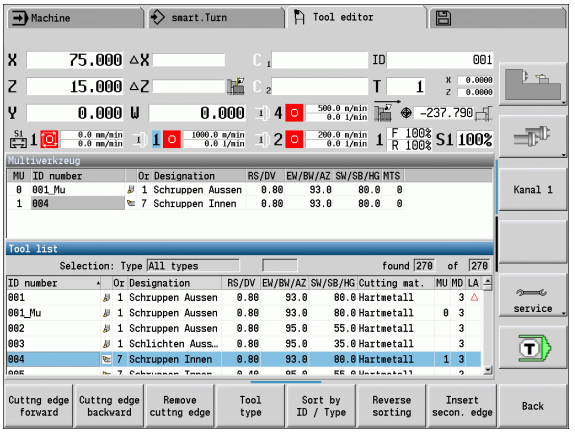
Place the cursor on a next cutting edge.

Insert secon. edge	Press the soft key. The tool editor adds this cutting edge to the multipoint tool chain.

Repeat these steps for any further cutting edges of the multipoint tool.

Back	Press the soft key.





Removing a cutting edge from the multipoint tool

Place the cursor on a cutting edge of the multipoint tool.

Editing	Press the soft key.
Multipoint tool	Press the soft key. The tool editor lists all cutting edges of the multipoint tool.
Cuttng edge forward Cuttng edge backward	Select the cutting edge.
Remove cuttng edge	Remove the cutting edge from the multipoint tool chain.

Removing complete multipoint tools

Place the cursor on a cutting edge of the multipoint tool.

Editing

Press the soft key.

Multipoint tool

Press the soft key. The tool editor lists all cutting edges of the multipoint tool.

Cuttng edge forward

Cuttng edge backward

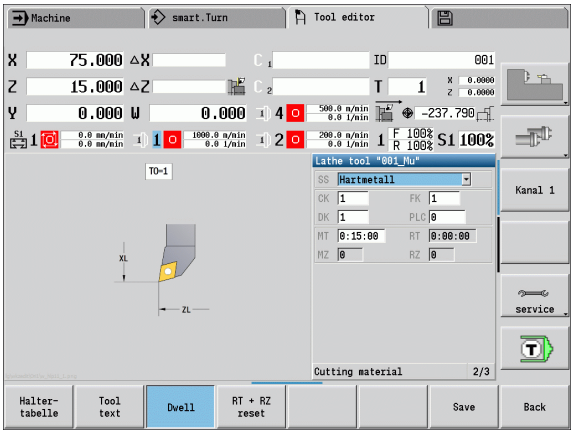
Place the cursor on a cutting edge no. 0 of the multipoint tool.

Remove cuttng edge

The multipoint tool chain is removed.

Editing tool-life data

The CNC PILOT counts the tool age in RT and the quantity of finished parts in RZ. When the predefined tool age or the part quantity limit has been reached, the tool is considered to be worn out.



Setting a limit to tool life

Dwell

Set the soft key to "Tool life." The tool editor opens the **Tool life MT** input field for editing.

Enter the tool life in the "h:mm:ss" format (h=hours; m=minutes; s=seconds). You can use the cursor keys to switch between hours, minutes and seconds.

Entering the quantity

No.pieces

Set the soft key to "No. of pieces." The tool editor opens the **No. of pieces MZ** input field for editing.

Enter the limit quantity, i.e. the number of workpieces that can be machined with one cutting edge.

New cutting edge

Insert a new cutting edge.

Call the corresponding data record in the tool editor.

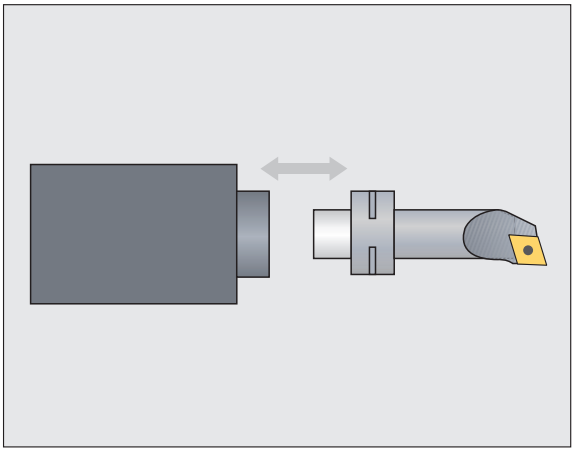
RT + RZ reset

Press the soft key. The tool life/limit quantity is reset to zero.

- Tool life monitoring is activated/deactivated in the **Tool life monitoring** user parameter Page 523.
- The quantity is added when the end of the program has been reached.
- Tool life / quantity monitoring is also continued after a change of program.

Manual change systems

Your machine must be configured by the machine manufacturer if you want to use manual change systems. Refer to your machine manual.



A tool holder is designated as a manual change system if it can accommodate various tool inserts via an integral clamping device. Most clamping devices designed as polygon coupling enable rapid, position-precise replacement of tool inserts.

With a manual change system it is possible to insert tools that are not in the turret during a program run. For this, the control checks whether the called tool is in the turret or has to be loaded. If a tool change is required then the control interrupts the program run. After you have manually loaded the tool insert, confirm the tool change and continue the program run.

The following steps are necessary for using manual change systems:

- Create the tool holder in the holder table
- Select the tool holder in the turret assignment
- Enter the tool data for the manual change tool

Holder table

In the "to_hold.hld" holder table, define the holder type and the tool setting dimensions of the holder. Because the geometric information is currently only evaluated with holders of the "manual change system" type, the management of standard tool holders in the holder table is not required.

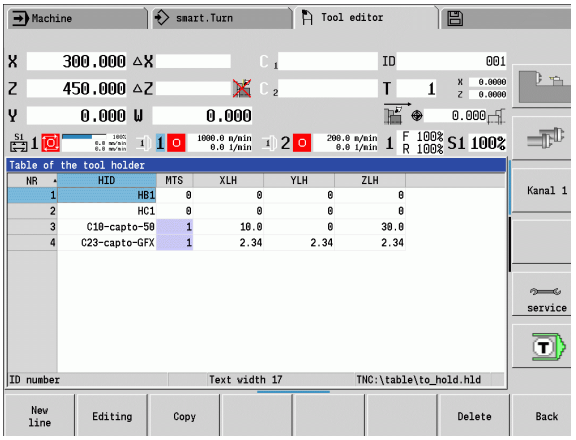
Process holder table in the tool editor:

Other tables

▶ Press the "Other tables" soft key

Holder editor

▶ Open the holder table: Press the "Holder Editor" soft key



The holder table contains the following entries:

NR Line number
HID ID number: Unique holder name (up to 16 characters)
MTS Manual change system:

- 0: Tool holder
- 1: Manual change system

ZLH Tool setting dimension in Z
XLH Tool setting dimension in X
YLH Tool setting dimension in Y

HC Holder type:

- A1: Boring bar holder
- B1: Right-hand, short design
- B2: Left-hand, short design
- B3: Right-hand, short design, overhead
- B4: Left-hand, short design, overhead
- B5: Right-hand, long design
- B6: Left-hand, long design
- B7: Right-hand, long design, overhead
- B8: Left-hand, long design, overhead
- C1: Right-hand
- C2: Left-hand
- C3: Right-hand, overhead
- C4: Left-hand, overhead
- D1: Multicarrier
- A: Boring bar holder
- B: Drill holder with coolant supply
- C: Square, longitudinal
- D: Square, transverse
- E: Rear-face machining
- E1: U drill
- E2: Cylindrical shank holder
- E3: Collet chuck holder
- F: Drill holder MK (Morse taper)
- K: Drill chuck
- T1: Driven, axial
- T2: Driven, radial
- T3: Boring bar holder
- X5: Driven, axial
- X6: Driven, radial

MP Tool position in turret:

- 0: –Z direction
- 1: –X/–Z direction
- 2: –X/+Z direction
- 3: +Z direction

WH Height of holder

WB Height of holder

AT Holder type

You can create a new holder with the "New line" soft key. The new line is always added at the end of the table.



You may only use ASCII characters in the holder table for holder names. Diacritics or Asian characters are not permitted.

You can also view and edit the holder table in opened tool forms. Use the 'Holder Editor" soft key for this purpose on the third form page (MTS entry).

If you use tool inserts in various manual change system holders, you need to manage the tool setting dimensions of the holder and tool insert separately. Enter the tool setting dimensions of the tool inserts into the tool table. Enter the tool setting dimensions of the manual change system holders in the holder table.

Entries for the standard tool holders are currently not evaluated. Thus management of the standard tool holders is not necessary.

Setting up the holder for manual change systems

Set up the manual change system holder in the turret assignment:

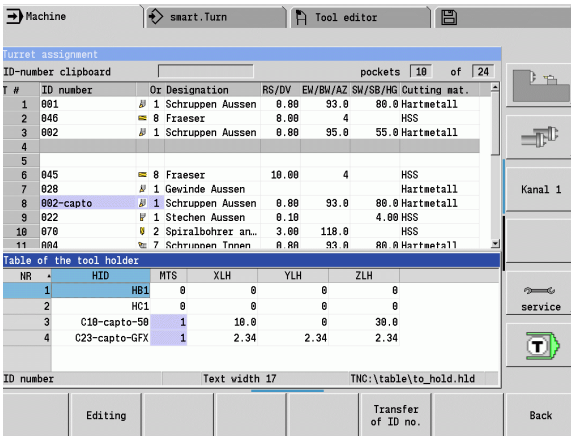
Turret list

Special Functions

Set-up the holder

Transfer of ID no.

- Select turret assignment: Press the "Turret list" soft key
- Select an unassigned turret pocket and press the "Special functions" soft key
- Open the holder table: Press the "Set up holder" soft key
- Select the holder and press the "Adopt ID no." soft key



After setting up a holder for a manual change system in the turret assignment, the first three fields of the specific line are color-highlighted.

You can remove a manual change system holder again with the "Remove holder" soft key.

in the turret assignment you can only set up the holder type **MTS 1** (manual change system). With holder type **MTS 0** (standard holder), the control displays an error message.

Selecting the manual change system in the tool data

Define the tool as a manual change tool in the tool data form:

Editing

- Open the tool data form: Press the "Edit" soft key
- On the third form page, select **MTS 1: MANUAL CHANGE TOOL**
- Confirm the entry by pressing the "Save" soft key

If you define a tool as a manual change system, in the tool list the tool type field (tool symbol) is color-highlighted.

With manual change tools you must not select a **HID** tool holder (empty field). Assignment of holder and tool is implemented via the turret assignment. A manual change system has to have been set up on the corresponding turret pocket for this.

With multipoint tools you must assign the **MTS** entry value for all cutting edges equally.

7.3 Tool data

General tool parameters

The parameters listed in the following table are available for all tool types. Parameters for specific tool types are described in the later chapters.

General tool parameters

ID ID number - Name of the tool (up to 16 characters)

TO Tool orientation (for identification number, see graphics)

XL Tool setting dimension in X

ZL Tool setting dimension in Z

DX Wear compensation in X (Range: –100 mm < DX < 100 mm)

DZ Wear compensation in Z (Range: –100 mm < DZ < 100 mm)

DS Special compensation (Range: –100 mm < DZ < 100 mm)

MD Direction of rotation (default: Not defined)

- 3: M3
- 4: M4

QT (Reference to) tool text

CW C tilting plane angle: Position of the C axis to determine the work position of the tool (machine-dependent function)

SS Cutting material (name of the cutting material for access to the technology database)

CK G96 compensation factor (default: 1)

FK G95 compensation factor (default: 1)

DK Depth compensation factor (default: 1)

PLC Additional information (see machine manual)

MT Tool life. Preset value for tool life monitoring (default: not defined)

MZ No. of pieces. Preset value for tool life monitoring (default: not defined)

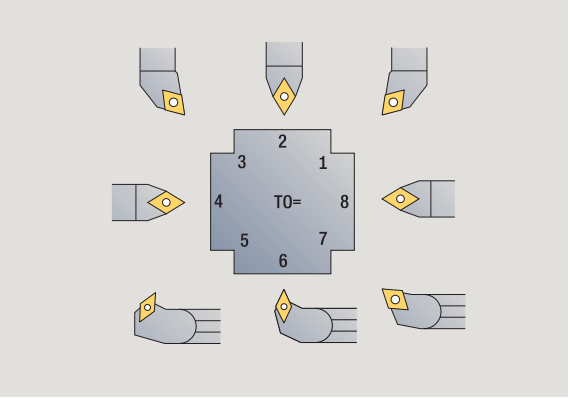
RT Display field for remaining tool life

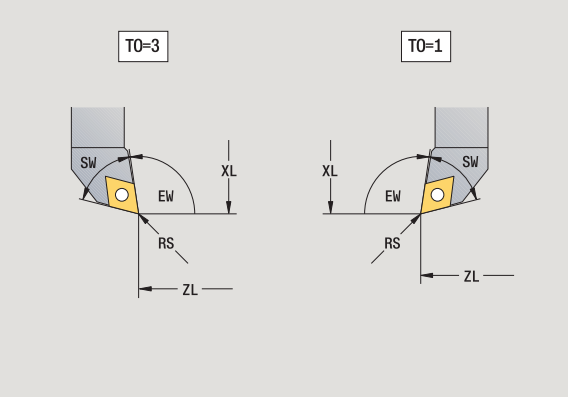
RZ Display field for remaining no. of pieces

HID ID number: Unique holder name (up to 16 characters)

MTS Manual change system:

- 0: Tool holder
- 1: Manual change system





Parameters for drilling tools

DV Drill diameter

BW Drill angle: Point angle of the drill

AW Driven tool : This parameter specifies for drilling and tapping tools during cycle programming whether switching commands are generated for the spindle or the driven tool.

- 0: Stationary tool
- 1: Driven tool

NL Usable length

RW Angle of orientation: Deviation from the main machining direction (input range: –90° to +90°)

AX Salient length in X

FH Chuck height

FD Diameter of the chuck

Description of the tool parameters

- **Identification number (ID):** The CNC PILOT needs a unique name for each tool. This "ID number" can have up to 16 alphanumeric characters.
- **Tool orientation (TO):** From the tool orientation, the CNC PILOT determines the position of the tool tip and, depending on the selected tool type, additional information such as the tool angle direction, reference-point position, etc. This information is necessary, for example, for calculating the cutting radius compensation, plunge angle, etc.
- The **setting dimensions (XL, ZL)** refer to the tool reference point. The position of the reference point depends on the tool type (see graphic support window).
- **Compensation values (DX, DZ, DS)** compensate for the wear of the cutting edge. For recessing and button tools, DS stands for the compensation value of the third side of the tool, which is away from the tool reference point. The cycles automatically shift to the special compensation. With G148, you can also switch to special compensation DS for single-path machining operations.
- **Direction of rotation (MD):** If you define a direction of rotation, a switching command (M3 or M4) is automatically generated for the spindle or, with driven tools, for the auxiliary spindle in all cycles that use this tool.



It depends on the PLC software of your machine whether the generated switching commands are evaluated. If the PLC does not execute the switching commands, they should not be defined. Refer to your machine manual.

- **Tool text (QT):** You can assign a tool text to each tool. The text is shown in the tool list. Because the tool texts are shown in a separate list, the reference to the text is entered in QT (see "Tool texts" on page 488).
- **Cutting material (SS):** This parameter is required if you want to use the cutting data from the technology database (see "Technology database" on page 514).
- **Compensation factors (CK, FK, DK):** These parameters serve for the tool-specific adjustment of the cutting values. The cutting data from the technology database are multiplied with the compensation factors before they are entered as default values.
- **Additional information (PLC):** You can find information on this parameter in the machine manual. This date can be used for machine-specific settings.
- **Tool life (MT, RT):** If you use the tool life management, specify in MT the tool life of the cutting edge. In RT the CNC PILOT shows the tool life already passed.
- **Quantity (MT, RT):** If you use the tool life management, specify in MZ the number of workpieces that can be machined with one tool cutting edge. In RZ the CNC PILOT shows the number of workpieces that have already been machined with this cutting edge.
- **Manual change system (MTS):** Definition of the tool holder



The tool life management and the quantity count are used alternatively.

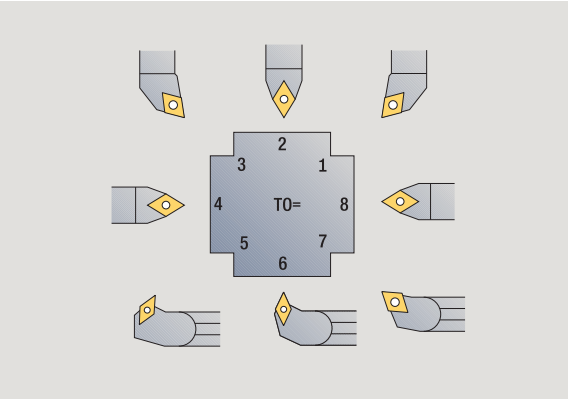
Standard turning tools

New tool

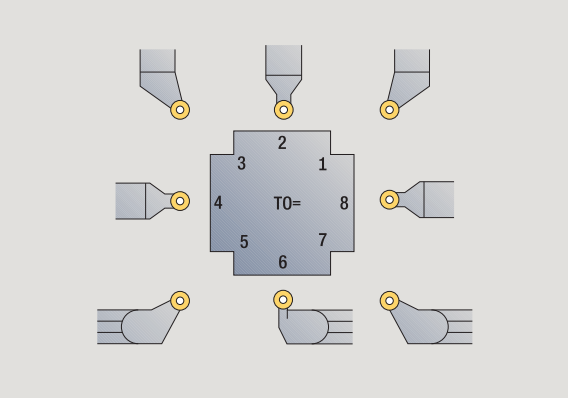
Select "New tool."

Select lathe tools.

For tools with round cutting edge, switch to dialog for button tools.



For tool orientations TO=1, 3, 5 and 7, you can enter the tool angle EW. The tool orientation values TO=2, 4, 6, 8 are used for **neutral tools**. "Neutral" tools are tools that are centered precisely in the tool tip. One of the setting dimensions for neutral tools refers to the center of the tool-tip radius.



Special parameters for roughing and finishing tools

CO Cutting edge position: The main machining direction of the tool influences the orientation of the tool angle **EW** and the point angle **SW** (required for AWG with TURN PLUS).

- 1: Longitudinal preferred
- 2: Transverse preferred
- 3: Only longitudinal
- 4: Only transverse

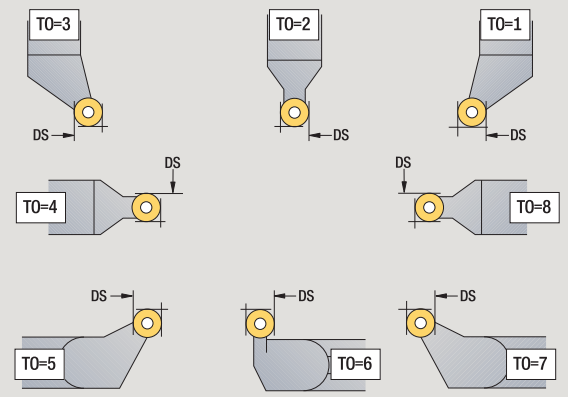
RS Cutting radius

EW Tool angle (range: 0° <= EW <= 180°)

SW Point angle (range: 0° <= SW <= 180°)

SUT Tool type (required for AWG in TURN PLUS)

For further tool parameters, see Page 497.



Special parameters for button tools

RS Cutting radius

EW Tool angle (range: 0° <= EW <= 180°)

DS Special compensation (position of the special compensation: see figure)

For further tool parameters, see Page 497.

The **wear compensation DX, DZ** compensates for wear on the two sides of the tool tip that lie next to the reference point. The **special compensation DS** compensates for wear on the third side of the tool tip.

Recessing tools

New tool

Select "New tool."

Select recessing tools.

Recessing tools are used for recessing, parting, recess turning and finishing (only smart.Turn).

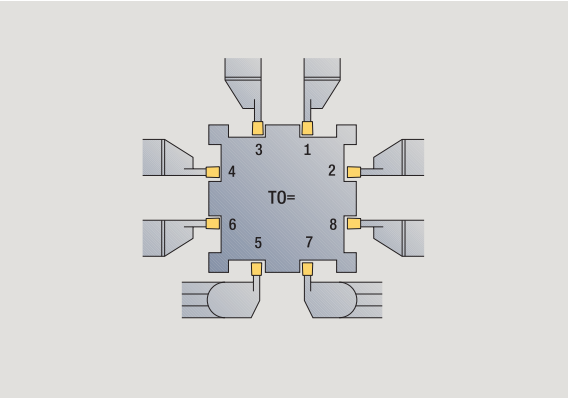
Special parameters for recessing tools

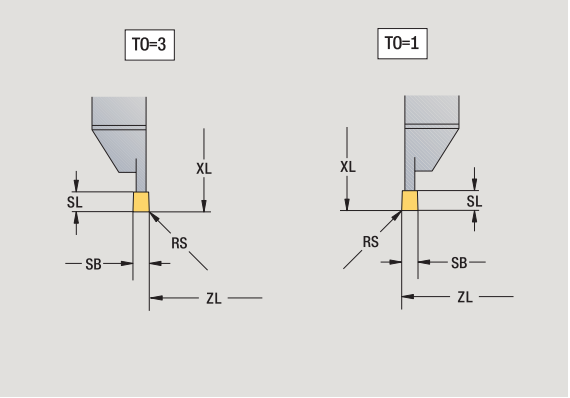
RS Cutting radius
SW Point angle
SB Cutting width
SL Cutting length
DS Special compensation
SUT Tool type (required for AWG in TURN PLUS):
- 0: Recessing
- 1: Parting
- 2: Recess turning

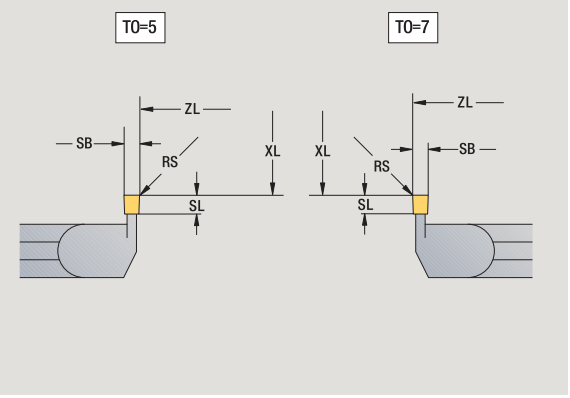
DN Tool width
SD Shank diameter
ET Maximum plunging depth
NL Usable length
RW Angular offset (only for B axis)

For further tool parameters, see Page 497.

The **wear compensation DX, DZ** compensates for wear on the two sides of the tool tip that lie next to the reference point. The **special compensation DS** compensates for wear on the third side of the tool tip.





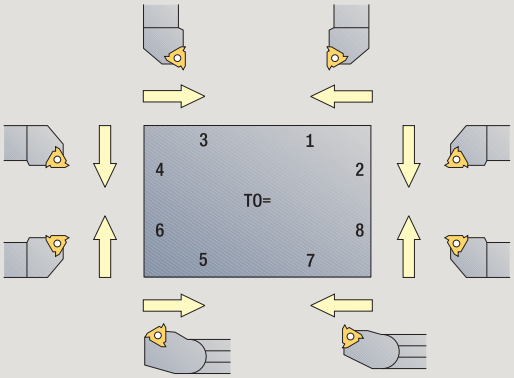


Thread-cutting tools

New tool

Select "New tool."

Select thread-cutting tools.

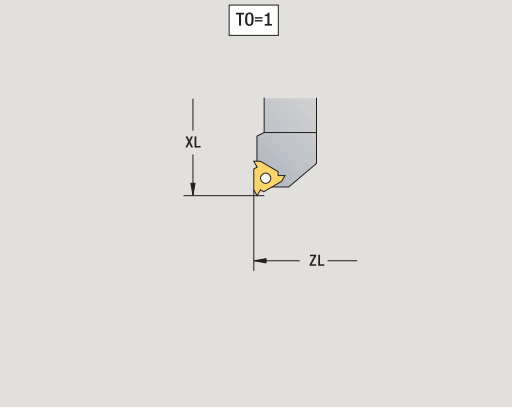


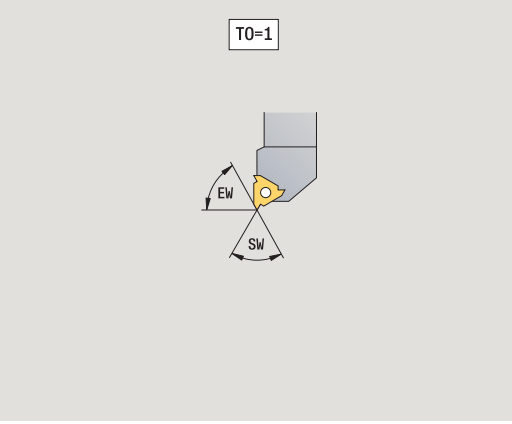
The help graphics illustrate the dimensions of the tools.

Special parameters for thread cutting tools

RS Cutting radius
SB Cutting width
EW Tool angle (range: 0° <= EW <= 180°)
SW Point angle (range: 0° <= SW <= 180°)
DN Tool width
SD Shank diameter
ET Maximum plunging depth
NL Usable length

For further tool parameters, see Page 497.





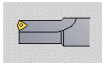
Twist drills and indexable-insert drills



Select "New tool."



Select drilling tools.



For indexable-insert drills, switch to dialog for indexable-insert drills.

The help graphics illustrate the dimensions of the tools.

Special parameters for twist drills

DV Drill diameter

BW Drill angle: Point angle of the drill

AW Driven tool : This parameter specifies for drilling and tapping tools during cycle programming whether switching commands are generated for the spindle or the driven tool.

- 0: Stationary tool
- 1: Driven tool

NL Usable length

RW Angle of orientation: Deviation from the main machining direction (input range: –90° to +90°)

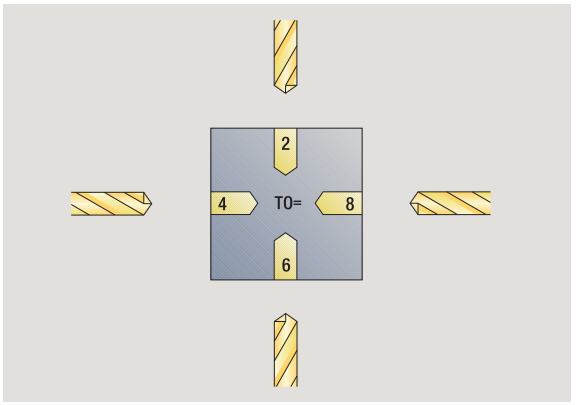
AX Salient length in X

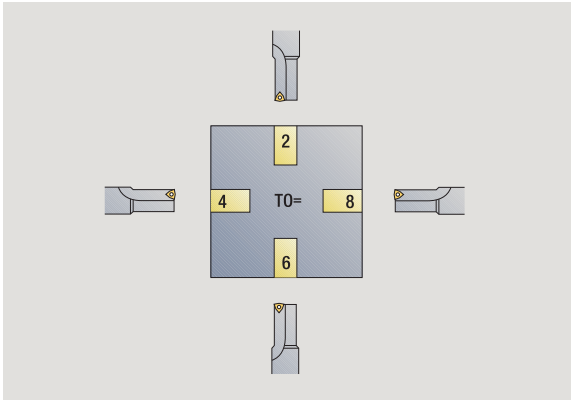
FH Chuck height

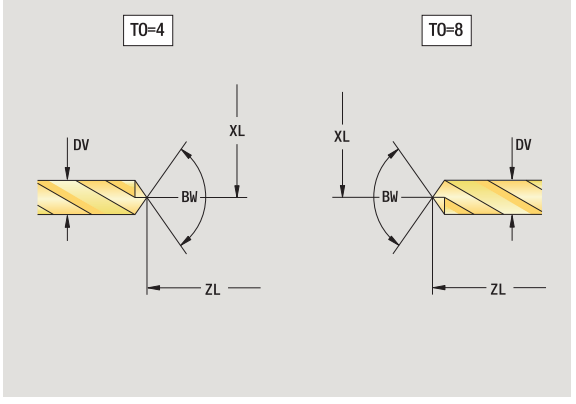
FD Diameter of the chuck

For further tool parameters, see Page 497.

For drilling operations with constant cutting speed, the **drilling diameter (DV)** is used to calculate the spindle speed.

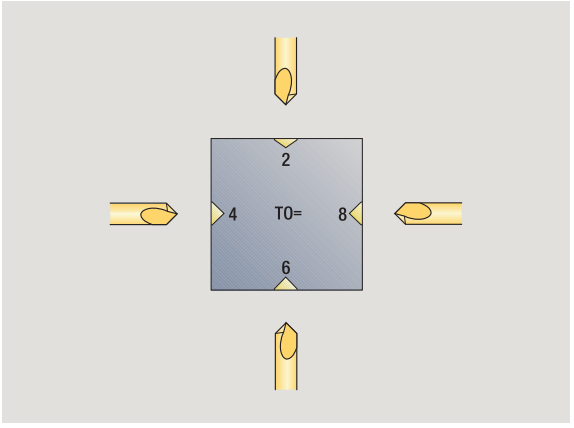






NC center drill

New tool	Select "New tool."
Special tool	Select special tools.
	Select special drilling tools.
	Select NC center drill.



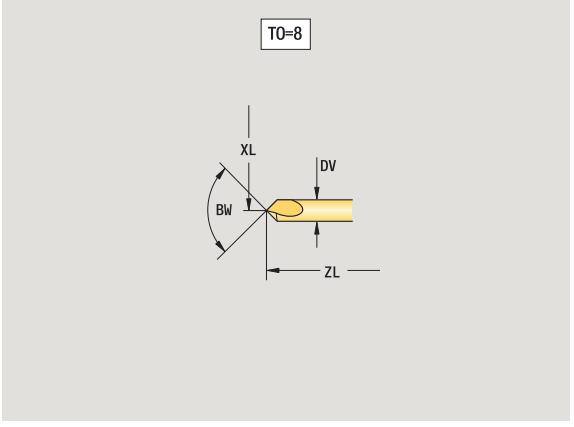
The help graphics illustrate the dimensions of the tools.

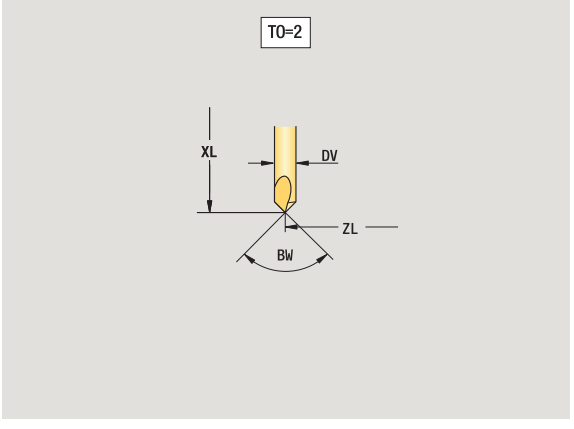
Special parameters for NC center drills

DV Hole diameter

BW Point angle

For further tool parameters, see Page 497.

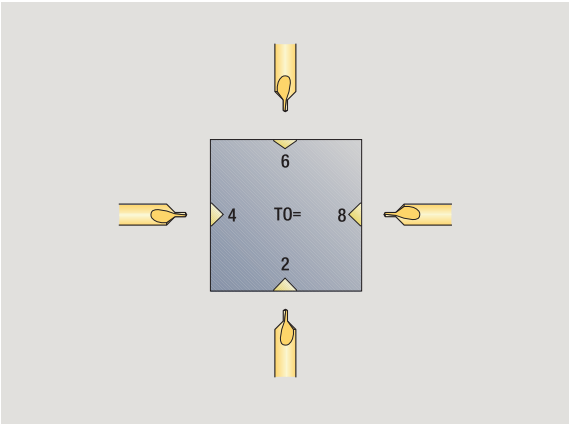




For drilling operations with constant cutting speed, the **`drilling diameter (DV)`** is used to calculate the spindle speed.

Centering tool

New tool	Select "New tool."
Special tool	Select special tools.
	Select special drilling tools.
	Select centering tools.

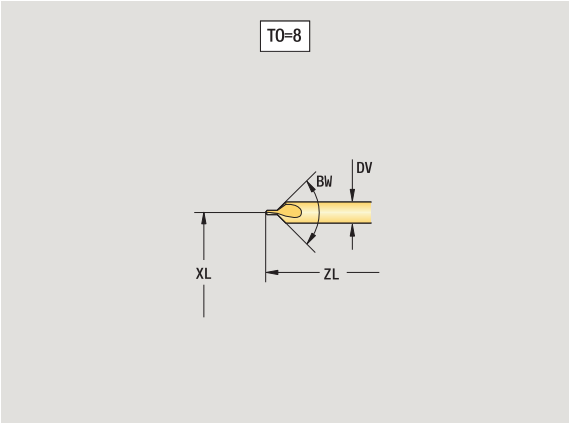


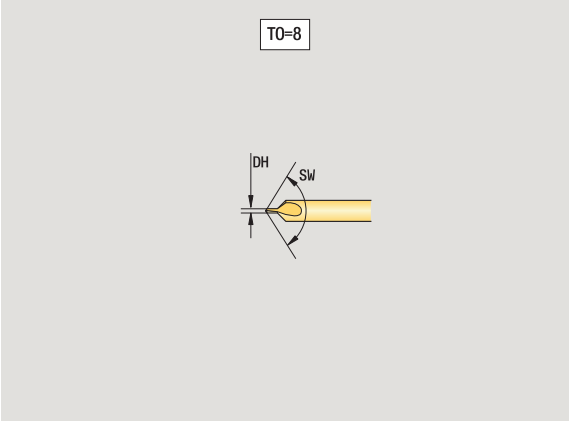
The help graphics illustrate the dimensions of the tools.

Special parameters for centering tools

DV Hole diameter
DH Stud diameter
BW Drill angle
SW Point angle
ZA Stud length

For further tool parameters, see Page 497.





For drilling operations with constant cutting speed, the **drilling diameter (DV)** is used to calculate the spindle speed.

Counterbore

New tool

Select "New tool."

Special tool

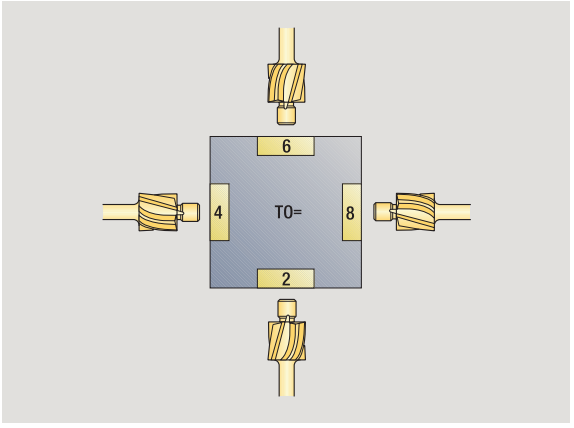
Select special tools.



Select special drilling tools.



Select counterbore.



The help graphics illustrate the dimensions of the tools.

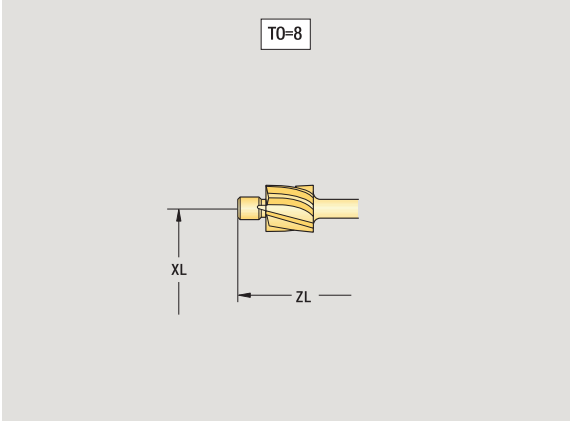
Special parameters for counterbores

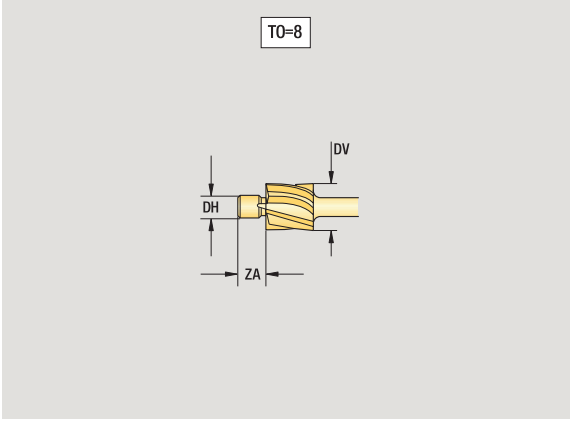
DV Hole diameter

DH Stud diameter

ZA Stud length

For further tool parameters, see Page 497.





For drilling operations with constant cutting speed, the **drilling diameter (DV)** is used to calculate the spindle speed.

Countersink



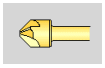
Select "New tool."



Select special tools.



Select special drilling tools.



Select counterbore.

The help graphics illustrate the dimensions of the tools.

Special parameters for countersinks

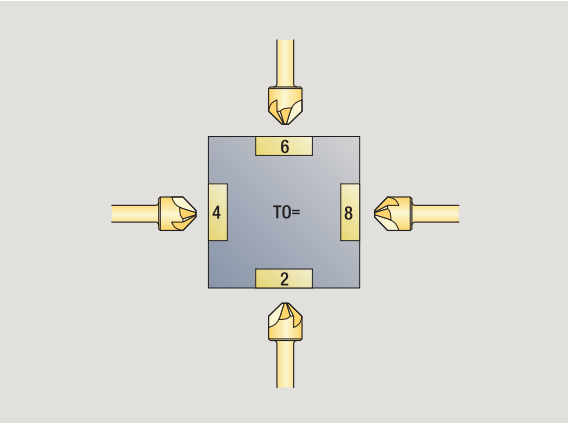
DV Hole diameter

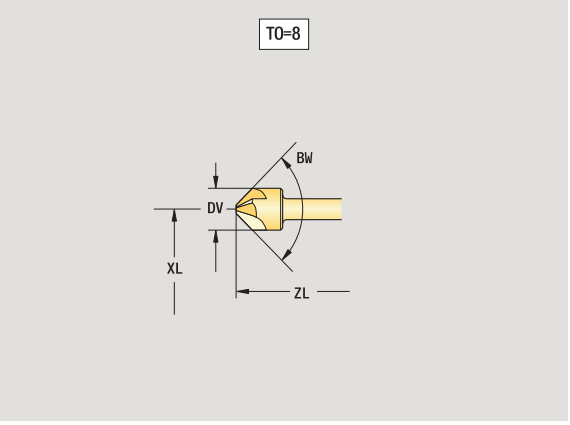
DH Stud diameter

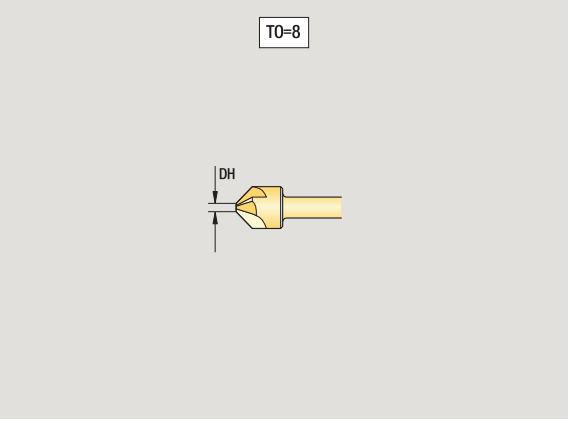
BW Drill angle

For further tool parameters, see Page 497.

For drilling operations with constant cutting speed, the **`drilling diameter (DV)`** is used to calculate the spindle speed.





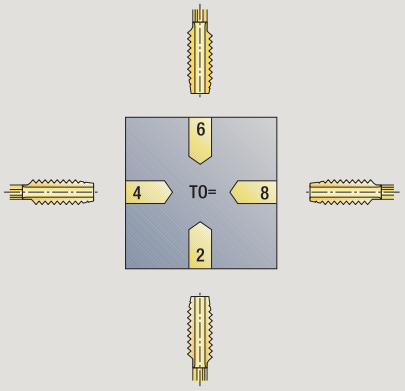


Tap

New tool

Select "New tool."

Select taps.



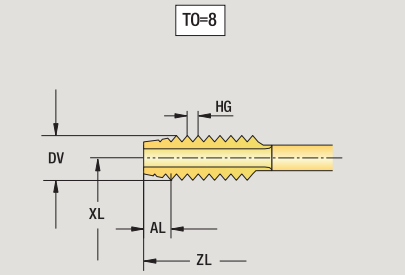
The help graphics illustrate the dimensions of the tools.

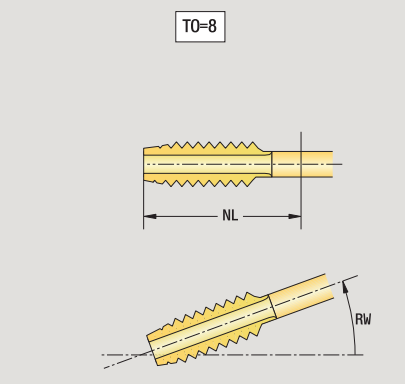
Special parameters for taps

DV Thread diameter
HG Thread pitch
AL Length of first cut

For further tool parameters, see Page 497.

The **thread pitch (HG)** is evaluated if the corresponding parameter is not defined in the tapping cycle.





Standard milling tools

New tool

Select "New tool."

Select milling tools.

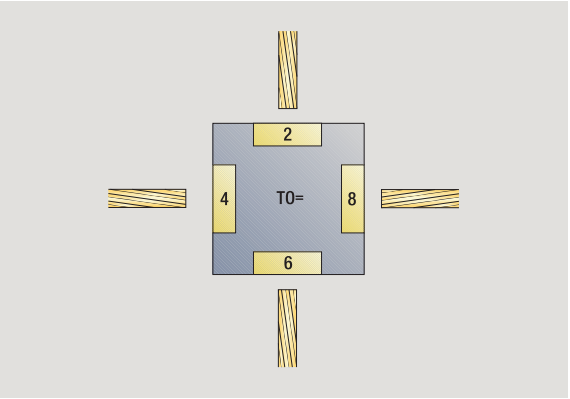
The help graphics illustrate the dimensions of the tools.

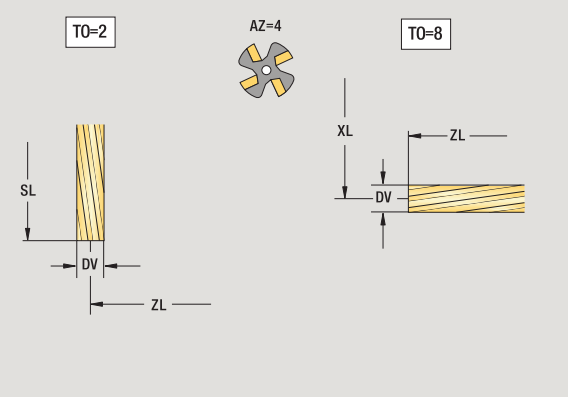
Special parameters for standard milling tools

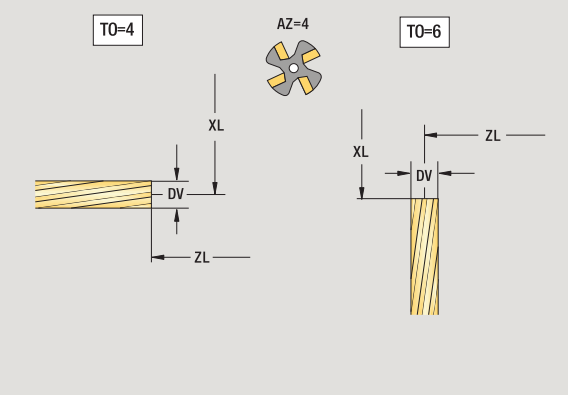
DV Cutter diameter
AZ Number of teeth
DD Cutter diameter compensation
SL Cutting length

For further tool parameters, see Page 497.

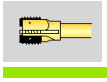
- For milling operations with constant cutting speed, the **milling cutter diameter (DV)** is used to calculate the spindle speed.
- The **number of teeth (AZ)** parameter is evaluated for **G193 Feed per tooth**.

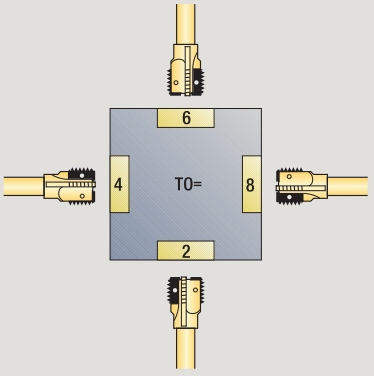






Thread milling tools

New tool	Select "New tool."
Special tool	Select special tools.
	Select special milling tools.
	Select the thread milling tool.

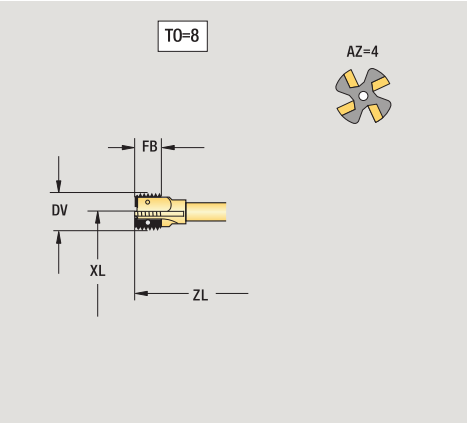


The help graphics illustrate the dimensions of the tools.

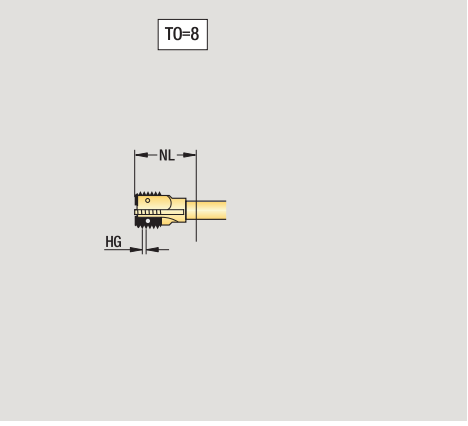
Special parameters for thread milling tools

DV Cutter diameter
AZ Number of teeth
FB Cutter width
HG Pitch
DD Cutter diameter compensation

For further tool parameters, see Page 497.

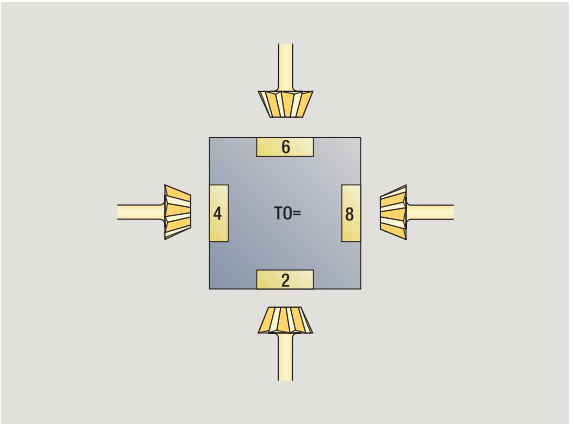


- For milling operations with constant cutting speed, the **`milling cutter diameter (DV)`** is used to calculate the spindle speed.
- The **`number of teeth (AZ)`** parameter is evaluated for **`G193 Feed per tooth`**.



Angle cutters

New tool	Select "New tool."
Special tool	Select special tools.
	Select special milling tools.
	Select angle cutters.



The help graphics illustrate the dimensions of the tools.

Special parameters for angle cutters

DV (Large) milling diameter

AZ Number of teeth

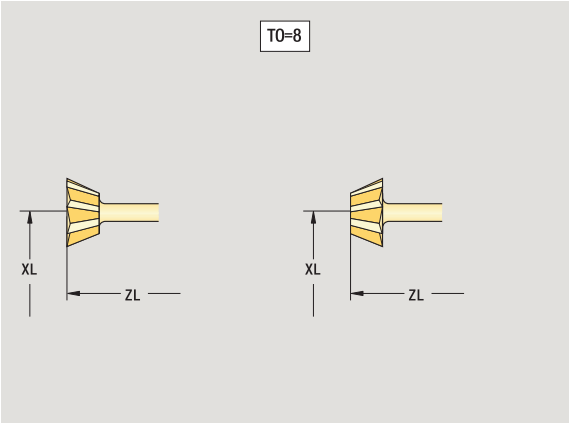
FB Cutter width

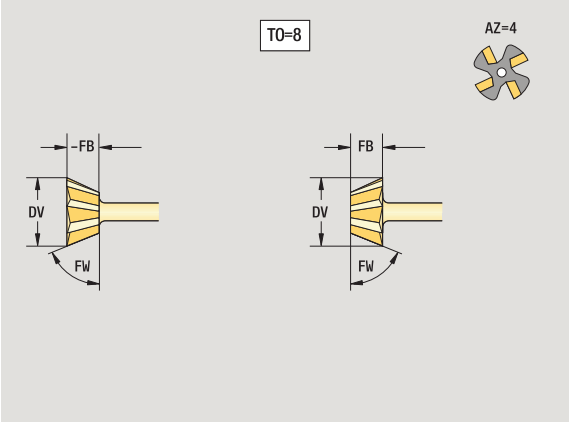
- FB<0: Large cutter diameter on front
- FB>0: Large cutter diameter on back

FW Cutter angle

DD Cutter diameter compensation

For further tool parameters, see Page 497.





- For milling operations with constant cutting speed, the **`milling cutter diameter (DV)`** is used to calculate the spindle speed.
- The **`number of teeth (AZ)`** parameter is evaluated for **`G193 Feed per tooth`**.

Milling pins



Select "New tool."



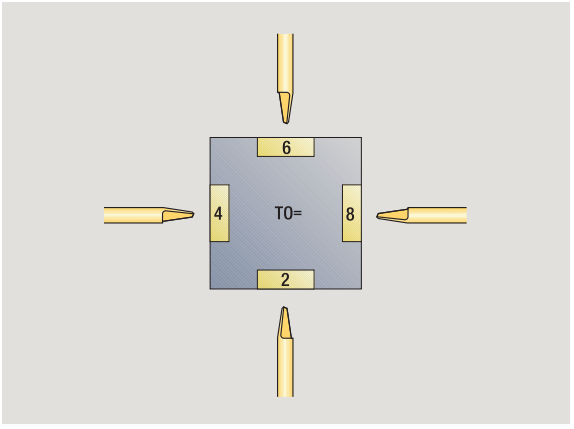
Select special tools.



Select special milling tools.



Select milling pins.

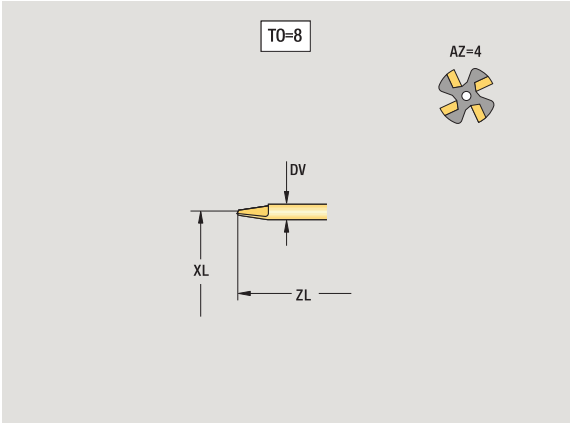


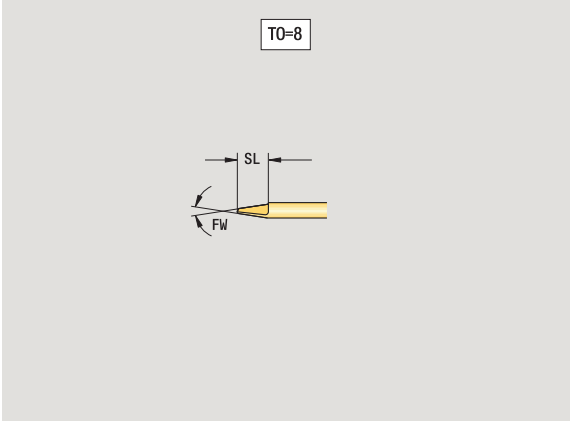
The help graphics illustrate the dimensions of the tools.

Special parameters for milling pins

DV Cutter diameter
AZ Number of teeth
SL Cutting length
FW Cutter angle
DD Cutter diameter compensation

For further tool parameters, see Page 497.





- For milling operations with constant cutting speed, the **`milling cutter diameter (DV)`** is used to calculate the spindle speed.
- The **`number of teeth (AZ)`** parameter is evaluated for **`G193 Feed per tooth`**.

Touch probes



Select "New tool."



Select special tools.



Select handling systems and touch probes.



Select touch probes.

The help graphics illustrate the dimensions of the tools.

Special parameters for touch probes

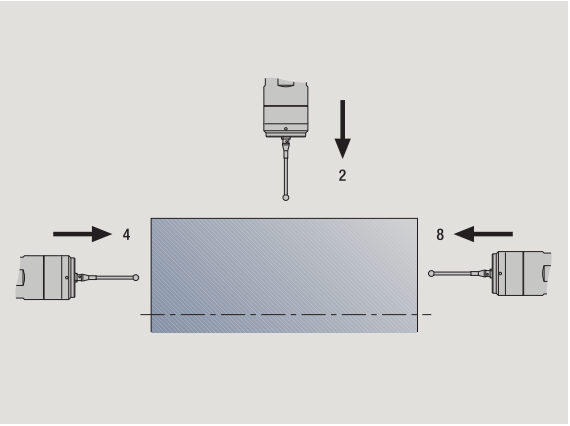
SL Cutting length

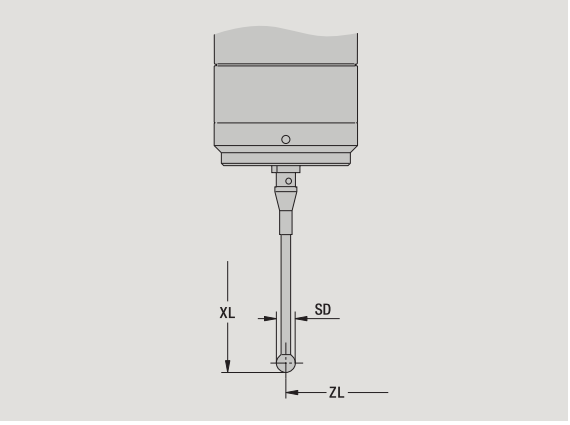
TP Selection of touch probes

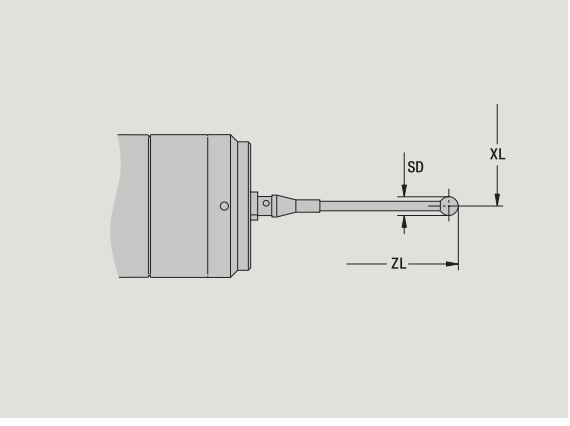
For further tool parameters, see Page 497.



The CNC PILOT must be specially prepared by the machine tool builder for the use of a 3-D touch probe.







7.4 Technology database

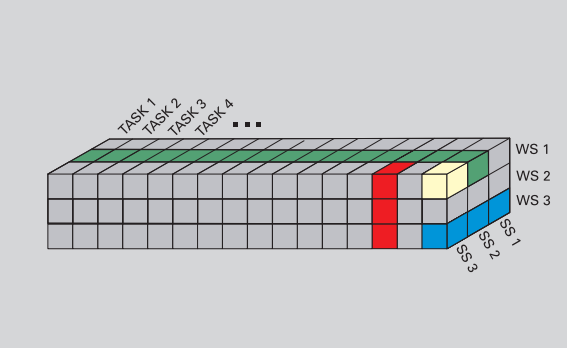
The technology database manages the cutting data according to the machining mode, the workpiece material and the cutting material. The graphic on this page shows the composition of the database. Each cube represents a data record with cutting data.

In its standard version, the technology database is designed for 9 workpiece-material/tool-material combinations. You can optionally expand the database to 62 workpiece material-cutting material combinations.

The CNC PILOT ascertains these criteria from the following information:

- **Operating mode:** An operating mode is assigned in the cycle programming (Teach-in mode) to each cycle, and in smart.Turn to each unit (see table).
- **Work material:** The workpiece material is defined in the ITS menu during cycle programming, and in the program header in smart.Turn.
- **Cutting material:** Each tool description contains the cutting material.

Using these three criteria, the CNC PILOT accesses the cutting data record (shown yellow in the figure) and generates from it the technology-data proposal.



Explanation of abbreviations used in the graphic:

- Task: Machining mode
- WS: Workpiece material
- SS: Cutting material

Types of machining	
Predrilling	Not used
Roughing	2
Finishing	3
Thread cutting	4
Contour recessing	5
Parting	6
Centering	9
Drilling	8
Countersinking	9
Reaming	Not used
Tapping	11
Milling	12
Finish milling	13
Deburring	14
Engraving	15
Recess turning	16

Technology editor

The technology editor can be called from the Tool Editor and smart.Turn operating modes.

Database access of the following combinations are supported:

- Work material/operating mode combinations (blue)
- Cutting material/operating-mode combinations (red)
- Work-material/tool-material combinations (green)

Editing workpiece and cutting material designations: The technology editor keeps one list each with workpiece designations and cutting material designations. You have the following options:

- Insert **new workpiece/cutting materials**.
- **Do not change** the workpiece material or cutting material designations.
- **Delete** existing workpiece/cutting material designations. This also deletes the associated cutting data.



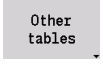
Remember when deleting workpiece material or cutting material designations:

- This also deletes the associated cutting data.
- The CNC PILOT cannot determine any cutting data for the affected programs or tools. The reason:
  - The workpiece material designations are saved in the program header of the smart.Turn programs.
  - Cutting material designations are saved together with the tool data.

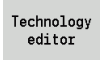
Editing the cutting data: The cutting data of a workpiece material/ cutting material combination are called a "data record". You have the following options:

- Assign cutting data to a workpiece material/cutting material combination to create a data record.
- Delete the cutting data of a workpiece material/cutting material combination (a data record).

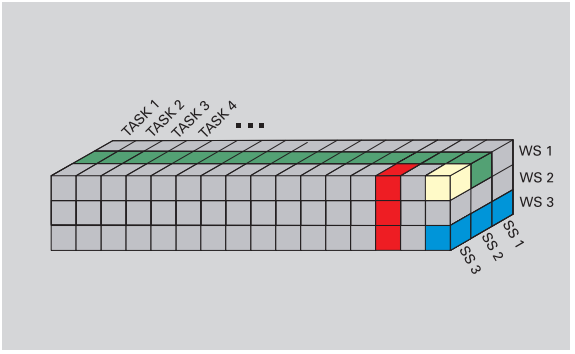
Call the technology editor in the tool editor operating modes:



▶ Press the "Other tables" soft key



▶ Call up the technology editor: Press the "technology editor" soft key



Explanation of abbreviations used in the graphic:

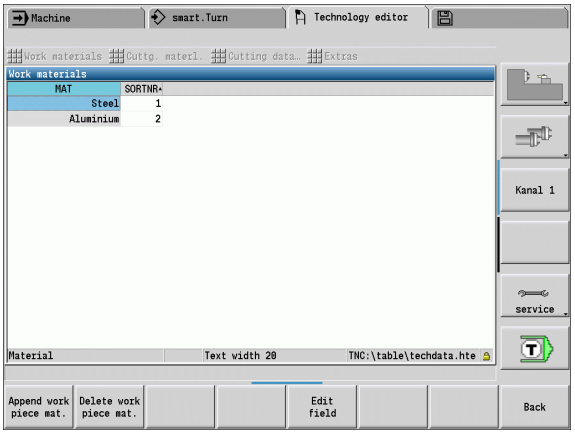
- Task: Machining mode
- WS: Workpiece material
- SS: Cutting material

Editing a workpiece material or cutting material list

Work material list



Select the "Work materials" menu item. The editor opens the list with the workpiece material designations.



Adding a workpiece material:

Append work piece mat.

Press the soft key. Enter the workpiece designation (maximum 16 characters). The sorting number is assigned sequentially.

Deleting a workpiece material:

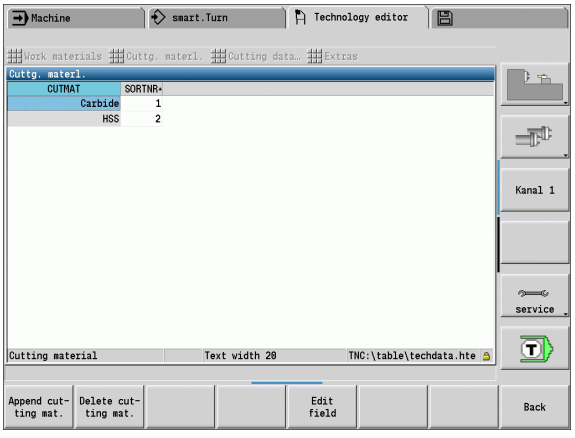
Delete work piece mat.

Press the soft key. After the confirmation request, the CNC PILOT deletes the workpiece material with **all associated cutting data**.

Cutting material list



Select the "Cutting materials" menu item. The editor opens the list with the cutting material designations.



Adding a cutting material:

Append cut-ting mat.

Press the soft key. Enter the cutting material designation (maximum 16 characters). The sorting number is assigned sequentially.

Deleting a cutting material:

Delete cut-ting mat.

Press the soft key. After the confirmation request, the CNC PILOT deletes the cutting material with **all associated cutting data**.

The **sorting number** only specifies the sequence within the list. To change the sorting number: Select the sorting number, press the **Edit field** soft key and enter the new number.



Expanding the workpiece-material or cutting-material lists does not create cutting data. The data record for cutting data of a new workpiece material/cutting material combination is not created until you have requested it through the **New data record** soft key.

Displaying/editing cutting data

Displaying cutting data of the machining modes:



- Select the "Cutting data" menu item. The editor opens the dialog for selecting a workpiece material/cutting material combination.
- Select the desired combination and press **OK**.
- The technology editor displays the cutting data.

Displaying cutting data of the workpiece materials:



- "Extras/.." menu item



- "... Select "Work material table..." The editor opens the dialog for selecting an operating mode/cutting material combination.
- Select the desired combination and press **OK**.
- The technology editor displays the cutting data.

Displaying cutting data of the cutting materials:



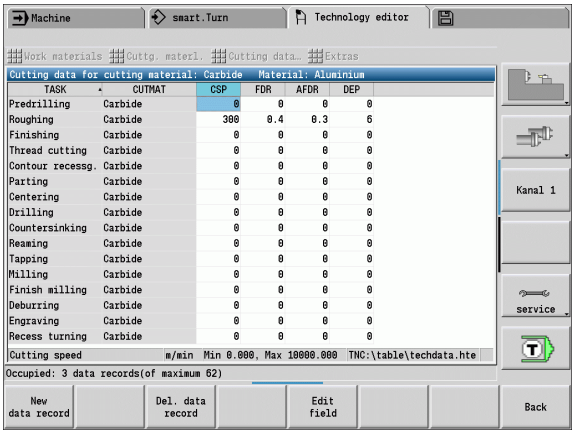
- "Extras/.." menu item

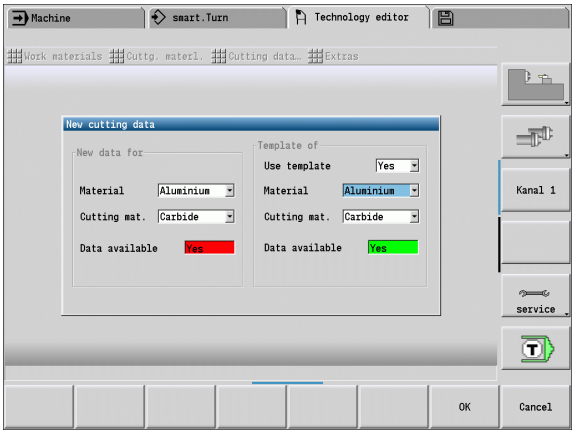


- "... Select "Cutting material table." The editor opens the dialog for selecting a workpiece material/ operating mode combination.
- Select the desired combination and press **OK**.
- The technology editor displays the cutting data.



The **value 0** in a data record means that no value is transferred to the unit or cycle dialog.





Editing cutting data:

- Call the table with cutting data.
- With the **arrow keys**, select the cutting data field you want to edit.
- Press the soft key

Edit field

- Enter the value and confirm with the **Enter** key.

Adding new cutting data:

- Set any workpiece-material/cutting material combinations.
- Press the soft key. The editor opens the "New cutting data" dialog box.

New data record

- Set the new material/cutting material combinations.
- Decide whether an existing workpiece material/cutting material combination should be used as a template. Otherwise, all entries with be preset with "0."
- Press **OK** to create the new cutting data records.

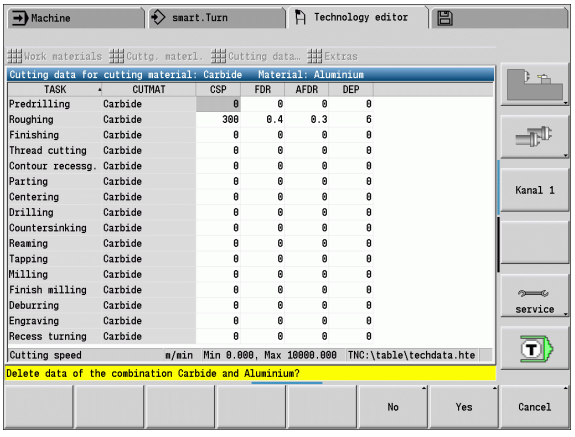
Deleting a data record with cutting data:

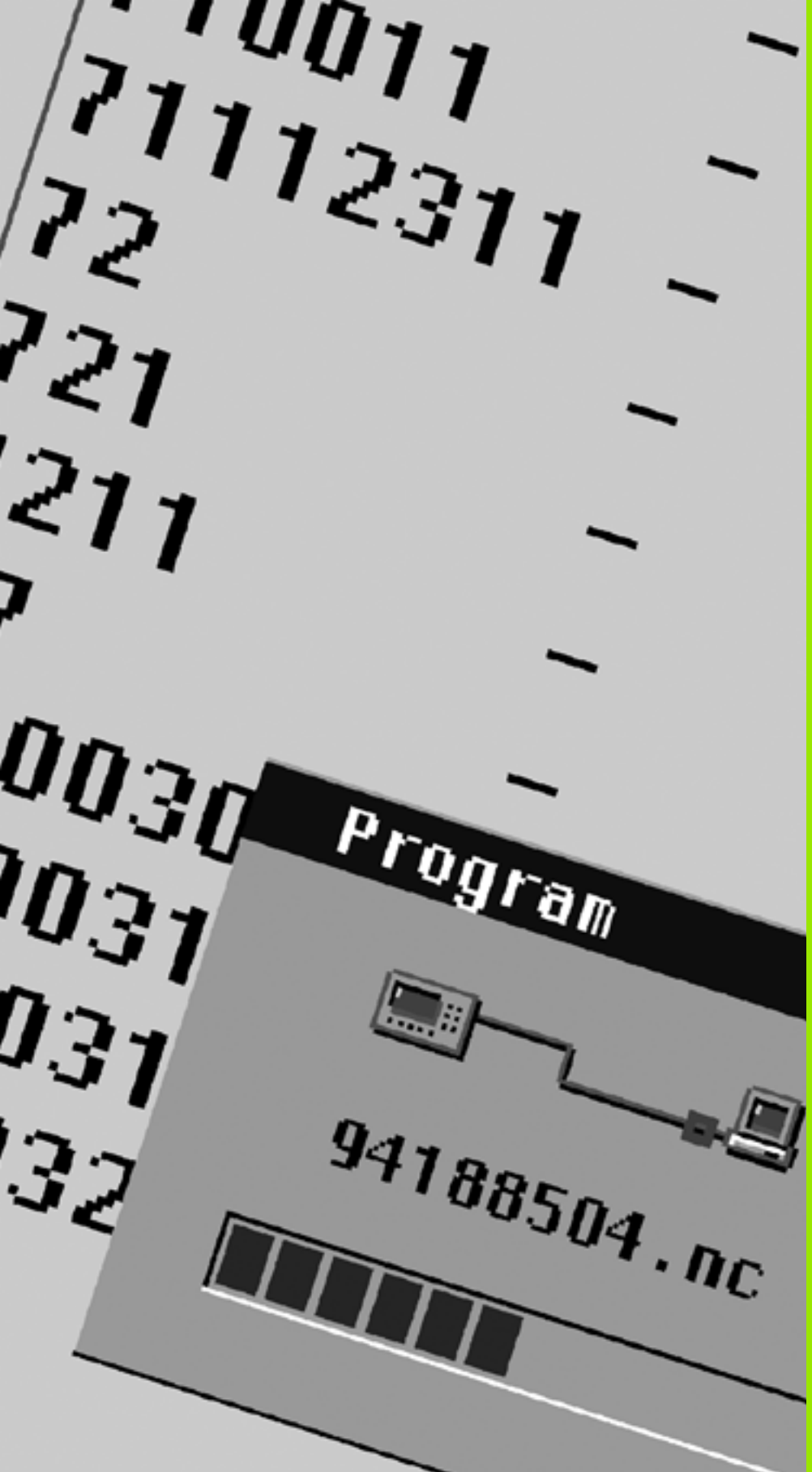
- Set the workpiece material/cutting material combination (data record) to be deleted.
- Press the soft key. The technology editor asks you if you really want to delete the data record.

Del. data record

- Press the soft key. The technology editor deletes the data record of the given workpiece material/cutting material combination.

YES





8

Organization mode of operation

8.1 Organization mode of operation

This mode of operation offers various functions for communication with other systems, data backup, setting of parameters, and diagnosis.

The following functions are available:

- **Login code**
  Some parameter settings and functions may only be accessed by qualified personnel. Users need to enter a code number to log in to this mode.
- **Parameter settings**
  Parameters enable you to adapt CNC PILOT to your specific requirements. The **User parameter** menu provides functions to display and edit parameters.
- **Transfer**
  The transfer functions are used either for exchange of data with other systems or for data backup. You can input and output programs, parameters, and tool data.
- **Diagnosis**
  The Diagnosis menu provides functions for checking the system and for locating errors.



The Configuration and Diagnosis functions can only be accessed by authorized commissioning and service personnel.

Login code	
Code number	Possibilities
	Editing user parameters Transfer: ■ Transmit/Receive programs ■ Create service files
123	Editing all user parameters Transfer ■ Parameter backup ■ Backup/Restore tools
net123	Configuring the network (control name / DHCP) Transfer ■ Parameter backup ■ Backup/Restore tools
sik	Options dialog box Opens the dialog box for activating software options via SIK (system identification key)
Service code	Editing configuration data Diagnostic functions Restoring the parameters

8.2 Parameters

Parameter editor

The parameter values are entered in the **configuration editor**.

Each parameter object has a name (e.g. **CfgDisplayLanguage**) that gives information about the parameters it contains. Each object has a **key** for unique identification.

The CNC PILOT displays an icon at the beginning of each line in the parameter tree showing additional information about this line. The icons have the following meanings:

	Branch exists but is closed
	Branch is open
	Empty object, cannot be opened
	Initialized machine parameter
	Uninitialized (optional) machine parameter
	Can be read but not edited
	Can neither be read nor edited

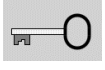
User parameter (user parameters)

Parameters that are preset for the usual "daily operations" are organized as **User parameters**.

To enable you to set machine-specific functions, your machine tool builder make further parameters available as user parameters.

Refer to your machine manual.

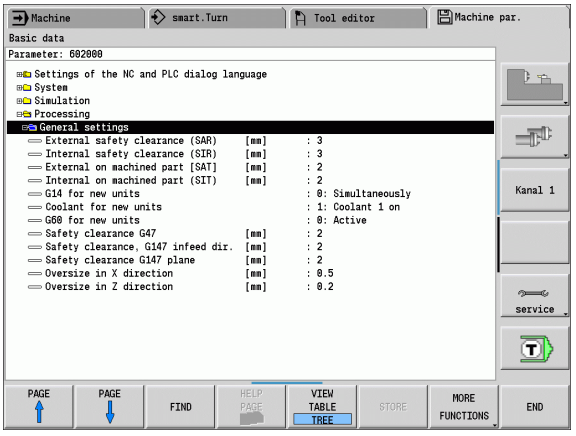
Editing user parameters



Press soft key and enter the key number **123**.

USER PARAMETER

Press the **User parameters** soft key.



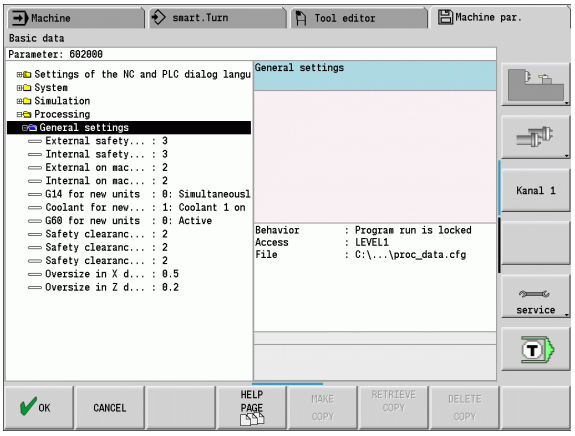
Displaying help texts

Position the cursor on the parameter.

Press the info key.

The parameter editor opens the window with information on these parameters.

Press the info key again to close the information window.



Searching for parameters

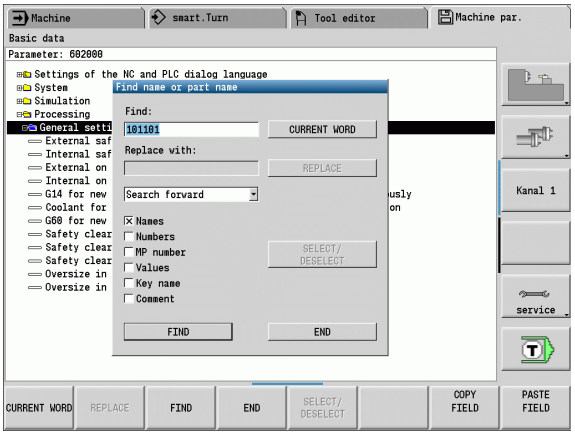
FIND

Press the **Find** soft key

Enter the search criteria.

FIND

Press the **Find** soft key again.



Exiting the parameter editor

End

Press the **End** soft key.

List of user parameters

Language setting:

Parameter: NC and PLC conversational language settings / ...

... / NC conversational language (101301)

- ENGLISH
- GERMAN
- CZECH
- FRENCH
- ITALIAN
- SPANISH
- PORTUGUESE
- SWEDISH
- DANISH
- FINNISH
- DUTCH
- POLISH
- HUNGARIAN
- RUSSIAN
- CHINESE
- CHINESE_TRAD
- SLOVENIAN
- ESTONIAN
- KOREAN
- LATVIAN
- NORWEGIAN
- ROMANIAN
- SLOVAK
- TURKISH
- LITHUANIAN

... / PLC conversational language (101302)

- See NC conversational language

... / PLC error message language (101303)

- See NC conversational language

... / Language for online help (101304)

- See NC conversational language

General settings:

Parameter: System / ...	Meaning
... / Definition of the units of measure valid for the display (101100) / ...	
... / Unit of measure for display and user interface (101101)	
metric	Use the metric system
inch	Use the inch system
... / General settings for automatic operation (601800) / ...	
.../ Service life management (601801)	
ON	Tool life monitoring is active
OFF	Tool life monitoring is inactive
.../ End mid-program startup after start block (601810)	
TRUE	In mid-program startup, program execution begins with the subsequent NC block
FALSE	In mid-program startup, program execution begins with the selected NC block
... / Tool measurement (604600)	
Measuring feed rate [mm/min] (604602)	Feed rate for approaching the touch probe
Measuring range [mm] (604603)	The touch probe must be triggered within the measuring range. Otherwise, an error message is issued.

Settings for the simulation:

Parameter: Simulation / ...	Meaning
... / General settings (114800) / ...	
... / Restart with M99 (114801)	
ON	Simulation begins again at beginning of program
OFF	Simulation stops
... / Traverse delay [s] (114802)	Delay time after each path has been graphically simulated. The simulation speed can thus be influenced.
... / Machining times for the NC functions in general (115000) / ...	These times are used as nonmachining times for the time calculation function.
... / Time allowance for tool change [s] (115001)	
... / Time allowance for gear shifting [s] (115002)	
... / General time allowance for M functions [s] (115003)	
... / Machining times for M functions (115100) / ...	Individual time allowances for a maximum of 14 M functions
... / T01 / ...	
... / Number of the M function	
... / Operating time of the M function [s]	The time calculation adds this time to the "General time allowance for M functions."
... / T14	
... / Specification of the (standard) window size (115200)	The simulation function adapts the window size to the workpiece blank. If no blank part is programmed, the simulation uses a standard window size.
... / Zero point position in X [mm] (115201)	Distance of the coordinate origin from the lower window edge.
... / Zero point position in Z [mm] (115202)	Distance of the coordinate origin from the left edge of the window.
... / Delta X [mm] (115203)	Vertical expansion of the graphic window
... / Delta Z [mm] (115204)	Horizontal expansion of the graphic window
... / Specification of the (standard) workpiece blank size (115300)	If no blank part is programmed in DIN PLUS, the simulation uses a "standard blank."
... / Outside diameter [mm] (115301)	
... / Workpiece blank length [mm] (115302)	
... / Right edge of workpiece blank [mm] (115303)	
... / Inside diameter [mm] (115304)	

Settings for fixed cycles and units:

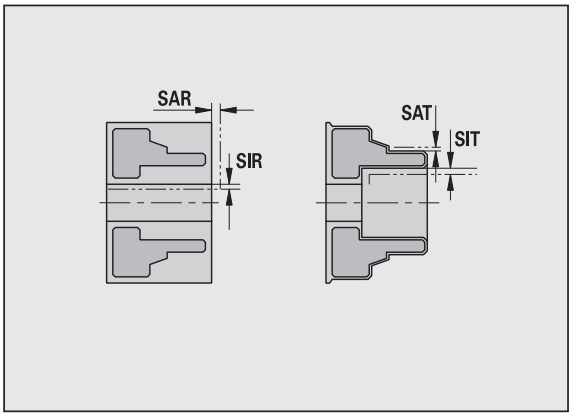
Parameters: processing / ...	Meaning
... / General settings (602000) / ...	
... / External safety clearance (SAR) [mm] (602005)	External safety clearance on workpiece blank
... / Internal safety clearance (SIR) [mm] (602006)	Internal safety clearance on workpiece blank
... / External on machined part (SAT) [mm] (602007)	External safety clearance on machined part
... / Internal on machined part (SIT) [mm] (602008)	Internal safety clearance on machined part
... / G14 for new units (602009)	Default value for "tool change point G14."
... / Coolant for new units (602010)	Default value for "coolant CLT": ■ 0: Without (coolant) ■ 1: Circuit 1 on ■ 2: Circuit 2 on
... / G60 for new units (602011)	Default value for "protection zone": ■ 0: Active ■ 1: Inactive
... / Safety clearance G47 [mm] (602012)	Default value for "safety clearance G47":
... / Safety clearance, G147 infeed direction [mm] (602013)	Default value for "safety clearance SCK"
... / Safety clearance, G147 plane [mm] (602014)	Default value for "safety clearance SCI"
... / Oversize in X direction [mm] (602015)	Default value for "oversize (X) I"
... / Oversize in Z direction [mm] (602016)	Default value for "oversize (Z) K"

Machining parameters (processing)

Machining parameters are used by the work plan generation (TURN PLUS) and various machining cycles.

General settings

Global technology parameters – Safety clearances



Global safety clearance

Speed limiting [SMAX]

Global speed limiting. You can define a small speed limit in the program head of the TURN PLUS program.

- **External on blank part [SAR]**
- **Internal on blank part [SIR]**

TURN PLUS takes account of **SAR/SIR**

- for all rough-turning operations
- for centric predrilling

- **External on machined part [SAT]**
- **Internal on machined part [SIT]**

TURN PLUS takes **SAT/SIT** into account on premachined workpieces for:

- Finish-machining
- Recess turning
- Contour recessing (contour cutting)
- Recessing
- Thread cutting
- Measuring

G14 for new units

Default setting for the axis sequence (start unit: **GWW** parameter) that is used to approach the tool change position:

- No axis
- 0: Simultaneously
- 1: First X, then Z
- 2: First Z, then X
- 3: Only X
- 4: Only Z

Global safety clearance

Coolant for new units

Default setting for the coolant (start unit: **CLT** parameter):

- 0: Without coolant
- 1: Coolant circuit 1 on
- 2: Coolant circuit 2 on

Protection zone "G60" for new units

Default setting for the protection zone (start unit: **G60** parameter):

- 0: Active
- 1: Inactive

Global safety clearance G47

Default setting for the global safety clearance (start unit: **G47** parameter)

Global safety clearance G147 in the plane

Default setting for the global safety clearance in the plane (start unit: **G147** parameter)

Global safety clearance G147 in infeed direction

Default setting for the global safety clearance in infeed direction (start unit: **G147** parameter)

Global oversize in X direction

Default setting for the global safety clearance in X direction (start unit: **I** parameter)

Global oversize in Z direction

Default setting for the global safety clearance in X direction (start unit: **K** parameter)

Rotational direction for new units

Default for the spindle direction of rotation **MD** when creating/ opening a new unit ("Tool" tab)

Front edge of chuck on main spindle

Z position of the front edge of the chuck for calculating the workpiece zero point (AWG)

Front chuck edge on counterspindle

Z position of the front edge of the chuck for calculating the workpiece zero point (AWG)

Jaw width on main spindle

Jaw width in Z direction for calculating the workpiece zero point (AWG)

Jaw width on counterspindle

Jaw width in Z direction for calculating the workpiece zero point (AWG)

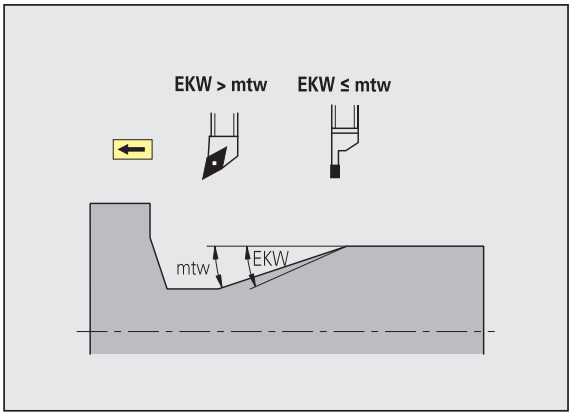
Global parameters for finished parts

Global parameters for finished parts

Max. inward copying angle [EKW]

Tolerance angle for recess areas, used for distinguishing turning from recessing (mtw = contour angle).

- EKW > mtw: Relief turn
- EKW <= mtw: Undefined recess (no form element)



Centric predrilling

Centric predrilling – Tool selection

Tool selection

1st drilling diameter limit [UBD1]

- 1st predrilling step: if UBD1 < DB1max
- Tool selection: UBD1 <= db1 <= DB1max

2nd drilling diameter limit [UBD2]

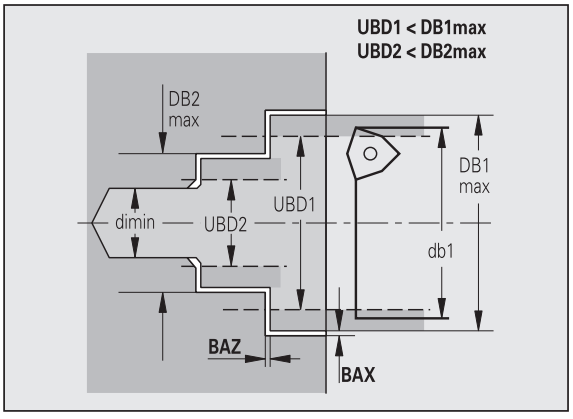
- 2nd predrilling step: if UBD2 < DB2max
- Tool selection: UBD2 <= db2 <= DB2max

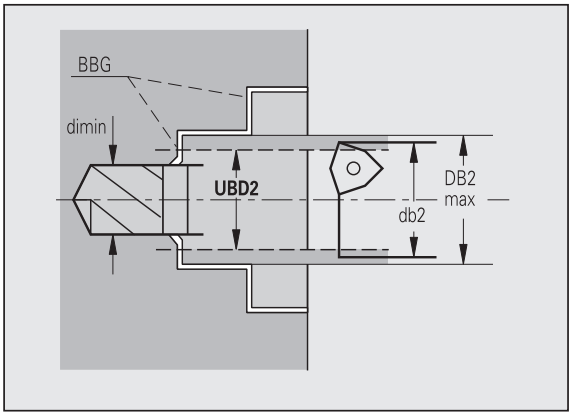
For **predrilling**, a maximum of three drilling steps is used:

- 1st predrilling step (limit diameter UBD1)
- 2nd predrilling step (limit diameter UBD2)
- Finish-drilling step
  - Finish-drilling is performed with: dimin <= UBD2
  - Tool selection: db = dimin

Designations in the figures at right:

- db1, db2: Drill diameter
- DB1max: Maximum inside diameter for 1st drilling step
- DB2max: Maximum inside diameter for 2nd drilling step
- dimin: Minimum inside diameter





- BBG (drilling limitation elements): Contour elements intersected by UBD1/UBD2

- UBD1/UBD2 have no effect when "Centric predrilling" has been defined as main machining operation followed by "Finish-drilling" as submachining operation in the machining sequence (see smart.Turn and DIN Programming User's Manual).
- Prerequisite: UBD1 > UBD2
- UBD2 must permit subsequent inside machining with boring bars.

Centric predrilling – Oversizes

Oversizes

Point angle tolerance [SWT]

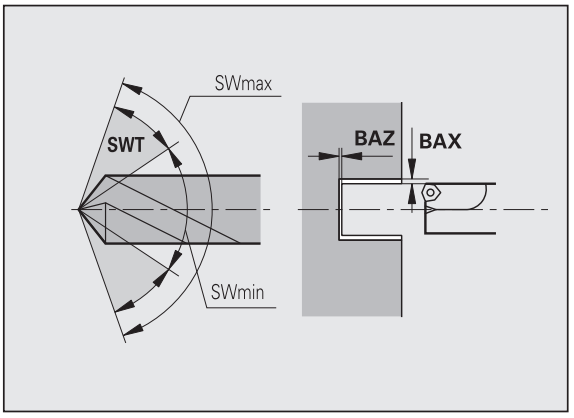
If the drilling limitation element is a diagonal element, TURN PLUS prefers using a twist drill with suitable point angle. If no suitable twist drill is available, an indexable-insert drill is selected for the predrilling operation. SWT defines the permissible point angle tolerance.

Drilling oversize – Diameter [BAX]

Machining oversize on drilling diameter (X direction – radius value).

Drilling oversize – Depth [BAZ]

Machining oversize on drilling depth (Z direction – radius value).



BAZ is not considered if:

- A subsequent inside finishing operation is not possible due to the small diameter.
- "dimin < 2* UBD2" in the finish-drilling step for blind holes.

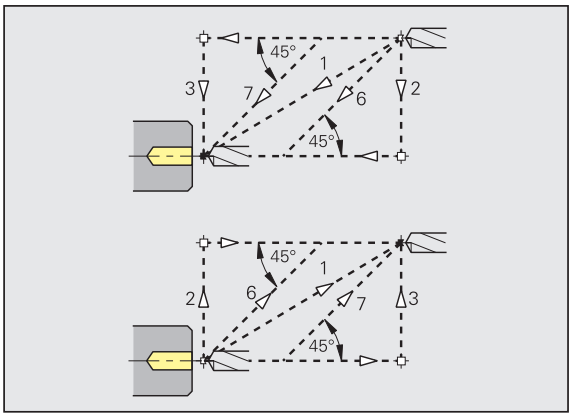
Centric predrilling – Approach and departure

Approach and departure

- **Traverse for predrilling [ANB]**
- **Traverse for tool change [ABW]**

  Approach/departure strategy:

  - 1: Move simultaneously in X and Z directions
  - 2: First X, then Z direction
  - 3: First Z, then X direction
  - 6: Coupled motion; X precedes Z direction
  - 7: Coupled motion; Z precedes X direction



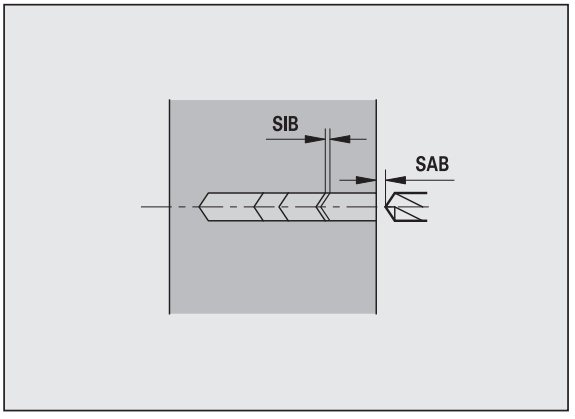
Centric predrilling – Safety clearances

Safety clearances

Safety clearance on blank part[SAB]

Internal safety clearance [SIB]

Retraction distance for deep-hole drilling ("B" for G74)



Machining

Drilling depth ratio [BTV]

TURN PLUS checks the 1st and 2nd drilling steps. The predrilling step is performed with:

BTV <= BT / dmax

Drilling depth factor [BTF]

1st drilling depth for deep-hole drilling cycle (G74):

bt1 = BTF * db

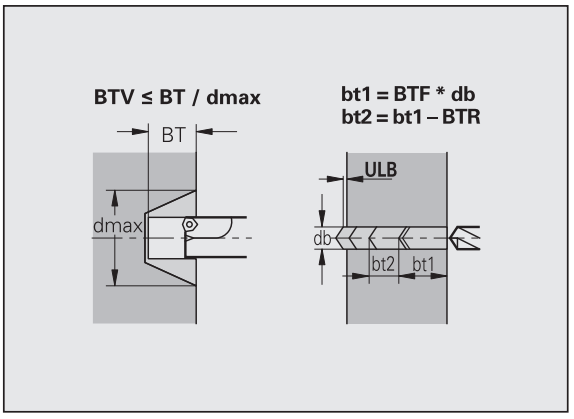
Depth reduction [BTR]

Reduction for deep-hole drilling cycle (G74):

bt2 = bt1 – BTR

Overhang length – Predrilling [ULB]

Through-drilling length



Roughing

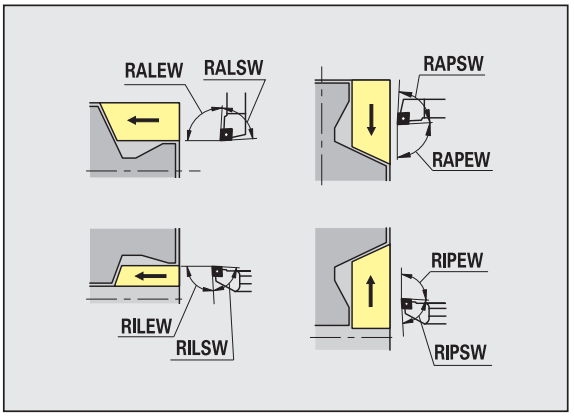
Roughing – Tool standards

Furthermore:

- Roughing cycles are primarily executed with standard roughing tools.
- Alternatively, tools that allow full-surface machining are used.

Tool standards

- **Tool angle – external/longitudinal [RALEW]**
- **Point angle – external/longitudinal [RALSW]**
- **Tool angle – external/transverse [RAPEW]**
- **Point angle – external/transverse [RAPSW]**
- **Tool angle – internal/longitudinal [RILEW]**
- **Point angle – internal/longitudinal [RILSW]**
- **Tool angle – internal/transverse [RIPEW]**
- **Point angle – internal/transverse [RIPSW]**

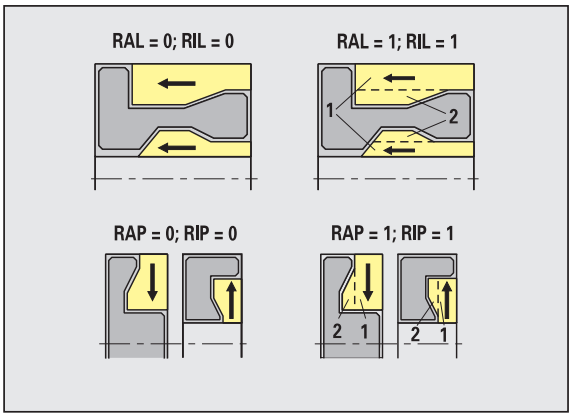


Machining standards

- **Standard/Full-surface – external/longitudinal [RAL]**
- **Standard/Full-surface – internal/longitudinal [RIL]**
- **Standard/Full-surface – external/transverse [RAP]**
- **Standard/Full-surface – internal/transverse [RIP]**

Input for RAL, RIL, RAP, RIP:

- 0: Full-surface roughing cycle, including plunge-cutting. TURN PLUS looks for a tool for full-surface machining.
- 1: Standard roughing cycle without plunge-cutting



Roughing – Tool tolerances

For tool selection, the following applies:

- Tool angle (EW): EW >= mkw (mkw: ascending contour angle)
- Tool angle (EW) and point angle (SW): NWmin < (EW+SW) < NWmax
- Adjacent angle (RNWT): RNWT = NWmax – NWmin

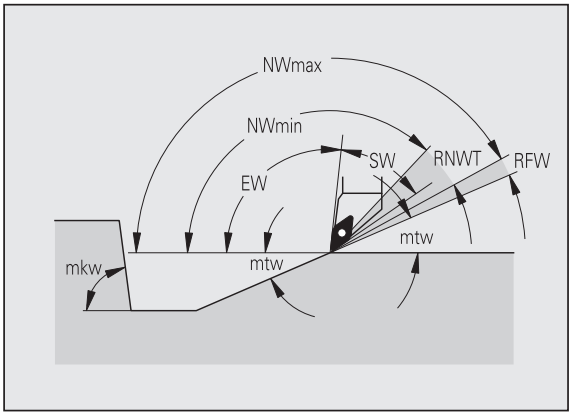
Tool tolerances

Adjacent angle tolerance [RNWT]

Tolerance range for secondary cutting edge

Relief cutting angle [RFW]

Minimum angle differential between the contour and secondary cutting edge



Roughing – Oversizes

Oversizes

Type of oversize [RAA]

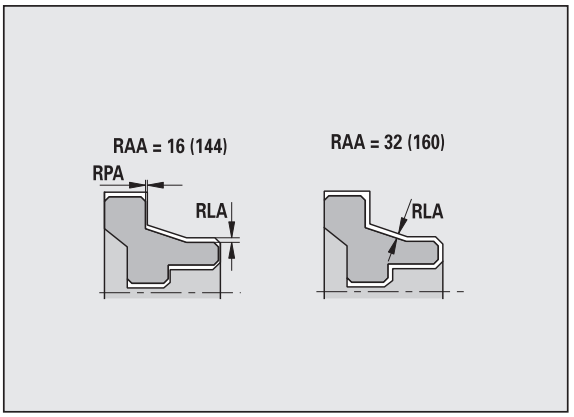
- 16: Longitudinal and transverse oversizes differ – no single oversizes
- 144: Longitudinal and transverse oversizes differ – with single oversizes
- 32: Equidistant oversize – no single oversizes
- 160: Equidistant oversize – with single oversizes

Equidistant or longitudinal [RLA]

Equidistant oversize or longitudinal oversize

None or transverse [RPA]

Transverse oversize



Roughing – Approach and departure

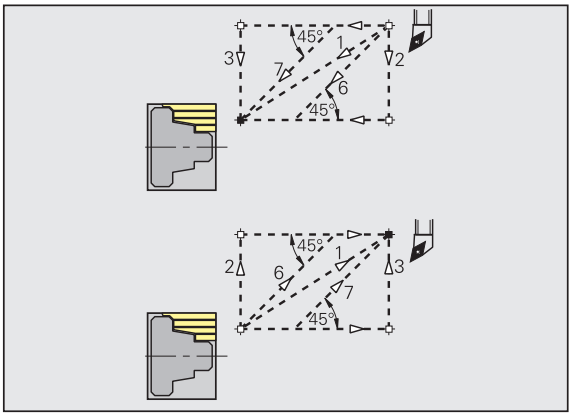
Approach and departure are at rapid traverse (G0).

Approach and departure

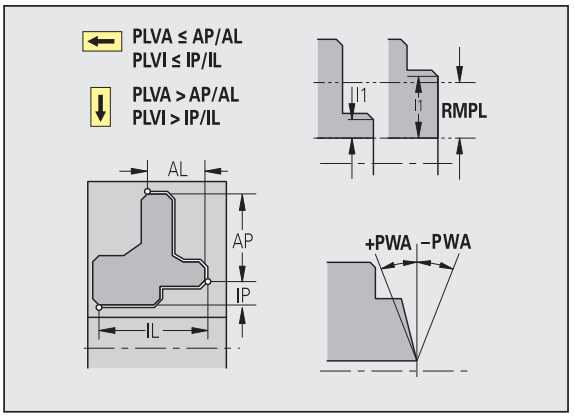
- **Approach – external roughing [ANRA]**
- **Approach – internal roughing [ANRI]**
- **Departure – external roughing [ABRA]**
- **Departure – internal roughing [ABRI]**

Approach/departure strategy:

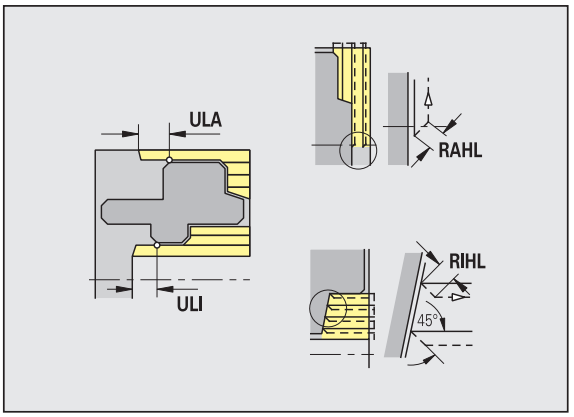
- 1: Move simultaneously in X and Z directions
- 2: First X, then Z direction
- 3: First Z, then X direction
- 6: Coupled motion; X precedes Z direction
- 7: Coupled motion; Z precedes X direction



TURN PLUS uses the PLVA/PLVI parameters to define whether a roughing area is to be rough-machined longitudinally or transversely.



Machining analysis
Transverse/longitudinal ratio – external [PLVA]
■ PLVA <= AP/AL: Longitudinal machining ■ PLVA > AP/AL: Transverse machining
Transverse/longitudinal ratio – internal [PLVI]
■ PLVI <= IP/IL: Longitudinal machining ■ PLVI >= IP/IL: Transverse machining
Minimum roughing transverse length [RMPL] (radius value)
Defines whether transverse roughing is used for the front transverse element of the outside contour of a finish part. ■ RMPL > I1: Without additional transverse roughing ■ RMPL < I1: With additional transverse roughing ■ RMPL = 0: Special case
Transverse angle variation [PWA]
The first front element is declared a transverse element when it is within +PWA and –PWA.



Fixed cycles

Overhang length outside [ULA]

Relative length for external rough-machining enabling roughing beyond the target position in longitudinal direction. ULA is not considered when the cutting limitation is in front of or within the overhang.

Overhang length inside [ULI]

- Relative length for internal rough-machining enabling roughing beyond the target position in longitudinal direction. ULA is not considered when the cutting limitation is in front of or within the overhang.
- This parameter is used for calculating the drilling depth for centric predrilling.

Retracting length outside [RAHL]

Retracting length for smoothing variants (H=1, 2) of roughing cycles G810 and G820 for external machining (RAHL).

Retracting length inside [RIHL]

Retracting length for smoothing variants (H=1, 2) of roughing cycles G810 and G820 for internal machining (RIHL).

Cutting depth reduction factor [SRF]

For rough-machining with tools machining opposite to the main machining direction, the infeed value (cutting depth) is reduced.

Infeed (P) for roughing cycles (G810, G820):

P = ZT * SRF

(ZT: Infeed value from technology database)

Machining standards

- **Tool angle – external/longitudinal [FALEW]**
- **Point angle – internal/longitudinal [FILEW]**
- **Tool angle – external/transverse [FAPEW]**
- **Point angle – internal/transverse [FIPEW]**

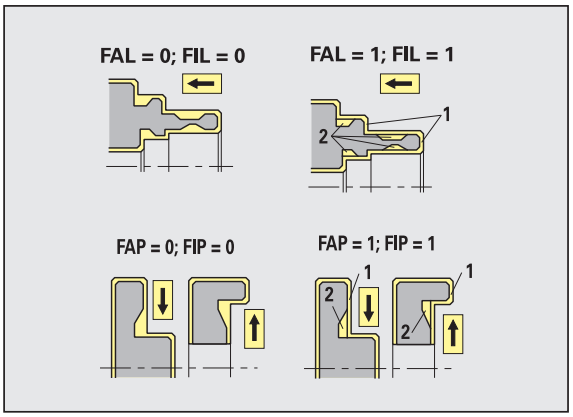
  Tool selection

  - Finishing cycles are primarily executed with standard finishing tools.
  - If form elements such as recesses (type FD) and undercuts (type E, F, G) cannot be machined with a standard finishing tool, one form element after the other is skipped. TURN PLUS starts a renewed attempt to machine the remaining contour with the standard tool. Subsequently, the skipped form elements are machined individually with a suitable finishing tool.

- **Standard/Full-surface – external/longitudinal [FAL]**
- **Standard/Full-surface – internal/longitudinal [FIL]**
- **Standard/Full-surface – external/transverse [FAP]**
- **Standard/Full-surface – internal/transverse [FIP]**

  Machining of contour areas:

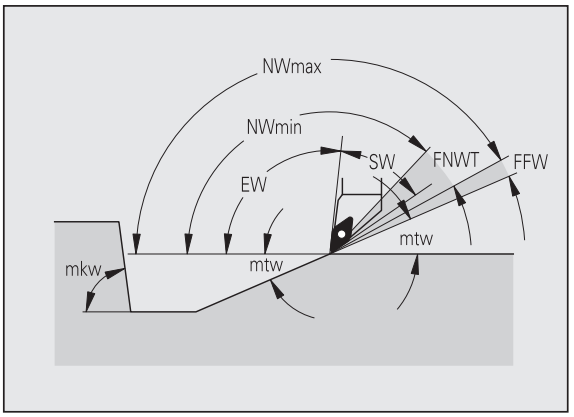
  - Complete: TURN PLUS searches for an optimum tool for machining the complete contour area.
  - Standard:
    - The machining operation is primarily executed with standard finishing tools. Relief turns and undercuts are machined with a suitable tool.
    - If the standard finishing tool cannot be used for relief turns and undercuts, TURN PLUS splits up the machining operation into standard machining operations and machining the form elements.
    - If a division into standard machining and form-element machining is not possible, TURN PLUS switches to "full-surface machining."



Finishing – Tool tolerances

For tool selection, the following applies:

- Tool angle (EW): EW >= mkw
  (mkw: ascending contour angle)
- Tool angle (EW) and point angle (SW):
  NWmin < (EW+SW) < NWmax
- Adjacent angle (FNWT): FNWT = NWmax – NWmin



Tool tolerances

Adjacent angle tolerance [FNWT]

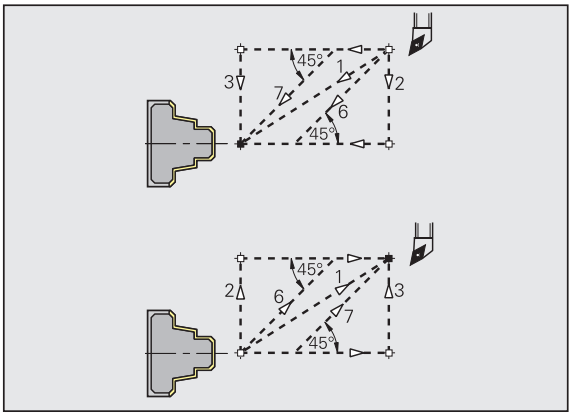
Tolerance range for secondary cutting edge

Relief cutting angle [FFW]

Minimum angle differential between the contour and secondary cutting edge

Finishing – Tool tolerances

Approach and departure are at rapid traverse (G0).



Approach and departure

- **Approach – external finishing [ANFA]**
- **Approach – internal finishing [ANFI]**
- **Departure – external finishing [ABFA]**
- **Departure – internal finishing [ABFI]**

Approach/departure strategy:

- 1: Move simultaneously in X and Z directions
- 2: First X, then Z direction
- 3: First Z, then X direction
- 6: Coupled motion; X precedes Z direction
- 7: Coupled motion; Z precedes X direction

Machining analysis

Minimum finishing transverse length [FMPL]

TURN PLUS checks the frontmost element of the outside contour to be finish-machined. Remember that:

- Without inside contour: Always with additional transverse cut
- With inside contour – FMPL >= I1: Without additional transverse cut
- With inside contour – FMPL < I1: With additional transverse cut

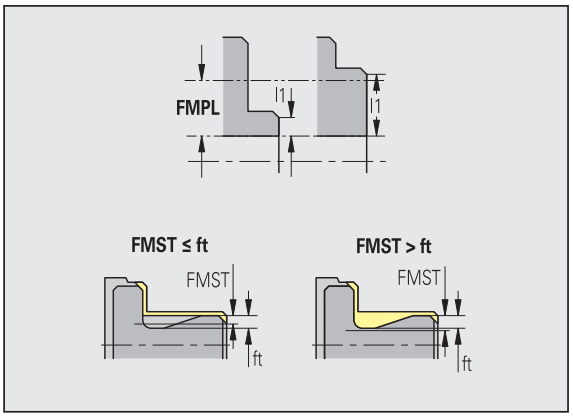
Maximum finishing cut depth [FMST]

FMST defines the permissible infeed depth for non-machined undercuts. The finishing cycle (G890) uses this parameter to determine whether undercuts (type E, F, G) will be machined with a contour-finishing operation. Remember that:

- FMST > ft: With undercut machining (ft: undercut depth)
- FMST <= ft: Without undercut machining

Number of revolutions for chamfer or rounding [FMUR]

The feed rate is reduced such that at least FMUR revolutions can be executed (evaluation: finishing cycle G890).



The following rules apply to FMPL:

- The additional transverse cut is executed from the outside toward the inside.
- The "Transverse angle variation PWA" has no effect on the analysis of the transverse elements.

Recessing and contour recessing

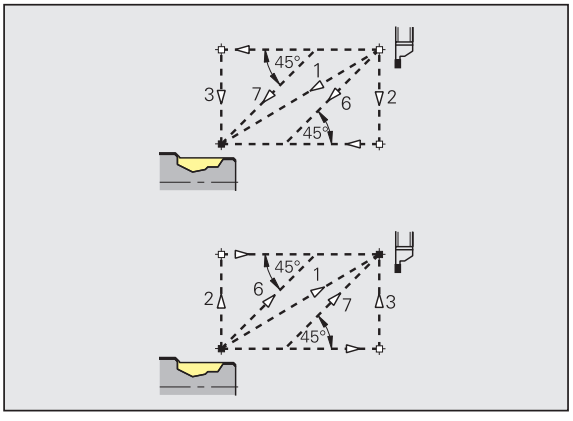
Recessing, contour recessing – Approach and departure

Approach and departure are at rapid traverse (G0).

Approach and departure

- **Approach – external recessing [ANESA]**
- **Approach – internal recessing [ANESI]**
- **Departure – external recessing [ABESA]**
- **Departure – internal recessing [ABESI]**

- **Approach – external contour recessing [ANKSA]**
- **Approach – internal contour recessing [ANKSI]**
- **Departure – external contour recessing [ABKSA]**
- **Departure – internal contour recessing [ABKSI]**



Approach and departure

Approach/departure strategy:

- 1: Move simultaneously in X and Z directions
- 2: First X, then Z direction
- 3: First Z, then X direction
- 6: Coupled motion; X precedes Z direction
- 7: Coupled motion; Z precedes X direction

Recessing, contour recessing – Tool selection, oversizes

Tool selection, oversizes

Recessing width divisor [SBD]

If a recess base machined with a contour-recessing cycle does not contain paraxial elements but only linear elements, a suitable tool is selected using the "recessing width divisor SBD."

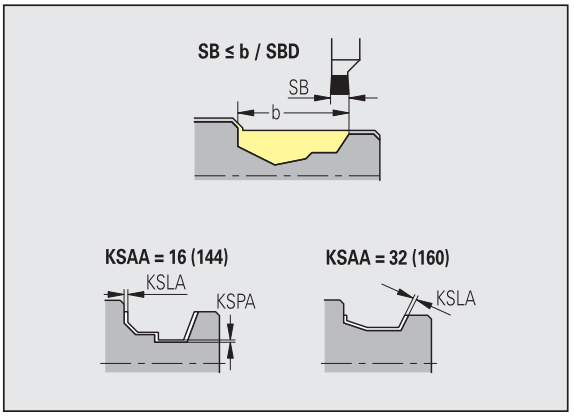
SB <= b / SBD

(SB: width of recessing tool; b: width of machining area)

Type of oversize [KSAA]

The recessing area to be machined can be assigned oversizes. When oversizes have been defined, a recess is first rough-machined and then finish-machined. Inputs:

- 16: Longitudinal and transverse oversizes differ – no single oversizes
- 144: Longitudinal and transverse oversizes differ – with single oversizes
- 32: Equidistant oversize – no single oversizes
- 160: Equidistant oversize – with single oversizes



Tool selection, oversizes

Equidistant or longitudinal [KSLA]

Equidistant oversize or longitudinal oversize

None or transverse [KSPA]

Transverse oversize

- The oversizes are accounted for when machining contour valleys with a contour-recessing operation.
- Standardized recesses such as recess types D, S, A are completed in one machining cycle. A division into rough-machining and finish-machining is only possible in DIN PLUS.

Recessing, contour recessing – Machining

Evaluation: DIN PLUS

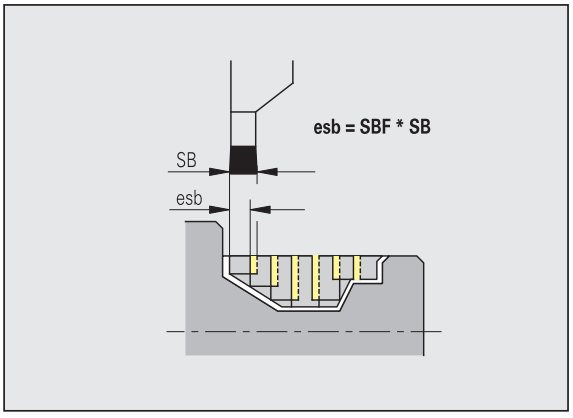
Machining

Recessing width factor [SBF]

SBF is used for calculating the maximum offset for recessing cycles G860, G866:

esb = SBF * SB

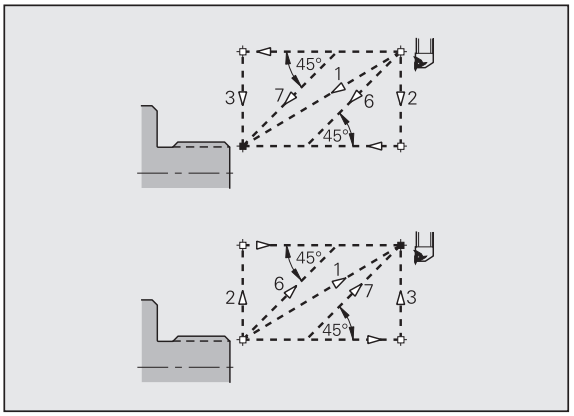
(esb: effective recessing width; SB: width of recessing tool)



Thread cutting

Thread cutting – Approach and departure

Approach and departure are at rapid traverse (G0).



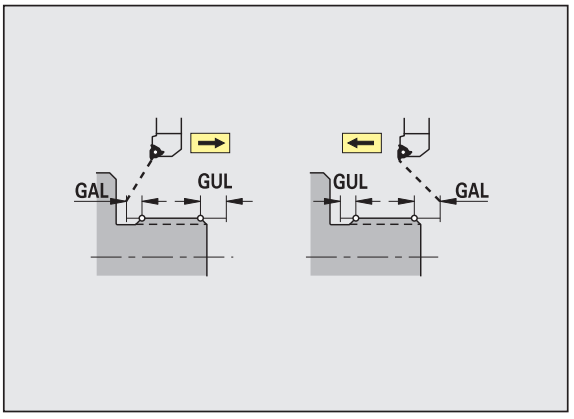
Approach and departure

- **Approach – external thread [ANGA]**
- **Approach – internal thread [ANGI]**
- **Departure – external thread [ABGA]**
- **Departure – internal thread [ABGI]**

  Approach/departure strategy:

  - 1: Move simultaneously in X and Z directions
  - 2: First X, then Z direction
  - 3: First Z, then X direction
  - 6: Coupled motion; X precedes Z direction
  - 7: Coupled motion; Z precedes X direction

Thread cutting – Machining



Machining

Thread starting length [GAL]

Starting length before threading cut.

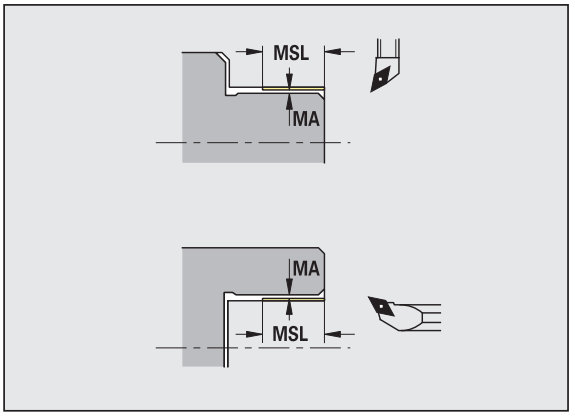
Thread run-out length [GUL]

Overrun length after threading cut.

GAL/GUL are automatically transferred to the thread attributes "starting length B/overrun length P" if they have not been entered as attributes.

Measuring

The measuring parameters are assigned to the fit elements as an attribute.

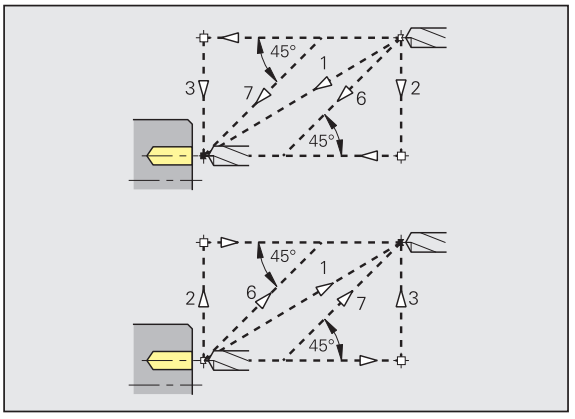


Measurement procedure
Measuring loop counter [MC]
Defines the measurement/loop intervals
Measuring path length in Z [MLZ]
Distance in Z for departure movement
Measuring path length in X [MLX]
Distance in X for departure movement
Measuring oversize [MA]
Oversize still applied to the element to be measured.
Measuring cut length [MSL]

Drilling

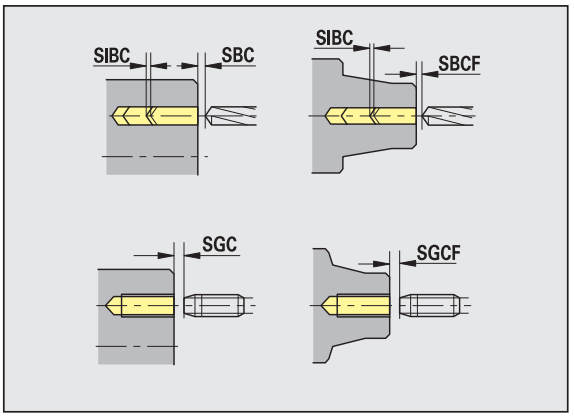
Drilling – Approach and departure

Approach and departure are at rapid traverse (G0).



Approach and departure
■ **Approach – face [ANBS]**
■ **Approach – lateral surface [ANBM]**
■ **Departure – face [ABGA]**
■ **Departure – lateral surface [ABBM]**
Approach/departure strategy:
■ 1: Move simultaneously in X and Z directions
■ 2: First X, then Z direction
■ 3: First Z, then X direction
■ 6: Coupled motion; X precedes Z direction
■ 7: Coupled motion; Z precedes X direction

Drilling – Safety clearances



Safety clearances

Internal safety clearance [SIBC]

Retraction distance for deep-hole drilling ("B" for G74)

Driven drills [SBC]

Safety clearance for driven tools on face and lateral surface.

Stationary drills [SBCF]

Safety clearance on face and lateral surface for tools that are not driven.

Driven taps [SGC]

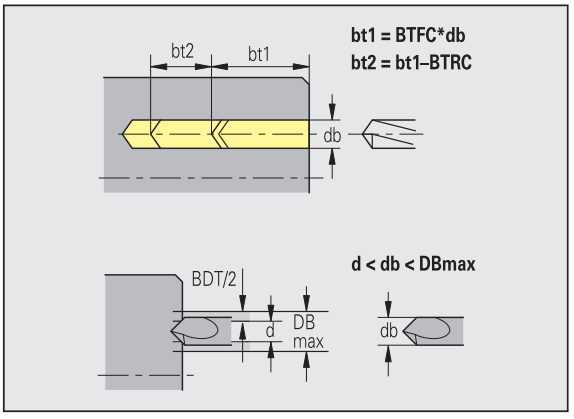
Safety clearance for driven tools on face and lateral surface.

Stationary taps [SGCF]

Safety clearance on face and lateral surface for tools that are not driven.

Drilling – Machining

The parameters apply to drilling with deep-hole drilling cycle (G74).



Machining

Drilling depth factor [BTFC]

1st drilling depth: bt1 = BTFC * db

(db: drill diameter)

Depth reduction [BTRC]

2nd drilling depth: bt2 = bt1 – BTRC

The subsequent drilling steps are reduced accordingly.

Diameter tolerance for drill [BDT]

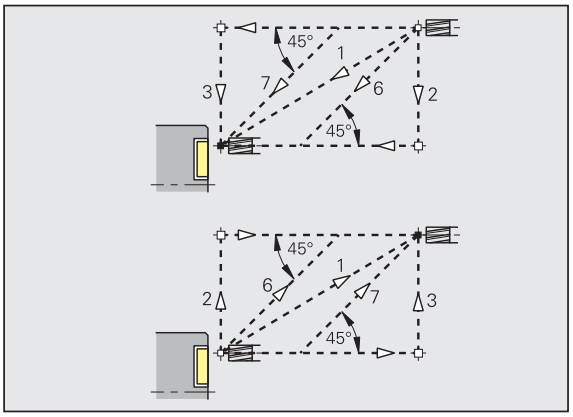
For selecting the desired drill (centering drills, countersinks, stepped drill, taper reamers).

- Drilling diameter: DBmax = BDT + d (DBmax: maximum drilling diameter)
- Tool selection: DBmax > DB > d

Milling

Milling – Approach and departure

Approach and departure are at rapid traverse (G0).



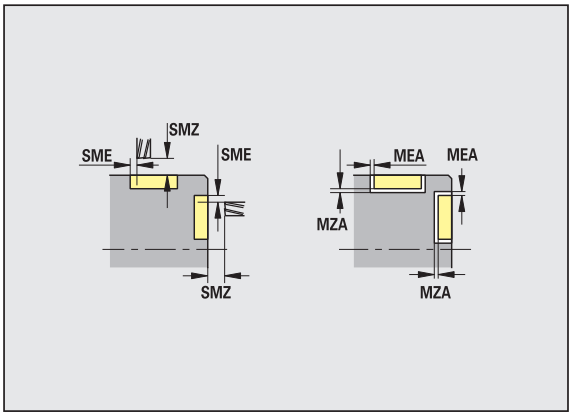
Approach and departure

- **Approach – end face [ANMS]**
- **Approach – lateral surface [ANMM]**
- **Departure – end face [ABMS]**
- **Departure – lateral surface [ABMM]**

Approach/departure strategy:

- 1: Move simultaneously in X and Z directions
- 2: First X, then Z direction
- 3: First Z, then X direction
- 6: Coupled motion; X precedes Z direction
- 7: Coupled motion; Z precedes X direction

Milling – Safety clearances and oversizes



Safety clearances and oversizes

Safety clearance in infeed direction [SMZ]

Distance between starting position and top edge of object to be milled.

Safety clearance in milling direction [SME]

Distance between milling contour and side of mill.

Oversize in milling direction [MEA]

Oversize in infeed direction [MZA]

8.3 Transfer

The Transfer mode is used for **data backup** and **data exchange** via networks or USB devices. When we speak of "files" in the following, we mean programs, parameters and tool data. The following file types can be transferred:

- Programs (cycle programs, smart.Turn programs, DIN main and subprograms, ICP contour descriptions)
- Parameters
- Tool data

Data backup

HEIDENHAIN recommends backing up the tool data and programs created on CNC PILOT on an external device at regular intervals.

You should also back up the parameters. Since the parameters are not changed very often, however, you only need to back up the parameters from time to time, as required.

Data exchange with TNCremo

HEIDENHAIN offers the PC program TNCremo to complement the CNC PILOT control. This program enables you to access the control data from a PC.

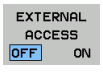
External access

The machine tool builder can configure the external access options. Refer to your machine manual.

The soft key SERVICE can be used to grant or restrict access through the LSV-2 interface.

Permitting/Restricting external access:

- Select the Organization mode



- Enable a connection to the control: Set the EXTERNAL ACCESS soft key to ON. The TNC will then permit data access through the LSV-2 interface.
- Disable connections to the control: Set the EXTERNAL ACCESS soft key to OFF. The TNC will then disable access through the LSV-2 interface

Connections

You can establish connections over the network (Ethernet) or with a USB storage device. Data is transferred over the **Ethernet** or **USB interface**.

- **Network** (via Ethernet): The CNC PILOT supports **SMB** networks (Server Message Block, WINDOWS) and **NFS** networks (Network File Service).
- **USB** storage devices can be connected directly to the control. The CNC PILOT uses only the first partition of a USB storage device.

Danger of collision!

Other computer systems in the network may overwrite CNC PILOT programs. Organize your network in such a way that only authorized persons have access to CNC PILOT.

You can also create new folders on a connected USB data carrier or network drive: Press the **Create transfer folder** soft key and enter a name for the folder.

The control displays all active connections in a selection window. If a folder contains subfolders, you can also open and select these subfolders.

Select the Organization mode and log on with the code number "net123".

TRANSFER

Press the **Transfer** soft key (login required).

Con-nections

Press the **Connections** soft key.

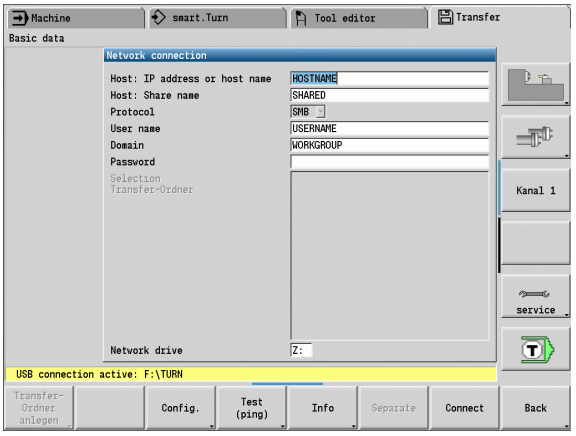
Network

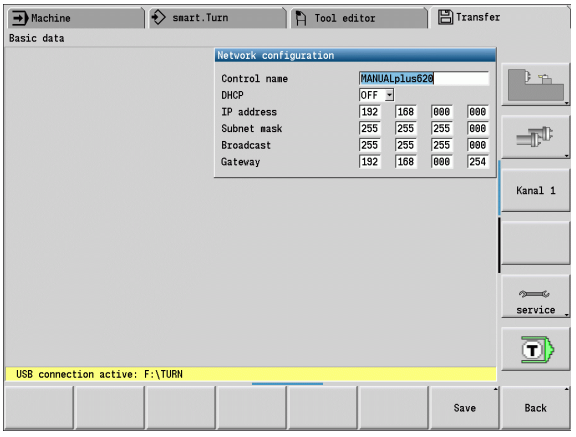
Press the **Network** soft key.

The CNC PILOT opens the **Network connection** dialog box. This dialog box is for the settings for the connection target.

Config.

Press the **Config.** soft key (login required). The **Network configuration** dialog box opens.





Ethernet interface CNC PILOT 620

Network configuration settings

- **Control name -** Computer name of the control
- **DHCP** (Dynamic Host Configuration Protocol)
  - **OFF:** The other network settings have to be configured manually. Static IP address.
  - **ON:** The network settings are automatically configured by a DHCP server.
- **Settings for DHCP OFF**
  - IP address
  - Subnet mask
  - Broadcast
  - Gateway

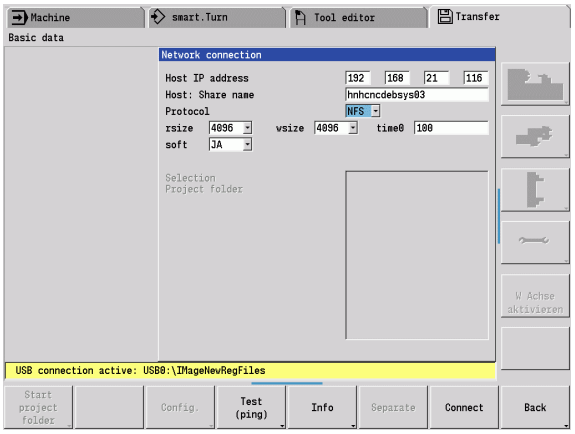
Network connection settings (SMB)

- **Protocol**
  - SMB – Windows network
- **Host IP address/host name** – Computer name or IP address of the target computer.
- **Host release** – Share name of the target computer. (Sharename)
- **User name** – For logging on to the target computer.
- **Workgroup/Domain** – Name of the workgroup/domain.
- **Password** – For logging on to the target computer.

Network connection settings (NFS)

- **Protocol**
  - NFS
- **Host IP address** – IP address of the target computer.
- **Host release** – Share name of the target computer. (Sharename)
- rsize -
- **wsize** -
- **timeO** -
- **soft** -

Select project folder: The CNC PILOT reads and writes all the data to/from a predefined project folder. Each project folder mirrors the folder structure on the control. Select the project folder you want to connect to. If no project folder exists in the target path, the folder is created when connecting.



Network configuration soft keys

Start project folder	Creates a folder of the specified name in the target path when a connection is established.
Config.	Opens the **Network configuration** dialog box.
Test (ping)	Opens the **Check network connection** dialog box and pings the specified target.
Info	Displays all the network information in a list box.
Separate	Terminates an existing network connection. When a USB storage device is active, the control switches to the USB connection.
Connect	Establishes the connection and opens the last project folder you selected.
Back	Returns to the Transfer soft-key menu.

Ethernet interface CNC PILOT 640

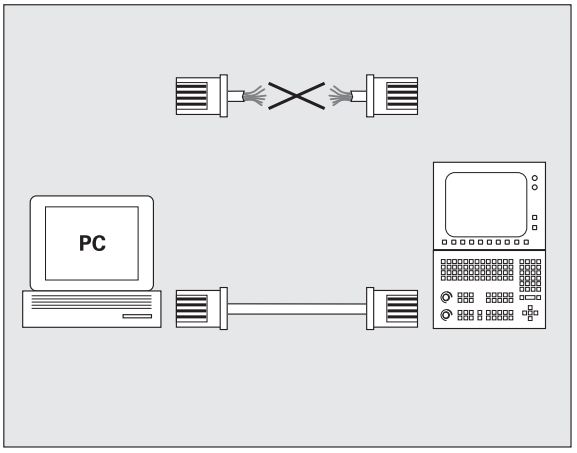
Introduction

The control is shipped with a standard Ethernet card to connect the control as a client in your network. The control transmits data via the Ethernet card with

- the **smb** protocol (Server Message Block) for Windows operating systems, or
- the **TCP/IP** protocol family (Transmission Control Protocol/Internet Protocol) and with support from the NFS (Network File System). The control also supports the NFS V3 protocol, which permits higher data transfer rates.

Connection possibilities

You can connect the Ethernet card in your control to your network through the RJ45 connection, or directly to a PC. The connection is metallically isolated from the control electronics.



The maximum cable length between control and a node depends on the quality grade of the cable, the sheathing and the type of network.

If you connect the control directly with a PC you must use a crossover cable.

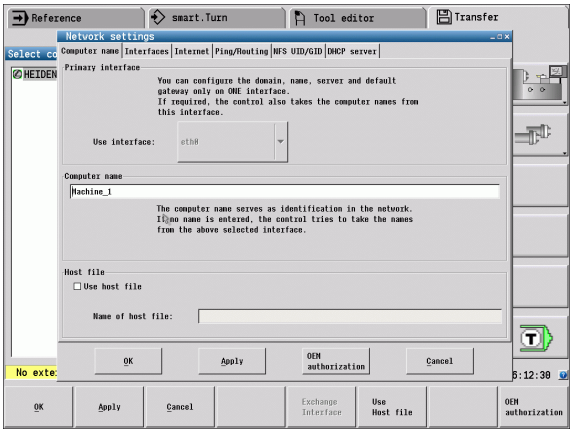
Make sure that the person configuring your control is a network specialist.

Please note that the control performs an automatic restart if you change the IP address of the control.

Control configuration

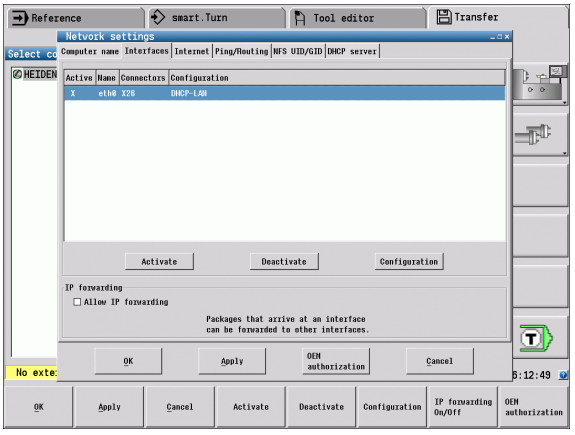
General network settings

▶ Press the DEFINE NET soft key to enter the general network settings. The **Computer name** tab is active:



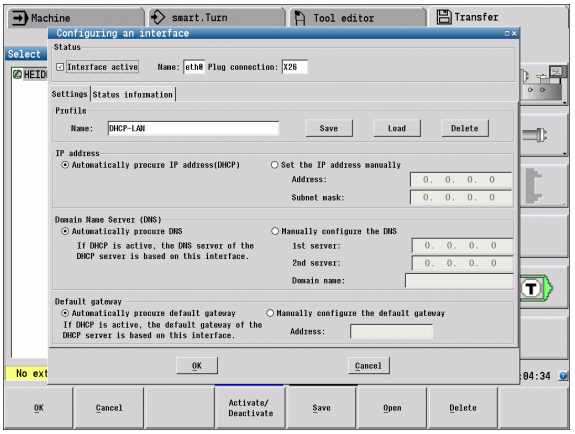
Setting	Meaning
Primary interface	Name of the Ethernet interface to be integrated in your company network. Only active if a second, optional Ethernet interface is available on the control hardware
Computer name	Name displayed for the control in your company network
Host file	**Only required for special applications**: Name of a file in which the assignments of IP addresses to computer names is defined

▶ Select the **Interfaces** tab to enter the interface settings:



Setting	Meaning
Interface list	List of the active Ethernet interfaces. Select one of the listed interfaces (via mouse or arrow keys) ■ **Activate** button: Activate the selected interface (an **X** appears in the **Active** column) ■ **Deactivate** button: Deactivate the selected interface (a hyphen **(-)** appears in the **Active** column) ■ **Configuration** button: Open the Configuration menu
Allow IP forwarding	**This function must be kept deactivated**. Only activate this function if external access via the second, optional Ethernet interface of the control is necessary for diagnostic purposes. Only do so after instruction by our Service Department

▶ Press the **Configuration** button to open the Configuration menu:

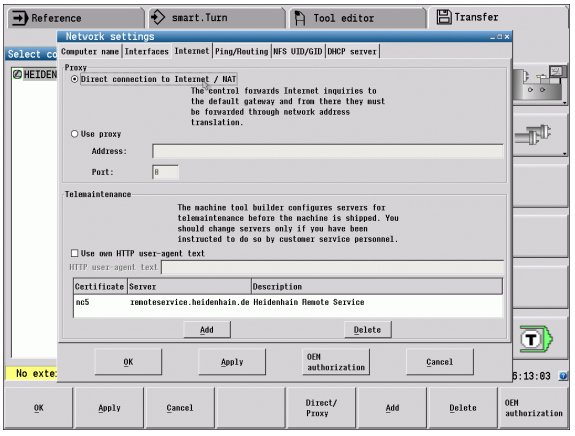


Setting	Meaning
Status	■ **Interface active**: Connection status of the selected Ethernet interface ■ **Name**: Name of the interface you are currently configuring ■ Plug connection: Number of the plug connection of this interface on the logic unit of the control
Profile	Here you can create or select a profile in which all settings shown in this window are stored. HEIDENHAIN provides two standard profiles: ■ **DHCP-LAN**: Settings for the standard Ethernet interface, should work in a standard company network. ■ **MachineNet**: Settings for the second, optional Ethernet interface; for configuration of the machine network Press the corresponding buttons to save, load and delete profiles
IP address	■ **Automatically procure IP address** option: The control is to procure the IP address from the DHCP server ■ **Set the IP address manually** option: Manually define the IP address and subnet mask. Input: Four numerical values separated by points, in each field, e.g. **160.1.180.20** and **255.255.0.0**

Setting	Meaning
Domain Name Server (DNS)	■ **Automatically procure DNS** option: The control is to automatically procure the IP address of the domain name server ■ **Manually configure the DNS** option: Manually enter the IP addresses of the servers and the domain name
Default gateway	■ **Automatically procure default gateway** option: The control is to automatically procure the default gateway ■ **Manually configure the default gateway** option: Manually enter the IP addresses of the default gateway

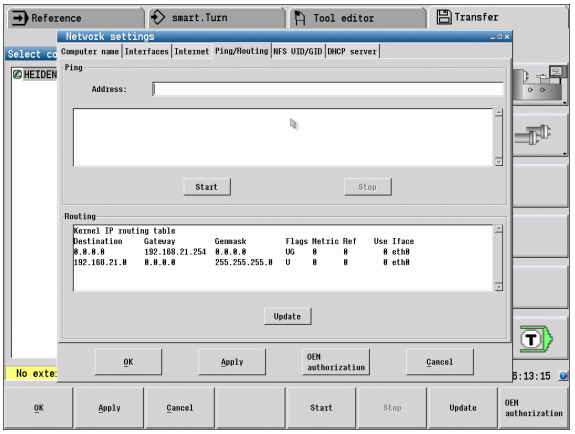
▶ Apply the changes with the **OK** button, or discard them with the **Cancel** button

▶ Select the **Internet** tab:



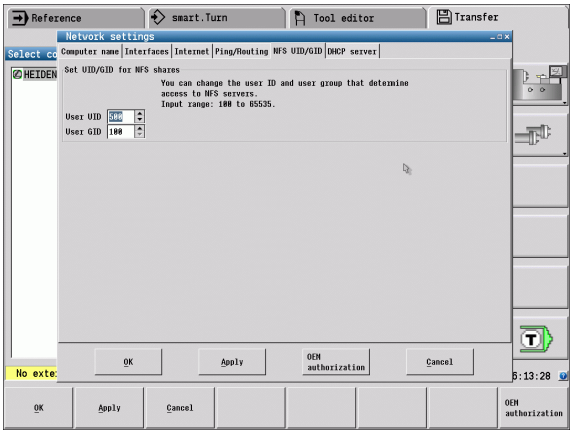
Setting	Meaning
Proxy	■ **Direct connection to Internet/NAT**: The control forwards Internet inquiries to the default gateway and from there they must be forwarded through network address translation (e.g. if a direct connection to a modem is available) ■ **Use proxy**: Define the **Address** and **Port** of the Internet router in your network, ask your network administrator for the correct address and port
Telemaintenance	The machine manufacturer configures the server for telemaintenance here. Changes must always be made in agreement with your machine tool builder

▶ Select the **Ping/Routing** tab to enter the ping and routing settings:



Setting	Meaning
Ping	In the **Address:** field, enter the IP number for which you want to check the network connection. Input: Four numerical values separated by periods, e.g. **160.1.180.20**. As an alternative, you can enter the name of the computer whose connection you want to check ■ Press the **Start** button to begin the test. The control shows the status information in the Ping field ■ Press the **Stop** button to conclude the test
Routing	For network specialists: Status information of the operating system for the current routing ■ **Update** button: Update routing

▶ Select the **NFS UID/GID** tab to enter the user and group identifications:

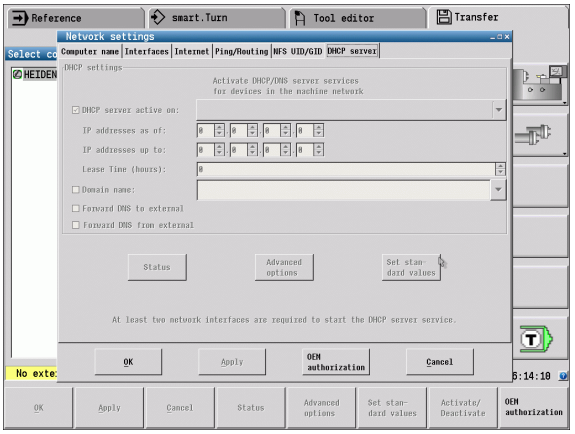


Setting	Meaning
Set UID/GID for NFS shares	■ **User ID**: Definition of which user identification the end user uses to access files in the network. Ask your network specialist for the proper value ■ **Group ID**: Definition of the group identification with which you access files in the network. Ask your network specialist for the proper value

▶ Select the **DHCP server** tab to configure the DHCP server settings of the machine network:

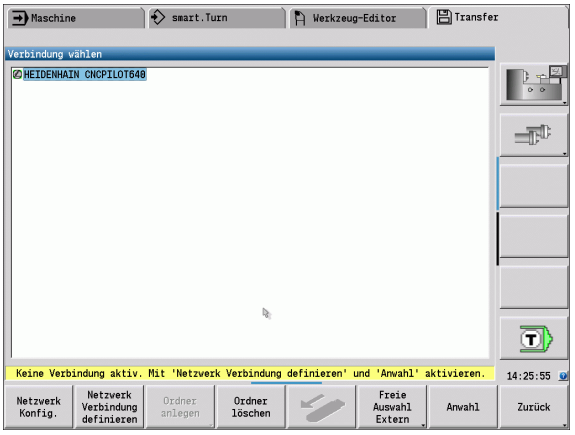
The configuration of the DHCP server is protected by password. Please contact the machine tool builder.

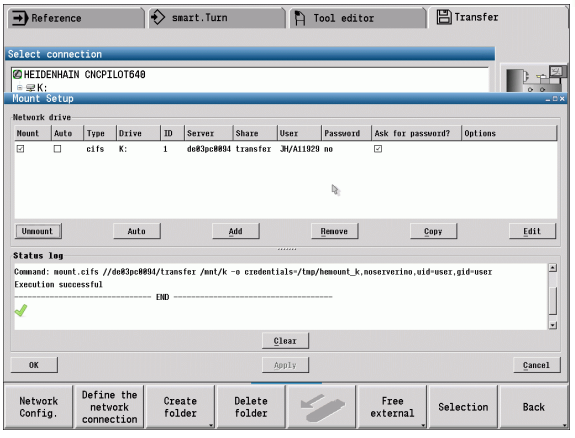
Setting	Meaning
DHCP server active on:	■ **IP addresses as of**: Define the IP address as of which the control is to derive the pool of dynamic IP addresses. The control transfers the values that appear dimmed from the static IP address of the defined Ethernet interface; these values cannot be edited. ■ **IP addresses up to**: Define the IP address up to which the control is to derive the pool of dynamic IP addresses. ■ **Lease Time (hours)**: Time within which the dynamic IP address is to remain reserved for a client. If a client logs on within this time, the control reassigns the same dynamic IP address. ■ **Domain name**: Here you can define a name for the machine network if required. This is necessary if the same names are assigned in the machine network and in the external network, for example. ■ **Forward DNS to external**: If **IP Forwarding** is active (**Interfaces** tab) and the option is active, you can specify that the name resolution for devices in the machine network can also be used by the external network. ■ **Forward DNS from external**: If **IP Forwarding** is active (**Interfaces** tab) and the option is active, you can specify that the control is to forward DNS inquiries from devices within the machine network to the name server of the external network if the DNS server of the MC cannot answer the inquiry. ■ **Status** button: Call an overview of the devices that are provided with a dynamic IP address in the machine network. You can also select settings for these devices. ■ **Advanced options** button: Additional settings for the DNS/DHCP server. ■ **Set standard values** button: Define the factory default settings.



Network settings specific to the device

▶ Press the **Network** soft key to enter the network settings for a specific device. You can define any number of network settings, but you can manage only seven at one time





Setting	Meaning
Network drive	List of all connected network drives. The control shows the respective status of the network connections in the columns: ■ **Mount**: Network drive connected / not connected ■ **Auto**: Network drive is to be connected automatically /manually ■ **Type**: Type of network connection. cifs and nfs are possible ■ **Drive**: Designation of the drive on the control ■ **ID**: Internal ID that identifies if a mount point has been used for more than one connection ■ **Server**: Name of the server ■ **Share name**: Name of the directory on the server that the control is to access ■ **User**: User name with which the user logs on to the network ■ **Password**: Network drive password protected / not protected ■ **Ask for password?**: Request / Do not request password during connection ■ **Options**: Display additional connection options To manage the network drives, use the screen buttons. To add network drives, use the **Add** button: The control then starts the connection wizard, which guides you by dialog through the required definitions.

USB connection

Select the Organization mode and plug the USB storage device in at the USB port on the CNC PILOT.

TRANSFER

Press the **Transfer** soft key (login required).

Con-
nections

Press the **Connections** soft key.

USB

Press the **USB** soft key.

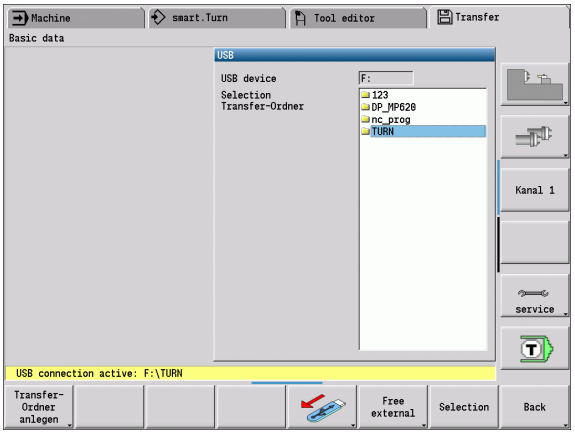
The CNC PILOT opens the **USB** dialog box. This dialog box is for the settings for the connection target.





Use the soft keys to disconnect and reconnect USB storage devices.

In principle, it should be possible to connect most USB devices to the control. It may happen that a USB device is not identified correctly by the control, for example when the cable between the control panel and the main computer is very long. In such cases, use another USB device.



Soft keys for USB connection

Start project folder	Creates a folder of the specified name on the USB storage device.
	Disconnects the USB storage device and prepares the device for removal.
Free external	Allows access to files that have not been properly saved in a project folder.
Selection	Selects the project folder highlighted previously using the arrow keys.
Back	Returns to the Transfer soft-key menu.

Data transfer options

The CNC PILOT manages DIN programs, DIN subprograms, cycle programs and ICP contours in different directories. When you select "Program group," the control automatically switches to the applicable directory.

Parameters and tool data are stored under the file name entered for **Backup name** and saved to a ZIP file located in the "para" or "tool" folder on the control. You can then send this backup file to a project folder in the remote station.



- If program files are opened in another operating mode, they are not overwritten.
- You can only load tool data and parameters when no program is executed in Program Run mode.

The following transfer functions are available:

- **Programs**: Transmitting and receiving files
- **Back up parameters**: Creating, transmitting and receiving
- **Restore parameters**: Reloading the parameter backup files
- **Backup tools**: Creating, transmitting and receiving
- **Restore tools**: Reloading the tool backup files
- **Service data**: Creating and transmitting service data
- **Make data backup**: Saving **all** data in a project folder
- **Free external**: Freely selecting program files on a USB storage device
- **Misc. functions**: Importing the cycle programs and DIN programs of the MANUALplus 4110

Transfer folder
The control exchanges data with an external storage device by accessing transfer folders created in the storage device. Files in transfer folders are organized using the same folder structure as on the control.

Transfer folders can only be used directly in the selected network path or in the root directory of the USB storage device.

Folder structure – Storage of files

Folders	File types
\dxf	Drawings in DXF format
\gtb	Machining sequences (TURN PLUS)
\gti	ICP contour descriptions ■ *.gmi (turning contour) ■ *.gmr (workpiece-blank contour) ■ *.gms (C axis, face) ■ *.gmm (C axis, surface)
\gtz	Cycle programs (teach-in) ■ *.gmz
\ncps	DIN programs (smart.Turn) ■ *.nc (main programs) ■ *.ncs (subprograms)
\para	Parameter backup files ■ PA_*.zip (parameters)
\table	Parameter backup files ■ TA*.zip (tables)
\tool	Tool backup files ■ TO*.zip (tool data and technology data)
\pictures	Image files for subprograms ■ *.bmp/png/jpg
\data	Service files ■ Service*.zip

Transferring programs (files)

Selecting the program group

TRANSFER — Press the **Transfer** soft key (login required).

Con-nections — Press the **Connections** soft key.

USB — Press the **USB** soft key.

Network — Press the **Network** soft key.

Selection — Select a project folder and press the **Selection** soft key (USB), or the

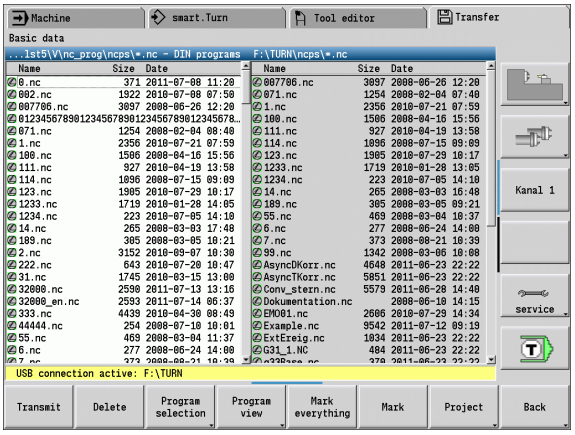
Connect — **Connect** (network) soft key.

Back — Return to data selection.

Programs — Switch to program transfer.

Program selection — Open a selection of program types.

DIN programs — Activate DIN programs (or other program types) for transfer.



Soft keys for program group selection

DIN programs	***.nc:** DIN and smart.Turn main programs. The transfer function scans the programs for subprograms and asks whether you want to transfer them as well.
DIN subprogram	***.ncs:** DIN and smart.Turn subprograms. Help graphics assigned to the subprogram are also transferred.
Cycle programs	***.gmz:** Cycle programs. The transfer function scans the programs for subprograms and ICP contours, and asks whether you want to transfer them as well.
ICP contours	ICP contours for cycle programs ■ ***.gmi** (turning contour) ■ ***.gmr** (workpiece-blank contour) ■ ***.gms** (C axis, face) ■ ***.gmm** (C axis, lateral surface)
Free external	Allows selecting program files on a USB storage device without using a project folder.
File mask	Masking the file names within the selected program group.

Selecting the program

In the window on the left, the CNC PILOT shows a file list of the control. The files of the remote station are displayed in the window on the right when a connection is established. With the **cursor keys** you can switch back and forth between the two windows.

When selecting a program, place the cursor on the desired program and press the **Mark** soft key. You can also select all programs with **Mark everything.**

Marked programs are highlighted in color. To unmark a program, simply press **Mark** once again.

The CNC PILOT displays the file size and the time of the last modification of the program in the list if permitted by the length of the file name.

With DIN programs/subprograms, you can also view the NC program with the **Program view** soft key.

Press the **Transmit** or **Receive** soft key to start transferring the files.

During transfer, the CNC PILOT displays the following information in a **transfer window** (see illustration):

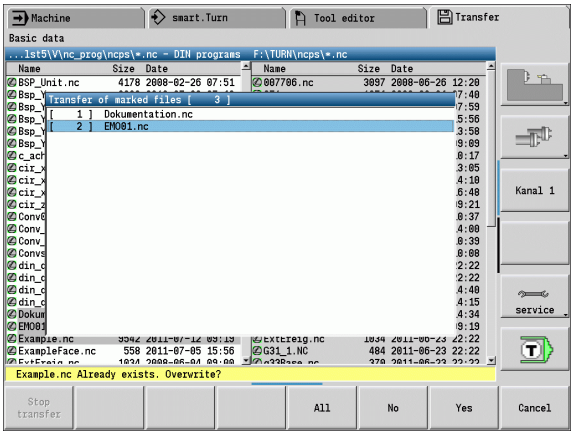
- The name of the program which is currently being transferred.
- If a file already exists in the target path, the CNC PILOT asks whether you want to overwrite the file. You can then activate overwriting for only that file or for all files in the transfer.

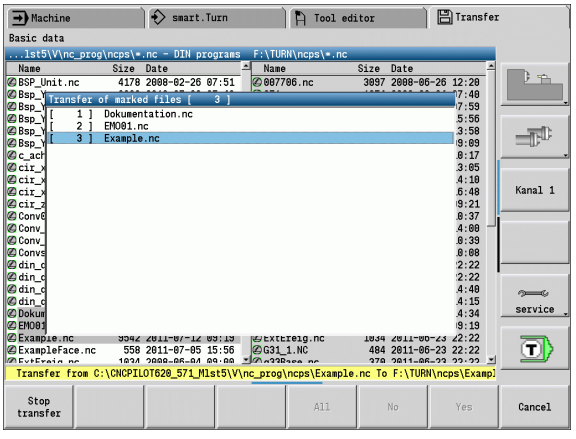
If the CNC PILOT detects during transfer that the transferred files are linked to other files (subprograms, ICP contours), a dialog box opens where you can select the linked files for transfer.

Transferring project files

If you wish to transfer files from a project, the "Project" soft key enables you to open the project management of the control and select the corresponding project (see "Project management" on page 118).

The **Internal project** soft key enables you to manage projects and transfer complete project folders (see also "Project management" on page 118).





Soft keys for program selection	
Mark everything	Marks all files in the current window.
Mark	Marks or unmarks the file at the cursor position and moves the cursor down one position.
Program view	Opens a DIN main program or subprogram for reading.

Transferring parameters

Parameters are backed up in two steps:

- **Creating a parameter backup:** The parameters are archived in ZIP files and stored in the control.
- **Transmitting/Receiving** the parameter backup files.
- **Restoring parameters:** Restores the backup files into the active data on the CNC PILOT (login required).

Parameter selection

You can also create a parameter backup without connecting to an external storage device.

TRANSFER	Press the **Transfer** soft key (login required).
Parameters	Open the parameter transfer.

Parameter backup data

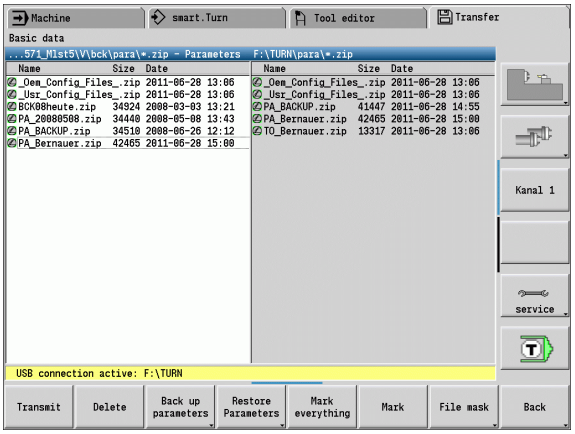
A parameter backup file contains all the parameters and tables of the CNC PILOT except tool and technology data.

Path and file names of the backup files:

- Config data: \para\PA_*.zip
- Tables: \table\TA_*.zip

The transfer window only shows the "para" folder. The associated file in the "table" folder is automatically created and transferred.

Press the **Transmit** or **Receive** soft key to start transferring the files.



Soft keys for parameter transfer

Transmit	Transmits all marked files from the control to the remote station.
Receive	Receives all files marked on the remote station.
Delete	Deletes all marked files after a confirmation prompt (login required).
Back up parameters	Creates a parameter backup data record as a .zip file.
Restore Parameters	Restores data from the selected backup data record to the active control system (login required).
Mark everything	Marks all files in the current window.
Mark	Marks or unmarks the file at the cursor position and moves the cursor down one position.

Transferring tool data

Tools are backed up in two steps:

- **Creating a tool backup:** The parameters are archived in ZIP files and stored in the control.
- **Transmitting/Receiving** the tool backup files.
- **Restoring tools:** Restores the backup files into the active data on the CNC PILOT (login required).

Tool selection

You can also create a tool backup without connecting to an external storage device.

TRANSFER	Press the **Transfer** soft key (login required).
Tools	Open the tool transfer.

Tool backup data

If you want to create a backup, you can define which tool data you want to save in a selection window.

Selection for the content of backup files:

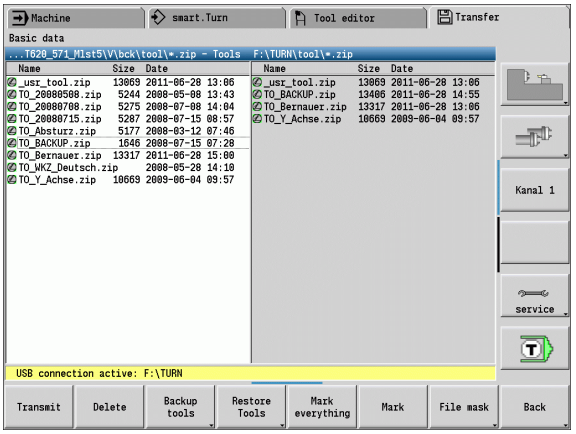
- Tools
- Tool texts
- Technology data
- Touch probes
- Tool holders

Path and file names of the backup files:

- \bck\tool\TO_*.zip

Press the **Transmit** or **Receive** soft key to start transferring the files.

When restoring backup data, all available content of the backup is displayed in a selection window. Here you can select the tool data you want to read in.



Soft keys for tool transfer

Transmit	Transmits all marked files from the control to the remote station.
Receive	Receives all files marked on the remote station.
Delete	Deletes all marked files after a confirmation prompt (login required).
Backup tools	Creates a tool backup data record as a .zip file.
Restore Tools	Restores data from the now selected backup data record to the active control system (login required).
Mark everything	Marks all files in the current window.
Mark	Marks or unmarks the file at the cursor position and moves the cursor down one position.
File mask	Selects the ZIP or HTT file type. The tool data can also be transferred directly as a HTT file (e.g. from a tool presetter).

Service files

Service files contain various log files used by the service department for troubleshooting. All important information is summarized in a services file record as a zip file.

Path and file names of the backup files:

- \data\SERVICEx.zip ("x" stands for a consecutive number)

The CNC PILOT always generates the service file with the number "1". Already existing files are renamed to the numbers 2 to 5. An existing file with the number 5 is deleted.

- **Creating service files:** The information is summarized in a ZIP file and stored in the control.
- **Transmit** service files.

Service selection

You can also create service files without connecting to an external storage device.

TRANSFER	Press the **Transfer** soft key (login required).
Service	Open the service data transfer.
Save	Enter a file name under which the service file is saved and confirm with the **Save** soft key.

Soft keys for transferring service files

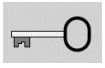
Transmit	Transmits all marked files from the control to the remote station.
Delete	Deletes all marked files after a confirmation prompt (login required).
Mark everything	Marks all files in the current window.
Mark	Marks or unmarks the file at the cursor position and moves the cursor down one position.
Create Service files	Creates a service files data record as a .zip file.

Creating a data backup file

A data backup performs the following steps:

- Copies the program files to the transfer folder.
  - NC main programs
  - NC subprograms (with graphics)
  - Cycle programs
  - ICP contours
- Creates a parameter backup and copies all backup files from "\para" and "\table" to the project folder. (PA_Backup.zip, TA_Backup.zip)
- Creates a tool backup file and copies all tool backup files from "\tool" to the project folder (TO_Backup.zip).
- Service files are **not** created and copied.

Data backup selection



Press the soft key and enter the login code.

TRANSFER

Press the **Transfer** soft key.

Data Backup

Open the data backup transfer.



- Existing files will be overwritten without a confirmation prompt.
- To cancel the data backup, press the **Cancel** soft key. The backup process stops as soon as the currently running partial backup is completed.

Soft keys for data backup

Start backup	Starts backing up the data to a complete transfer folder.

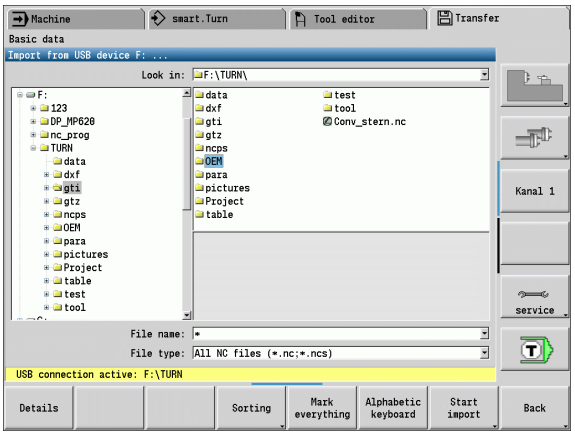
Importing NC programs from predecessor controls

The program formats of the predecessor controls MANUALplus 4110 and CNC PILOT 4290 differ from the format of the CNC PILOT 640. However, you can use the program converter to adapt programs of the predecessor control to the new control. This converter is a component of the CNC PILOT. The converter completes the required adaptations as automatically as possible.

Overview of convertible NC part programs:

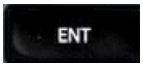
- MANUALplus 4110
  - Cycle programs
  - ICP contour descriptions
  - DIN programs
- CNC PILOT 4290: DIN PLUS programs

TURN PLUS programs of the CNC PILOT 4290 cannot be converted.



Importing NC programs from the connected data medium

TRANSFER	Press the **Transfer** soft key (login required).
Misc. functions	Open the menu with the miscellaneous functions.
Import functions	Open the menu with the import functions.
Cycle programs	Selection of cycle programs or ICP contours of the MANUALplus 4110 (*.gtz).
DIN programs	Selection of DIN programs ...
4110	... of the MANUALplus 4110 (*.nc/ *.ncs).
DIN programs	Selection of DIN programs ...
4290	... of the CNC PILOT 4290 (*.nc/ *.ncs).



Use the cursor keys to select the folder, then press the Enter key to switch to the right window.

With the cursor keys, select the NC program to be converted.



Mark all NC programs.



Start the import filter for converting the program(s) to the CNC PILOT format.

The names of imported cycle programs, ICP contour descriptions, DIN programs and DIN subprograms are given the prefix "CONV_..." In addition, the CNC PILOT adapts the extension and imports the NC programs to the correct folders.

Converting cycle programs

MANUALplus 4110 and CNC PILOT 640 have different solutions for tool management, technology data, etc. Also, the cycles of the CNC PILOT 640 have more parameters than those of the MANUALplus 4110.

Please note the following:

- **Tool call:** The loading of the T number depends on whether the program is a "multifix program" (2-digit T number) or "turret program" (4-digit T number).
  - 2-digit T number: The T number is loaded as "ID" and entered as the T number "T1".
  - 4-digit T number (Tddpp): The first two digits of the T number (dd) are loaded as "ID" and the last two (pp) as "T".
- **Moving to the tool change point:** In **`tool change point G14`**, the converter enters the "no axis" setting. In the MANUALplus 4110 this parameter is not used.
- **Safety clearance:** The safety clearances defined in the "general settings" parameter are entered by the converter in the **`safety clearance G47, ... SCI, ... SCK`**.

- **M functions** are left unchanged.
- **Calling ICP contours:** When an ICP contour is called, the converter prefixes "CONV_..." to the name.
- **Calling DIN cycles:** When a DIN cycle is called, the converter prefixes "CONV_..." to the name.



HEIDENHAIN recommends adapting converted NC programs to the circumstances of the CNC PILOT and then testing them before using them for production.

Converting DIN programs

DIN/ISO programs not only have new solutions for tool management, technology data, etc., but also for contour description and variable programming.

Remember the following when converting **DIN/ISO programs of the MANUALplus 4110**:

- **Tool call:** The loading of the T number depends on whether the program is a "multifix program" (2-digit T number) or "turret program" (4-digit T number).
  - 2-digit T number: The T number is loaded as "ID" and entered as the T number "T1".
  - 4-digit T number (Tddpp): The first two digits of the T number (dd) are loaded as "ID" and the last two (pp) as "T".
- **Workpiece-blank definition:** A G20/G21 workpiece-blank definition of the MANUALplus 4110 becomes an AUXILIARY BLANK on the CNC PILOT 640.
- **Contour descriptions:** In MANUALplus 4110 programs, the fixed cycles are followed by the contour description. During conversion the contour description is converted to an AUXILIARY CONTOUR. The associated cycle in the MACHINING section then refers to this auxiliary contour.
- **Variable programming:** Variable accesses to tool data, machine dimensions, D compensation values, parameter data and events cannot be converted. These program sequences have to be adapted.
- **M functions** are left unchanged.
- **Inches or metric:** The converter cannot detect the unit of measure of the MANUALplus 4110 program. Consequently, no unit of measure is entered in the target program. This has to be completed by the user.

Remember the following when converting **DIN programs of the CNC PILOT 4290**:

- **Tool call** (T commands of the TURRET section):
  - T commands containing a reference to the tool database are left unchanged (example: T1 ID“342-300.1“).
  - T commands containing tool data cannot be converted.
- **Variable programming:** Variable accesses to tool data, machine dimensions, D compensation values, parameter data and events cannot be converted. These program sequences have to be adapted.
- **M functions** are left unchanged.
- **Names of external subprograms:** When an external subprogram is called, the converter prefixes "CONV_..." to the name.



If the DIN program contains nonconvertible elements, the corresponding NC block is saved as a comment. The word WARNING is inserted in front of this comment. Depending on the situation, the nonconvertible command is taken into the comment line, or the nonconvertible NC block follows the comment.



HEIDENHAIN recommends adapting converted NC programs to the circumstances of the CNC PILOT and then testing them before using them for production.

8.4 Service pack

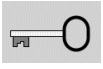
If changes or additional features are required in the control software, your machine tool builder will provide you with a service pack. The service pack is usually installed with the aid of a USB memory stick (1 GB or larger). The software required for the service pack is compressed in the **setup.zip** file. This file is saved on the USB stick.

Installing a service pack

The control has to be shut-down during the installation of the service pack. So end any NC program editing, etc. before you begin installation.

HEIDENHAIN recommends backing up your data before installing the service pack (see page 563).

Connect a USB stick and switch to the Organization mode of operation.



Press soft key and enter the code number **231019**.

UPDATE DATA

Press the soft key. (If the soft key is not visible, switch the soft-key menu.)

LOAD ZIP

Press the soft key.

PATH

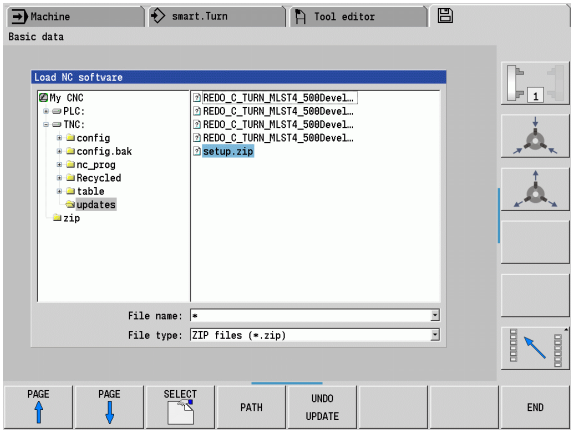
Press the **Path** soft key to select the directory in the left window.

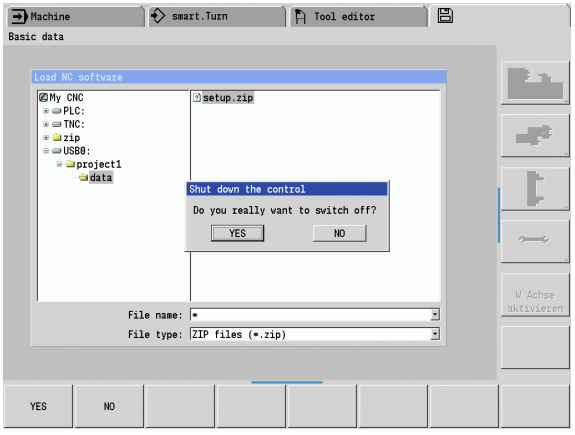
FILES

Press the **Files** soft key to select the file in the right window.

SELECT

Place the cursor on the "setup.zip" file and press the **SELECT** soft key.



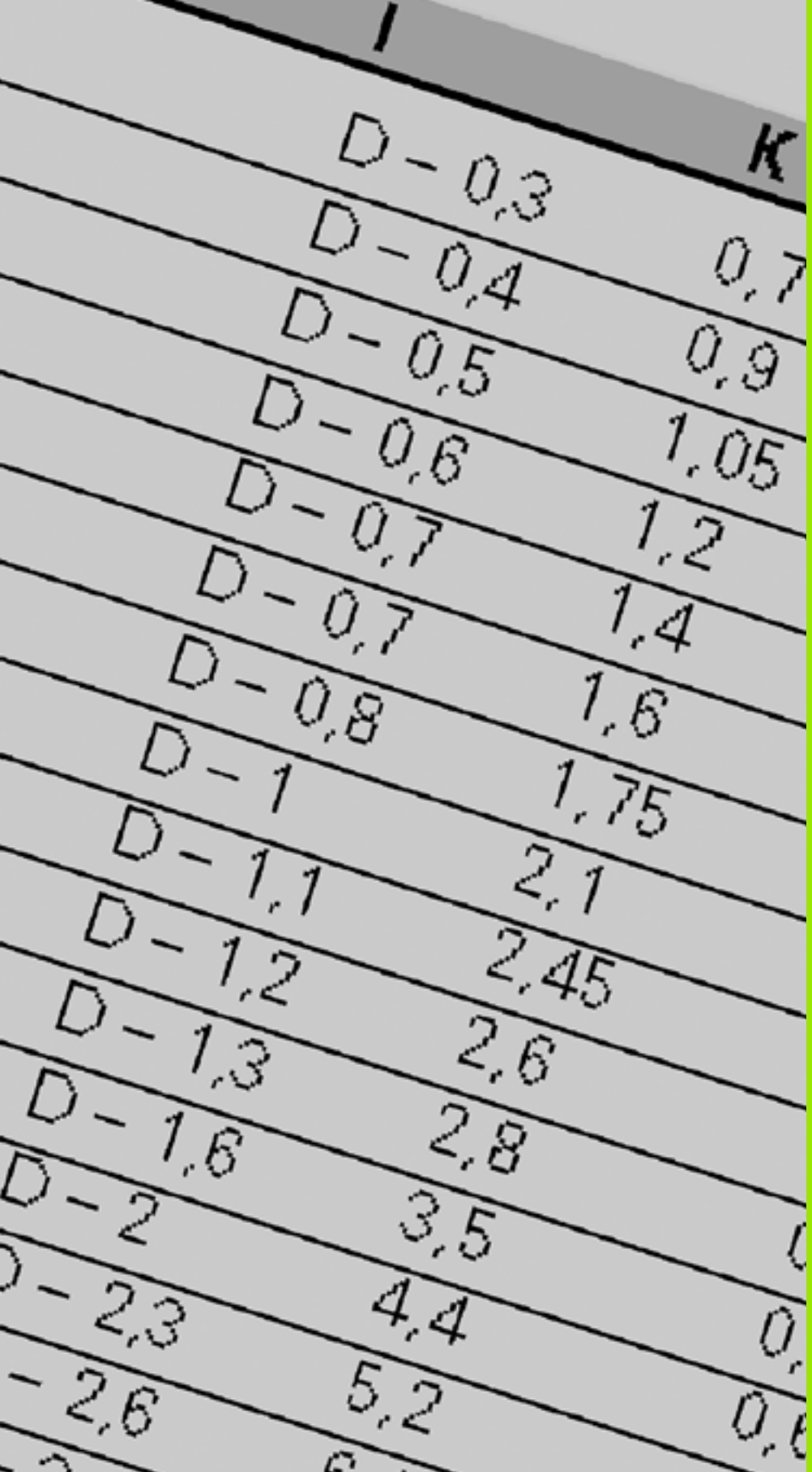


The CNC PILOT checks whether the service pack can be used for the current software version of the control.

Answer the confirmation prompt "Do you really want to switch off?" Then the actual update program starts.

Set the desired language (e.g. English) and do the update.

- After the software update the CNC PILOT automatically restarts.
- Keep the USB stick so that you can undo the installation of the service pack if necessary.



9

Tables and overviews

9.1 Thread pitch

Thread parameters

To determine the thread parameters, the CNC PILOT uses the following table.

Where:

- F: Thread pitch. Where an asterisk "*" is given in the table, the thread pitch is calculated from the diameter, depending on the thread type (See "Thread pitch" on page 573.) .
- P: Thread depth
- R: Thread width
- A: Thread angle, left
- W: Thread angle, right

Calculation formula: $Kb = 0.26384*F - 0.1*\sqrt{F}$

Thread backlash "ac" (depending on thread pitch):

- Thread pitch <= 1: ac = 0.15
- Thread pitch <= 2: ac = 0.25
- Thread pitch <= 6: ac = 0.5
- Thread pitch <= 13: ac = 1

Thread type Q		F	P	R	A	W
Q=1 Metric ISO fine-pitch thread	External	–	0.61343*F	F	30°	30°
	Internal	–	0.54127*F	F	30°	30°
Q=2 Metric ISO thread	External	*	0.61343*F	F	30°	30°
	Internal	*	0.54127*F	F	30°	30°
Q=3 Metric ISO tapered thread	External	–	0.61343*F	F	30°	30°
Q=4 Metric ISO tapered fine-pitch thread		–	0.61343*F	F	30°	30°
Q=5 Metric ISO trapezoid thread	External	–	0.5*F+ac	0.633*F	15°	15°
	Internal	–	0.5*F+ac	0.633*F	15°	15°
Q=6 Flat metric trapezoid thread	External	–	0.3*F+ac	0.527*F	15°	15°
	Internal	–	0.3*F+ac	0.527*F	15°	15°
Q=7 Metric buttress thread	External	–	0.86777*F	0.73616*F	3°	30°
	Internal	–	0.75*F	F–Kb	30°	3°
Q=8 Cylindrical round thread	External	*	0.5*F	F	15°	15°
	Internal	*	0.5*F	F	15°	15°
Q=9 Cylindrical Whitworth thread	External	*	0.64033*F	F	27,5°	27,5°
	Internal	*	0.64033*F	F	27,5°	27,5°
Q=10 Tapered Whitworth thread	External	*	0.640327*F	F	27,5°	27,5°
Q=11 Whitworth pipe thread	External	*	0.640327*F	F	27,5°	27,5°
	Internal	*	0.640327*F	F	27,5°	27,5°

Thread type Q		F	P	R	A	W
Q=12 Nonstandard thread		–	–	–	–	–
Q=13 UNC US coarse thread	External	*	0.61343*F	F	30°	30°
	Internal	*	0.54127*F	F	30°	30°
Q=14 UNF US fine-pitch thread	External	*	0.61343*F	F	30°	30°
	Internal	*	0.54127*F	F	30°	30°
Q=15 UNEF U.S. extra-fine-pitch thread	External	*	0.61343*F	F	30°	30°
	Internal	*	0.54127*F	F	30°	30°
Q=16 NPT U.S. taper pipe thread	External	*	0.8*F	F	30°	30°
	Internal	*	0.8*F	F	30°	30°
Q=17 NPTF U.S. taper dryseal pipe thread	External	*	0.8*F	F	30°	30°
	Internal	*	0.8*F	F	30°	30°
Q=18 NPSC U.S. cylindrical pipe thread with lubricant	External	*	0.8*F	F	30°	30°
	Internal	*	0.8*F	F	30°	30°
Q=19 NPFS U.S. cylindrical pipe thread without lubricant	External	*	0.8*F	F	30°	30°
	Internal	*	0.8*F	F	30°	30°

Thread pitch

Q = 2 Metric ISO thread

Diameter	Thread pitch	Diameter	Thread pitch	Diameter	Thread pitch
1	0,25	6	1	27	3
1,1	0,25	7	1	30	3,5
1,2	0,25	8	1,25	33	3,5
1,4	0,3	9	1,25	36	4
1,6	0,35	10	1,5	39	4
1,8	0,35	11	1,5	42	4,5
2	0,4	12	1,75	45	4,5
2,2	0,45	14	2	48	5
2,5	0,45	16	2	52	5
3	0,5	18	2,5	56	5,5
3,5	0,6	20	2,5	60	5,5
4	0,7	22	2,5	64	6
4,5	0,75	24	3	68	6
5	0,8				

Q = 8 Cylindrical round thread

Diameter	Thread pitch
12	2,54
14	3,175
40	4,233
105	6,35
200	6,35

Q = 9 Cylindrical Whitworth thread

Thread designation	Diameter (in mm)	Thread pitch	Thread designation	Diameter (in mm)	Thread pitch
1/4“	6,35	1,27	1 1/4“	31,751	3,629
5/16“	7,938	1,411	1 3/8“	34,926	4,233
3/8“	9,525	1,588	1 1/2“	38,101	4,233
7/16“	11,113	1,814	1 5/8“	41,277	5,08
1/2“	12,7	2,117	1 3/4“	44,452	5,08
5/8“	15,876	2,309	1 7/8“	47,627	5,645
3/4“	19,051	2,54	2“	50,802	5,645
7/8“	22,226	2,822	2 1/4“	57,152	6,35
1“	25,401	3,175	2 1/2“	63,502	6,35
1 1/8“	28,576	3,629	2 3/4“	69,853	7,257

Q = 10 Tapered Whitworth thread

Thread designation	Diameter (in mm)	Thread pitch	Thread designation	Diameter (in mm)	Thread pitch
1/16“	7,723	0,907	1 1/2“	47,803	2,309
1/8“	9,728	0,907	2“	59,614	2,309
1/4“	13,157	1,337	2 1/2“	75,184	2,309
3/8“	16,662	1,337	3“	87,884	2,309
1/2“	20,995	1,814	4“	113,03	2,309
3/4“	26,441	1,814	5“	138,43	2,309
1“	33,249	2,309	6“	163,83	2,309
1 1/4“	41,91	2,309			

Q = 11 Whitworth pipe thread

Thread designation	Diameter (in mm)	Thread pitch	Thread designation	Diameter (in mm)	Thread pitch
1/8“	9,728	0,907	2“	59,614	2,309
1/4“	13,157	1,337	2 1/4“	65,71	2,309
3/8“	16,662	1,337	2 1/2“	75,184	2,309
1/2“	20,995	1,814	2 3/4“	81,534	2,309
5/8“	22,911	1,814	3“	87,884	2,309
3/4“	26,441	1,814	3 1/4“	93,98	2,309
7/8“	30,201	1,814	3 1/2“	100,33	2,309
1“	33,249	2,309	3 3/4“	106,68	2,309
1 1/8“	37,897	2,309	4“	113,03	2,309
1 1/4“	41,91	2,309	4 1/2“	125,73	2,309
1 3/8“	44,323	2,309	5“	138,43	2,309
1 1/2“	47,803	2,309	5 1/2“	151,13	2,309
1 3/4“	53,746	1,814	6“	163,83	2,309

Q = 13 UNC US coarse thread

Thread designation	Diameter (in mm)	Thread pitch	Thread designation	Diameter (in mm)	Thread pitch
0,073“	1,8542	0,396875	7/8“	22,225	2,822222222
0,086“	2,1844	0,453571428	1“	25,4	3,175
0,099“	2,5146	0,529166666	1 1/8“	28,575	3,628571429
0,112“	2,8448	0,635	1 1/4“	31,75	3,628571429
0,125“	3,175	0,635	1 3/8“	34,925	4,233333333
0,138“	3,5052	0,79375	1 1/2“	38,1	4,233333333
0,164“	4,1656	0,79375	1 3/4“	44,45	5,08
0,19“	4,826	1,058333333	2“	50,8	5,644444444
0,216“	5,4864	1,058333333	2 1/4“	57,15	5,644444444
1/4“	6,35	1,27	2 1/2“	63,5	6,35
5/16“	7,9375	1,411111111	2 3/4“	69,85	6,35
3/8“	9,525	1,5875	3“	76,2	6,35
7/16“	11,1125	1,814285714	3 1/4“	82,55	6,35
1/2“	12,7	1,953846154	3 1/2“	88,9	6,35
9/16“	14,2875	2,116666667	3 3/4“	95,25	6,35
5/8“	15,875	2,309090909	4“	101,6	6,35
3/4“	19,05	2,54			

Q = 14 UNF US fine-pitch thread

Thread designation	Diameter (in mm)	Thread pitch	Thread designation	Diameter (in mm)	Thread pitch
0,06“	1,524	0,3175	3/8“	9,525	1,058333333
0,073“	1,8542	0,352777777	7/16“	11,1125	1,27
0,086“	2,1844	0,396875	1/2“	12,7	1,27
0,099“	2,5146	0,453571428	9/16“	14,2875	1,411111111
0,112“	2,8448	0,529166666	5/8“	15,875	1,411111111
0,125“	3,175	0,577272727	3/4“	19,05	1,5875
0,138“	3,5052	0,635	7/8“	22,225	1,814285714
0,164“	4,1656	0,705555555	1“	25,4	1,814285714
0,19“	4,826	0,79375	1 1/8“	28,575	2,116666667
0,216“	5,4864	0,907142857	1 1/4“	31,75	2,116666667
1/4“	6,35	0,907142857	1 3/8“	34,925	2,116666667
5/16“	7,9375	1,058333333	1 1/2“	38,1	2,116666667

Q = 15 UNEF US extra-fine-pitch thread

Thread designation	Diameter (in mm)	Thread pitch	Thread designation	Diameter (in mm)	Thread pitch
0,216“	5,4864	0,79375	1 1/16“	26,9875	1,411111111
1/4“	6,35	0,79375	1 1/8“	28,575	1,411111111
5/16“	7,9375	0,79375	1 3/16“	30,1625	1,411111111
3/8“	9,525	0,79375	1 1/4“	31,75	1,411111111
7/16“	11,1125	0,907142857	1 5/16“	33,3375	1,411111111
1/2“	12,7	0,907142857	1 3/8“	34,925	1,411111111
9/16“	14,2875	1,058333333	1 7/16“	36,5125	1,411111111
5/8“	15,875	1,058333333	1 1/2“	38,1	1,411111111
11/16“	17,4625	1,058333333	1 9/16“	39,6875	1,411111111
3/4“	19,05	1,27	1 5/8“	41,275	1,411111111
13/16“	20,6375	1,27	1 11/16“	42,8625	1,411111111
7/8“	22,225	1,27	1 3/4“	44,45	1,5875
15/16“	23,8125	1,27	2“	50,8	1,5875
1“	25,4	1,27			

Q = 16 NPT US taper pipe thread

Thread designation	Diameter (in mm)	Thread pitch	Thread designation	Diameter (in mm)	Thread pitch
1/16“	7,938	0,94074074	3 1/2“	101,6	3,175
1/8“	10,287	0,94074074	4“	114,3	3,175
1/4“	13,716	1,411111111	5“	141,3	3,175
3/8“	17,145	1,411111111	6“	168,275	3,175
1/2“	21,336	1,814285714	8“	219,075	3,175
3/4“	26,67	1,814285714	10“	273,05	3,175
1“	33,401	2,208695652	12“	323,85	3,175
1 1/4“	42,164	2,208695652	14“	355,6	3,175
1 1/2“	48,26	2,208695652	16“	406,4	3,175
2“	60,325	2,208695652	18“	457,2	3,175
2 1/2“	73,025	3,175	20“	508	3,175
3“	88,9	3,175	24“	609,6	3,175

Q = 17 NPTF US taper dryseal pipe thread

Thread designation	Diameter (in mm)	Thread pitch	Thread designation	Diameter (in mm)	Thread pitch
1/16“	7,938	0,94074074	1“	33,401	2,208695652
1/8“	10,287	0,94074074	1 1/4“	42,164	2,208695652
1/4“	13,716	1,411111111	1 1/2“	48,26	2,208695652
3/8“	17,145	1,411111111	2“	60,325	2,208695652
1/2“	21,336	1,814285714	2 1/2“	73,025	3,175
3/4“	26,67	1,814285714	3“	88,9	3,175

Q = 18 NPSC U.S. cylindrical pipe thread with lubricant

Thread designation	Diameter (in mm)	Thread pitch	Thread designation	Diameter (in mm)	Thread pitch
1/8“	10,287	0,94074074	1 1/2“	48,26	2,208695652
1/4“	13,716	1,411111111	2“	60,325	2,208695652
3/8“	17,145	1,411111111	2 1/2“	73,025	3,175
1/2“	21,336	1,814285714	3“	88,9	3,175
3/4“	26,67	1,814285714	3 1/2“	101,6	3,175
1“	33,401	2,208695652	4“	114,3	3,175
1 1/4“	42,164	2,208695652			

Q = 19 NPFS U.S. cylindrical pipe thread without lubricant

Thread designation	Diameter (in mm)	Thread pitch	Thread designation	Diameter (in mm)	Thread pitch
1/16“	7,938	0,94074074	1/2“	21,336	1,814285714
1/8“	10,287	0,94074074	3/4“	26,67	1,814285714
1/4“	13,716	1,411111111	1“	33,401	2,208695652
3/8“	17,145	1,411111111			

9.2 Undercut parameters

DIN 76—undercut parameters

The CNC PILOT determines the parameters for the thread undercut (undercut DIN 76) from the thread pitch. The undercut parameters are in accordance with DIN 13 for metric threads.

External thread					External thread				
Thread pitch	**I**	**K**	**R**	**W**	**Thread pitch**	**I**	**K**	**R**	**W**
0,2	0,3	0,7	0,1	30°	1,25	2	4,4	0,6	30°
0,25	0,4	0,9	0,12	30°	1,5	2,3	5,2	0,8	30°
0,3	0,5	1,05	0,16	30°	1,75	2,6	6,1	1	30°
0,35	0,6	1,2	0,16	30°	2	3	7	1	30°
0,4	0,7	1,4	0,2	30°	2,5	3,6	8,7	1,2	30°
0,45	0,7	1,6	0,2	30°	3	4,4	10,5	1,6	30°
0,5	0,8	1,75	0,2	30°	3,5	5	12	1,6	30°
0,6	1	2,1	0,4	30°	4	5,7	14	2	30°
0,7	1,1	2,45	0,4	30°	4,5	6,4	16	2	30°
0,75	1,2	2,6	0,4	30°	5	7	17,5	2,5	30°
0,8	1,3	2,8	0,4	30°	5,5	7,7	19	3,2	30°
1	1,6	3,5	0,6	30°	6	8,3	21	3,2	30°

Internal thread					Internal thread				
Thread pitch	**I**	**K**	**R**	**W**	**Thread pitch**	**I**	**K**	**R**	**W**
0,2	0,1	1,2	0,1	30°	1,25	0,5	6,7	0,6	30°
0,25	0,1	1,4	0,12	30°	1,5	0,5	7,8	0,8	30°
0,3	0,1	1,6	0,16	30°	1,75	0,5	9,1	1	30°
0,35	0,2	1,9	0,16	30°	2	0,5	10,3	1	30°
0,4	0,2	2,2	0,2	30°	2,5	0,5	13	1,2	30°
0,45	0,2	2,4	0,2	30°	3	0,5	15,2	1,6	30°
0,5	0,3	2,7	0,2	30°	3,5	0,5	17,7	1,6	30°
0,6	0,3	3,3	0,4	30°	4	0,5	20	2	30°
0,7	0,3	3,8	0,4	30°	4,5	0,5	23	2	30°
0,75	0,3	4	0,4	30°	5	0,5	26	2,5	30°
0,8	0,3	4,2	0,4	30°	5,5	0,5	28	3,2	30°
1	0,5	5,2	0,6	30°	6	0,5	30	3,2	30°

For internal threads. the CNC PILOT calculates the depth of the thread undercut according to the following formula:

Undercut depth = (N + I – K) / 2

Where:

- I: Undercut depth (radius)
- K: Undercut width
- R: Undercut radius
- W: Undercut angle
- N: Nominal diameter of the thread
- I: From the table
- K: Core diameter of the thread

DIN 509 E – undercut parameters

Diameter	I	K	R	W
<=1,6	0,1	0,5	0,1	15°
> 1,6 – 3	0,1	1	0,2	15°
> 3 – 10	0,2	2	0,2	15°
> 10 – 18	0,2	2	0,6	15°
> 18 – 80	0,3	2,5	0,6	15°
> 80	0,4	4	1	15°

The undercut parameters are determined from the cylinder diameter.

Where:

- I: Undercut depth
- K: Undercut width
- R: Undercut radius
- W: Undercut angle

DIN 509 F – undercut parameters

Diameter	I	K	R	W	P	A
<=1,6	0,1	0,5	0,1	15°	0,1	8°
> 1,6 – 3	0,1	1	0,2	15°	0,1	8°
> 3 – 10	0,2	2	0,2	15°	0,1	8°
> 10 – 18	0,2	2	0,6	15°	0,1	8°
> 18 – 80	0,3	2,5	0,6	15°	0,2	8°
> 80	0,4	4	1	15°	0,3	8°

The undercut parameters are determined from the cylinder diameter.

Where:

- I: Undercut depth
- K: Undercut width
- R: Undercut radius
- W: Undercut angle
- P: Face depth
- A: Transverse angle

9.3 Technical information

Specifications	
Components	■ MC 6441, MC6542 or MC 7420 main computer with ■ CC 61xx or UEC 11x controller unit ■ 15-inch or 19-inch TFT color flat-panel display ■ TE 735T or TE 745T keyboard unit
Operating system	■ HEROS real-time operating system for machine control
Memory	■ 1.8 GB (on CFR compact flash memory card) for NC programs
Input resolution and display step	■ X axis: 0.5 µm, diameter: 1 µm ■ Z and Y axes: 1 µm ■ U, V and W axes: 1 µm ■ C and B axes: 0.001°
Interpolation	■ Straight line: In 2 principal axes, optional in 3 principal axes (max. ±100 m) ■ Circle: in 2 axes (radius max. 999 m), optional additional linear interpolation of the third axis ■ C axis: Interpolation in the linear axes X and Z with the C axis
Feed rate	■ mm/min or mm/rev ■ Constant surface speed ■ Max. feed rate (60 000/pole pairs × ball screw pitch) at fPWM = 5000 Hz
Spindle	■ Maximum 60000 rpm (with 2 pole pairs)
Axis feedback control	■ Integrated digital drive control for synchronous and asynchronous motors ■ Position loop resolution: Signal period of the position encoder/1024 ■ Position control clock pulse: 3 ms ■ Speed control clock pulse: 0.2 ms ■ Current control: 0.1 ms
Error compensation	■ Linear and nonlinear axis error, backlash, reversal peaks during circular movements ■ Static friction
Data interfaces	■ 1000 BaseT Gigabit Ethernet interface ■ 4x USB 3.0 on the rear, 1x USB 2.0 on the front
Diagnosis	■ Fast and simple troubleshooting through integrated diagnostic aids
Surrounding temperature	■ Operation: 5 °C to 40 °C ■ Storage: –20 °C to +60 °C

User functions	
Configuration	■ Basic version: X and Z axis, spindle ■ Y axis (optional) ■ Positionable spindle and driven tool (optional) ■ C axis and driven tool (optional) ■ B axis (optional) ■ Digital current and speed control ■ Rear-face machining with the opposing spindle (optional)
Manual operation	■ Manual slide movement through axis-direction keys or electronic handwheel ■ Graphic support for entering and running Teach-in cycles without saving the machining steps in alternation with manual machine operation ■ Thread reworking (thread repair in a second workpiece setup) (optional)
Teach-in mode	■ Sequential linking of Teach-in cycles, where each machining cycle is run immediately after input, or is graphically simulated and subsequently saved.
Program Run mode	All are possible in single-block and full-sequence modes: ■ DINplus programs ■ smart.Turn programs (optional) ■ Teach-in programs (optional)
Setup functions	■ Workpiece datum setting ■ Definition of tool-change position ■ Definition of protection zone ■ Measurement of the tool by touch-off, touch probe, or optical gauge ■ Workpiece measurement with a TS workpiece touch probe

User functions	
Programming – Teach-in mode (optional)	■ Turning cycles for simple and complex contours, and contours described with ICP ■ Contour-parallel turning cycles ■ Recessing cycles for simple or complex contours, or contours defined with ICP ■ Repetitions with recessing cycles ■ Recess turning cycles for simple and complex contours, and contours described with ICP ■ Undercut and parting cycles (optional) ■ Engraving cycles ■ Threading cycles for single or multi-start longitudinal, taper or API threads ■ Cycles for axial and radial drilling, pecking and tapping operations with the C axis ■ Thread milling with the C axis ■ Axial and radial milling cycles for slots, figures, single surfaces and polygons as well as for complex contours defined with ICP for machining with the C axis ■ Helical slot milling with the C axis ■ Linear and circular patterns for drilling and milling operations with the C axis ■ Context-sensitive help graphics ■ Transfer of cutting values from technology database ■ Use of DIN macros in Teach-in programs ■ Conversion of Teach-in programs to smart.Turn programs
Interactive contour programming (ICP) (optional)	■ Contour definition with linear and circular contour elements ■ Immediate display of entered contour elements ■ Calculation of missing coordinates, intersections, etc. ■ Graphic display of all solutions for selection by the user if more than one solution is possible ■ Chamfers, rounding arcs and undercuts available as form elements ■ Input of form elements immediately during contour creation or by superimposition later ■ Changes to existing contours can be programmed ■ Programming of the rear face for full-surface machining with the C and Y axes
C-axis machining on face and lateral surface	■ Description of individual holes and hole patterns ■ Description of figures and figure patterns for milling operations ■ Creation of freely definable milling contours
Y-axis machining in the XY and ZY planes	■ Description of individual holes and hole patterns ■ Description of figures and figure patterns for milling operations ■ Creation of freely definable milling contours

User functions	
B-axis machining (optional)	■ Machining with the B axis ■ Tilting the working plane ■ Rotating the machining position of the tool
DXF import	■ Importing contours for turning ■ Importing contours for milling
smart.Turn programming (optional)	■ The basis is the unit, which is the complete description of a machining block (geometry, technology and cycle data) ■ Dialog boxes divided into overview and detail forms ■ Fast navigation between the fillable forms and input groups via the "smart" keys ■ Context-sensitive help graphics ■ Start unit with global settings ■ Transfer of global values from the start unit ■ Transfer of cutting values from technology database ■ Units for all lathe and recessing operations ■ Use of the contours described with ICP for turning and recess turning operations ■ Units for all milling and drilling operations with the C axis ■ Use of the patterns and contours described with ICP for C-axis operations ■ Units for activating/deactivating the C axis ■ Units for all milling and drilling operations with the Y axis ■ Use of the patterns and contours described with ICP for Y-axis operations ■ Special units for subprograms and program part repeats ■ Verification graphics for blank and finished part and for C and Y axis contours ■ Turret assignment and other setup information in the smart.Turn program ■ Parallel programming ■ Parallel simulation

User functions	
DINplus programming	■ Programming in DIN 66025 format ■ Extended command format (IF... THEN ... ELSE...) ■ Simple geometry programming (calculation of missing data) ■ Powerful fixed cycles for area clearance, recessing, recess turning and thread machining ■ Powerful fixed cycles for drilling and milling with the C axis (optional) ■ Powerful fixed cycles for drilling and milling with the Y axis (optional) ■ Subprograms ■ Programming with variables ■ Contour description with ICP (optional) ■ Program verification graphics for workpiece blank and finished part ■ Turret assignment and other setup information in the DINplus program ■ Conversion of smart.Turn units into DINplus command sequences (optional) ■ Parallel programming ■ Parallel simulation
Test run graphics	■ Graphic simulation of sequence of Teach-in cycle, Teach-in, smart.Turn or DINplus programs ■ Display of the tool paths as wire-frame or cutting-path graphics, special identification of the rapid-traverse paths ■ Machining simulation (2-D material-removal graphic) ■ Display of programmed contours ■ Side or face view, or 2-D view of cylindrical surface for verification of C axis machining ■ View of face (XY plane) and YZ plane for verification of Y-axis machining ■ Shifting and magnifying functions ■ 3-D graphics for display of workpiece blank and finished part as solid-model view
Machining time analysis	■ Calculation of machining times and idle times ■ Consideration of switching commands triggered by the CNC ■ Representation of time per individual cycle or per tool change
TURN PLUS	■ Automatic generation of smart.Turn programs ■ Automatic cutting limitation through the definition of chucking equipment ■ Automatic tool selection and turret assignment

User functions	
Tool database	■ For 250 tools ■ For 999 tools (optional) ■ Tool description can be entered for every tool ■ Automatic inspection of tool-tip position with respect to the contour ■ Compensation of tool-tip position in the X/Y/Z plane ■ High-precision correction via handwheel, capturing compensation values in the tool table ■ Automatic tool-tip and cutter radius compensation ■ Tool monitoring for lifetime of the insert (tool tip) or the number of workpieces produced ■ Tool monitoring with automatic tool change after tool insert wear (optional) ■ Management of multipoint tools (multiple inserts or reference points)
Technology database (optional)	■ Access to cutting data after definition of workpiece material, cutting material and machining mode. The CNC PILOT distinguishes between 16 machining modes. Each workpiece-material/tool-material combination includes the cutting speed, the main and secondary feed rates, and the infeed for 16 machining modes. ■ Automatic determination of the machining modes from the cycle or the machining unit ■ The cutting data are entered in the cycle or in the unit as default values. ■ 9 workpiece-material/tool-material combinations (144 entries) ■ 62 workpiece-material/tool-material combinations (992 entries) (optional)
Conversational languages	■ ENGLISH ■ GERMAN ■ CZECH ■ FRENCH ■ ITALIAN ■ SPANISH ■ PORTUGUESE ■ SWEDISH ■ DANISH ■ FINNISH ■ DUTCH ■ POLISH ■ HUNGARIAN ■ RUSSIAN ■ CHINESE ■ CHINESE_TRAD Further languages available as options (see option number 41).

Accessories	
Electronic handwheels	■ HR 180 panel-mounted handwheels with connection to position inputs, plus ■ One HR 130 panel-mounted serial handwheel, or one portable serial handwheel HR 410
Touch probe	■ **TS 230**: 3-D touch trigger probe with cable connection, or ■ **TS 440**: 3-D touch trigger probe with infrared transmission ■ **TS 444**: Battery-free 3-D touch trigger probe with infrared transmission ■ **TS 640**: 3-D touch trigger probe with infrared transmission ■ **TS 740**: High-precision 3-D touch trigger probe with infrared transmission ■ **TT 140**: 3-D touch trigger probe for tool measurement with cable connection ■ **TT 449**: 3-D touch trigger probe for tool measurement with infrared transmission
DataPilot CP 640, MP 620	Control software for PCs for programming, archiving, and training for the CNC PILOT: ■ Full version with license for single station or multiple stations ■ Demo version (free of charge)

Option number	Option	ID	Description
0 to 6	Additional axis	354540-01 353904-01 353905-01 367867-01 367868-01 370291-01 307292-01	**Additional control loops**
8	Software option 1	632226-01	**Cycle programming** ■ Contour description with ICP ■ Cycle programming ■ Technology database with 9 workpiece-material/tool-material combinations
9	Software option 2	632227-01	**smart.Turn** ■ Contour description with ICP ■ Programming with smart.Turn ■ Technology database with 9 workpiece-material/tool-material combinations
10	Software option 3	632228-01	**Tools and technology** ■ Tool database expanded to 999 entries ■ Technology database expanded to 62 workpiece-material/tool-material combinations ■ Tool life monitoring with exchange tools
11	Software option 4	632229-01	**Threads** ■ Thread recutting ■ Handwheel superimposition during thread cutting
17	Software option TCH PROBE functions	632230-01	**Tool measurement and workpiece measurement** ■ Determining tool-setting dimensions with a touch probe ■ Determining tool-setting dimensions with an optical gauge ■ Automatic workpiece measurement
41	Additional language	530184-01 530184-02 530184-03 530184-04 530184-06 530184-07 530184-08 530184-09 530184-10	Slovenian Slovak Latvian Norwegian Korean Estonian Turkish Romanian Lithuanian

Option number	Option	ID	Description
42	Software option DXF import	632231-01	**DXF import** ■ Loading of DXF contours
54	B-axis machining	825742-01	**Machining with the B axis** ■ Tilting the working plane ■ Rotating the machining position of the tool
55	Software option C-axis machining	633944-01	**C-axis machining**
63	TURN PLUS	825743-01	**Automatic generation of smart.Turn programs**
70	Y-axis machining	661881-01	**Y-axis machining**
94	W-axis machining	661881-01	**Support of parallel axes (U, V, W)**
131	Spindle synchronism	806270-01	**Synchronization** (of two or more spindles)
132	Opposing spindle	806275-01	**Opposing spindle** (spindle synchronism, rear-face machining)
143	Load Adaptive Control (LAC)	800545-01	**LAC**: Dynamic adaptation of the control parameters

9.4 Compatibility in DIN programs

The format of DIN programs of the CNC PILOT 4290 predecessor control differs from the format of the CNC PILOT 640. However, you can use the program converter to adapt programs of the predecessor control to the new control.

When opening an NC program, the CNC PILOT 640 recognizes the programs of the predecessor control. The program concerned will be converted after a confirmation prompt. "CONV_..." will be prefixed to the program name. The program converter is also part of the Transfer function (Organization mode of operation).

When converting DIN programs, it is also important to keep in mind that the predecessor controls use different solutions for tool management, parameter management, programming with variables and PLC programming.

Remember the following when converting DIN programs of the CNC PILOT 4290:

Tool call (T commands of the TURRET section):

- T commands containing a reference to the tool database are left unchanged (example: T1 ID“342-300.1“)
- T commands containing tool data cannot be converted

Programming variables

- D variables (# variables) are replaced by # variables of the new syntax. Depending on the range of numbers, #c or #l or #n or #i variables will be used.
- Special conversion rules: #0 is converted to #c30, #30 is converted to #c51
- V variables are replaced by #g variables. Braces used in assignments are removed. Braces used in expressions are changed to parentheses.
- Variable accesses to tool data, machine dimensions, D compensation values, parameter data and events cannot be converted. These program sequences have to be adapted. Exception: The "Mid-program startup active" E90[1] event is converted to #i6
- Remember that – unlike the 4290 – the interpreter of the CNC PILOT 640 re-evaluates the lines every time the program is executed.

M functions:

- M30 with NS.. is converted to M0 M99 NS
- M97 is removed for single-channel controls
- All other M functions are left unchanged.

G functions:

- The following G functions are currently not supported by the CNC PILOT 640: G48, G53, G54, G55, G62, G63, G98, G162, G204, G710, G906, G907, G915, G918, G975, G995, G996
- A warning will be issued if the following G functions are used in a contour description: G10, G38, G39, G52, G95, G149. These functions are now modal.

- Warnings may occur for the thread functions G31, G32, G33; it is recommended to test these functions.
- The "Contour mirroring/shifting G121" function is converted to G99, but the principle of function is compatible.
- A warning will be issued for G916, G917 and G930 because the principle of function has changed. Functions must be supported by the PLC.

Names of external subprograms:

- When an external subprogram is called, the converter prefixes "CONV_..." to the name.

Multi-channel programs:

- On single-channel controls, programs for two slides are converted to a single slide and the Z-axis movements of the second slide are converted to G1 W… or G701 W…
  - In the program head, #SLIDE $1$2 is replaced by #SLIDE $1
  - $ statements preceding the block number are removed
  - $2 G1 Z… is converted to G1 W… and G701 Z… to G701 W…
  - The word ZUORDNUNG [ASSIGNMENT] is removed (but remembered internally for converting the subsequent blocks)
  - Synchronization statements $1$2 M97 are removed
  - Zero point shifts for slide 2 are saved as comments; paths of traverse are provided with a warning.

Nonconvertible elements:

- If the DIN program contains nonconvertible elements, the corresponding NC block is saved as a comment. The word WARNING is inserted in front of this comment. Depending on the situation, the nonconvertible command is taken into the comment line, or the nonconvertible NC block follows the comment.



HEIDENHAIN recommends adapting converted NC programs to the circumstances of the control and then testing them before using them for production.



Syntax elements of the CNC PILOT 640

Meaning of the symbols used in the table:

- ✔ Compatible behavior; if necessary, the program converter converts the functions so that they are compatible with the CNC PILOT 640
- X Behavior changed; the program has to be tested in the individual case
- – Function is not available or is replaced by different functionality
- ◆ Function is planned for future software versions or will only be required for multi-channel systems

Section codes		
Program head	PROGRAMMKOPF [HEADER]	✔
	REVOLVER [TURRET]	✔
	SCHEIBENMAGAZIN [PLATE MAGZN.]	✔
	CHUCKING EQUIPMENT	X
Contour definition	KONTUR [CONTOUR]	◆
	ROHTEIL [BLANK]	✔
	FERTIGTEIL [FINISHED]	✔
	HILFSKONTUR [AUXIL_CONTOUR]	✔
C-axis contours	STIRN [FACE_C]	✔
	RUECKSEITE [REAR_C]	✔
	MANTEL [LATERAL_C]	✔
Workpiece machining	BEARBEITUNG [MACHINING]	✔
	ZUORDNUNG [ASSIGNMENT]	◆
	ENDE [END]	✔
Subprograms	UNTERPROGRAMM [SUBPROGRAM]	✔
	RETURN	✔
Other	CONST	✔
Y-axis contours	STIRN_Y [FACE_Y]	✔
	RUECKSEITE_Y [REAR_Y]	✔
	MANTEL_Y [LATERAL_Y]	✔

G commands for turning contours		
Workpiece-blank definition	**G20-Geo** Chuck part, cylinder/tube	✓
	G21-Geo Cast part	✓
Basic contour elements	**G0-Geo** Starting point of contour	✓
	G1-Geo Line segment	✓
	G2-Geo Circular arc with incremental center dimensioning	✓
	G3-Geo Circular arc with incremental center dimensioning	✓
	G12-Geo Circular arc with absolute center dimensioning	✓
	G13-Geo Circular arc with absolute center dimensioning	✓
Contour form elements	**G22-Geo** Recess (standard)	✓
	G23-Geo Recess/relief turn	✓
	G24-Geo Thread with undercut	✓
	G25-Geo Undercut contour	✓
	G34-Geo Thread (standard)	✓
	G37-Geo Thread (general)	✓
	G49-Geo Bore hole at turning center	✓
Help commands for contour definition	**G7-Geo** Precision stop on	✓
	G8-Geo Precision stop off	✓
	G9-Geo Precision stop blockwise	✓
	G10-Geo Peak-to-valley height	X
	G38-Geo Feed rate reduction	X
	G39-Geo Attributes for superimposed elements	–
	G52-Geo Blockwise oversize	X
	G95-Geo Feed per revolution	X
	G149-Geo Additive compensation	X

G commands for C-axis contours		
Overlapping contours	**G308-Geo** Start of pocket/island	✓
	G309-Geo End of pocket/island	✓
Front and rear face contours	**G100-Geo** Starting point of face contour	✓
	G101-Geo Line segment on front face	✓
	G102-Geo Circular arc on front face	✓
	G103-Geo Circular arc on front face	✓
	G300-Geo Bore hole on front face	✓
	G301-Geo Linear slot on front face	✓
	G302-Geo Circular slot on front face	✓
	G303-Geo Circular slot on front face	✓
	G304-Geo Full circle on front face	✓
	G305-Geo Rectangle on front face	✓
	G307-Geo Eccentric polygon on front face	✓
	G401-Geo Linear pattern on front face	✓
	G402-Geo Circular pattern on front face	✓
Lateral surface contours	**G110-Geo** Starting point of lateral surface contour	✓
	G111-Geo Line segment on lateral surface	✓
	G112-Geo Circular arc on lateral surface	✓
	G113-Geo Circular arc on lateral surface	✓
	G310-Geo Bore hole on lateral surface	✓
	G311-Geo Linear slot on lateral surface	✓
	G312-Geo Circular slot on lateral surface	✓
	G313-Geo Circular slot on lateral surface	✓
	G314-Geo Full circle on lateral surface	✓
	G315-Geo Rectangle on lateral surface	✓
	G317-Geo Eccentric polygon on lateral surface	✓
	G411-Geo Linear pattern on lateral surface	✓
	G412-Geo Circular pattern on lateral surface	✓

G commands for Y-axis contours		
XY plane	**G170-Geo** Starting point of contour	✓
	G171-Geo Line segment	✓
	G172-Geo Circular arc	✓
	G173-Geo Circular arc	✓
	G370-Geo Hole	✓
	G371-Geo Linear slot	✓
	G372-Geo Circular slot	✓
	G373-Geo Circular slot	✓
	G374-Geo Full circle	✓
	G375-Geo Rectangle	✓
	G376-Geo Single surface	✓
	G377-Geo Eccentric polygon	✓
	G471-Geo Linear pattern	✓
	G472-Geo Circular pattern	✓
	G477-Geo Centric polygon	✓

G commands for Y-axis contours		
YZ plane	**G180-Geo** Starting point of contour	✓
	G181-Geo Line segment	✓
	G182-Geo Circular arc	✓
	G183-Geo Circular arc	✓
	G380-Geo Hole	✓
	G381-Geo Linear slot	✓
	G382-Geo Circular slot	✓
	G383-Geo Circular slot	✓
	G384-Geo Full circle	✓
	G385-Geo Rectangle	✓
	G387-Geo Eccentric polygon	✓
	G481-Geo Linear pattern	✓
	G482-Geo Circular pattern	✓
	G386-Geo Single surface	✓
	G487-Geo Centric polygon	✓

G commands for machining		
Tool positioning without machining	**G0** Positioning at rapid traverse	✓
	G14 Move to the tool change position	✓
	G701 Rapid traverse to machine coordinates	✓
Simple linear and circular movements	**G1** Linear movement	✓
	G2 Circular movement with incremental center dimensioning	✓
	G3 Circular movement with incremental center dimensioning	✓
	G12 Circular movement with absolute center dimensioning	✓
	G13 Circular movement with absolute center dimensioning	✓

G commands for machining		
Feed rate and spindle speed	**Gx26** Speed limitation	✓
	G48 Acceleration (slope)	◆
	G64 Interrupted feed	✓
	G192 Feed per minute for rotary axis	–
	Gx93 Feed per tooth	✓
	G94 Feed per minute	✓
	Gx95 Feed per revolution	✓
	Gx96 Constant surface speed	✓
	Gx97 Speed	✓
Tool-tip radius compensation	**G40** Switch off TRC/MCRC	✓
	G41 TRC/MCRC, left	✓
	G42 TRC/MCRC, right	✓
Zero point shifts	**G51** Relative zero point shift	✓
	G53 Parameter-dependent zero point shift	◆
	G54 Parameter-dependent zero point shift	◆
	G55 Parameter-dependent zero point shift	◆
	G56 Additive zero point shift	✓
	G59 Absolute zero point shift	✓
	G121 Contour mirroring/shifting	✓
	G152 Zero point shift, C axis	✓
	G920 Deactivate zero point shift	✓
	G921 Deactivate zero point shift, tool dimensions	✓
	G980 Activate zero point shift	✓
	G981 Activate zero point shift, tool dimensions	✓
Oversizes	**G50** Switch off oversize	✓
	G52 Switch off oversize	✓
	G57 Axis-parallel oversize	✓
	G58 Contour-parallel oversize	✓

G commands for machining		
Safety clearances	**G47** Set safety clearances	✓
	G147 Safety clearance (milling cycles)	✓
Tools, types of compensation	**T** Tool call	✓
	G148 Switching the tool edge compensation	✓
	G149 Additive compensation	✓
	G150 Compensation of right-hand tool tip	✓
	G151 Compensation of left-hand tool tip	✓
	G710 Adding tool dimensions	◆

Cycles for turning		
Simple turning cycles	**G80** End of cycle	✓
	G81 Simple longitudinal roughing	✓
	G82 Simple face roughing	✓
	G83 Simple contour repeat cycle	✓
	G85 Undercut	✓
	G86 Simple recessing cycle	✓
	G87 Transition radii	✓
	G88 Chamfers	✓
Drilling cycles	**G36** Tapping	✓
	G71 Simple drilling cycle	✓
	G72 Counterboring, countersinking, etc.	✓
	G73 Tapping cycle	✓
	G74 Deep hole drilling cycle	✓

Cycles for turning		
Contour-based turning cycles	**G810** Longitudinal roughing cycle	✓
	G820 Face roughing cycle	✓
	G830 Contour-parallel roughing cycle	✓
	G835 Contour-parallel with neutral tool	✓
	G860 Universal recessing cycle	✓
	G866 Simple recessing cycle	✓
	G869 Recess turning cycle	✓
	G890 Finishing cycle	✓
Thread cycles	**G31** Thread cycle	✓
	G32 Single thread cycle	✓
	G33 Single thread cut (Thread single path)	✓
	G933 Thread switch	–
	G799 Thread milling, axial	✓
	G800 Thread milling in XY plane	✓
	G806 Thread milling in YZ plane	✓

Synchronization commands		
Assigning contour to operation	**G98** Assignment of spindle to workpiece	–
	G99 Workpiece group	◆
Slide synchronization	**G62** One-sided synchronization	◆
	G63 Synchronous start of slides	◆
	G162 Synchronization marking	◆
Contour follow-up	**G702** Storing/loading contour follow-up	✓
	G703 Contour follow-up ON/OFF	✓
	G706 K default branch	–

Synchronization commands		
Spindle synchronization, workpiece transfer	**G30** Converting and mirroring	✓
	G121 Contour mirroring/shifting	✓
	G720 Spindle synchronization	✓
	G905 Measuring C-angle offset	✓
	G906 Measuring angular offset during spindle synchronization	–
	G916 Traversing to a fixed stop	✓
	G917 Controlled parting using lag error monitoring	✓
	G991 Controlled parting using spindle monitoring	–
	G992 Values for controlled parting	–

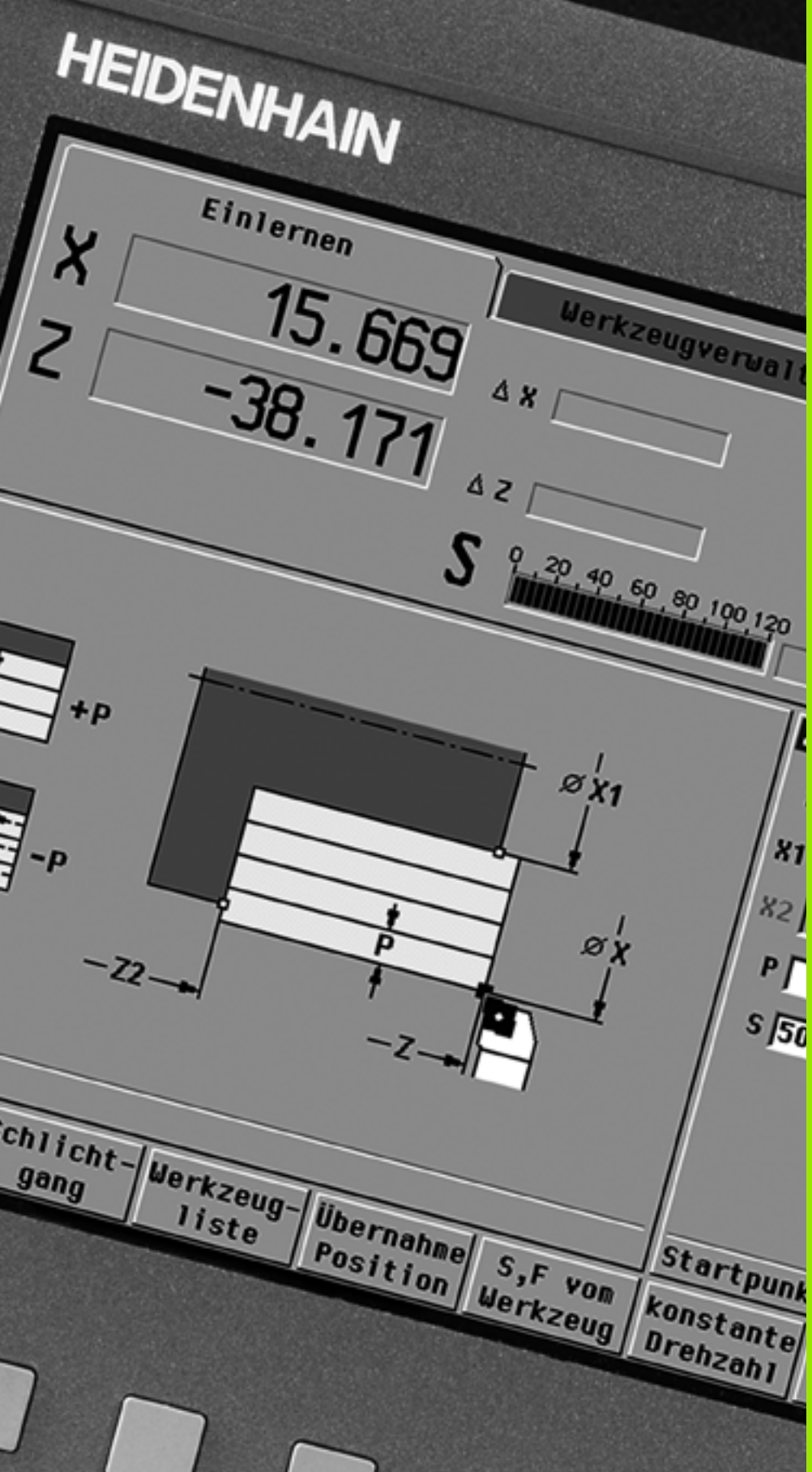
C-axis machining		
C axis	**G119** No. of C axis	✓
	G120 Reference diameter for lateral-surface machining	✓
	G152 Zero point shift, C axis	✓
	G153 Standardize C axis	✓
Front/rear-face machining	**G100** Rapid traverse, front face	✓
	G101 Synchronous start of slides	✓
	G102 Circular arc on front face	✓
	G103 Circular arc on front face	✓
Milling cycles	**G799** Thread milling, axial	✓
	G801 Engraving on front face	✓
	G802 Engraving on lateral surface	✓
	G840 Contour milling	✓
	G845 Pocket milling, roughing	✓
	G846 Pocket milling, finishing	✓
Lateral-surface machining	**G110** Rapid traverse, lateral surface	✓
	G111 Linear path on lateral surface	✓
	G112 Circular arc on lateral surface	✓
	G113 Circular arc on lateral surface	✓

Variable programming, program branches		
Programming with variables	**# variables** Evaluation during program conversion	✓
	V variables Evaluation during program run	✓
Program branches, program repeats	**IF..THEN..** Program branching	✓
	WHILE.. Program repeat	✓
	SWITCH.. Program branching	✓
Special functions	**$** Slide code	✓
	/ Skip level	◆
Data input and data output	**INPUT** Input (# variables)	✓
	WINDOW Open output window (# variables)	✓
	PRINT Output (# variables)	✓
	INPUTA Input (V variables)	✓
	WINDOWA Open output window (V variables)	✓
	PRINTA Output (V variables)	✓
Subprograms	**L** Subprogram call	✓

Measuring functions, load monitoring		
In-process measuring	**G910** Activate in-process measuring	✓
	G912 Actual-value determination for in-process measuring	✓
	G913 Deactivate in-process measuring	✓
	G914 Deactivate touch probe monitoring	✓
Post-process measuring	**G915** Post-process measuring	◆
Load monitoring	**G995** Specifying the monitoring zone	◆
	G996 Type of load monitoring	◆

Other G functions		
Other G functions	**G4** Dwell Time	✓
	G7 Precision stop on	✓
	G8 Precision stop off	✓
	G9 Precision stop (blockwise)	✓
	G15 Move rotary axes	–
	G60 Deactivate protection zone	✓
	G65 Display chucking equipment	✓
	G66 Component position	◆
	G204 Waiting for time	◆
	G717 Updating nominal values	–
	G718 Move lag error	–
	G901 Actual values in variables	✓
	G902 Zero point shift in variables	✓
	G903 Lag error in variables	✓
	G907 Zero point shift in variables	◆
	G908 Zero point shift in variables	✓
	G909 Zero point shift in variables	✓
	G918 Zero point shift in variables	–
	G919 Zero point shift in variables	✓
	G920 Zero point shift in variables	✓
	G921 Zero point shift in variables	✓
	G930 Zero point shift in variables	✓
	G975 Zero point shift in variables	◆
	G980 Zero point shift in variables	✓
	G981 Zero point shift in variables	✓
	G940 Zero point shift in variables	–
	G941 Zero point shift in variables	–

B-axis and Y-axis machining		
Working planes	**G16** Tilting working plane	✓
	G17 XY plane (front or rear face)	✓
	G18 XZ plane (turning)	✓
	G19 YZ plane (plan view / lateral surface)	✓
Tool positioning without machining	**G0** Positioning at rapid traverse	✓
	G14 Move to the tool change position	✓
	G600 Preselect tool	✓
	G701 Rapid traverse to machine coordinates	✓
	G714 Insert magazine tool	✓
	G712 Define tool position	✓
Milling cycles	**G841** Area milling, roughing	✓
	G842 Area milling, finishing	✓
	G843 Centric polygon milling, roughing	✓
	G844 Centric polygon milling, finishing	✓
	G845 Pocket milling, roughing	✓
	G846 Pocket milling, finishing	✓
	G800 Thread milling in XY plane	✓
	G806 Thread milling in YZ plane	✓
	G803 Engraving in XY plane	✓
	G804 Engraving in YZ plane	✓
	G808 Hobbing	✓
Simple linear and circular movements	**G1** Linear path	✓
	G2 Circular arc with incremental center dimensioning	✓
	G3 Circular arc with incremental center dimensioning	✓
	G12 Circular arc with absolute center dimensioning	✓
	G13 Circular arc with absolute center dimensioning	✓



10

Overview of cycles

10.1 Workpiece blank cycles, single cut cycles

Workpiece blank cycles		Page
	**Overview**	129
	**Standard blank**	130
	**ICP blank**	131

Single cut cycles		Page
	**Overview**	132
	**Rapid traverse positioning**	133
	**Move to the tool change position**	134
	**Linear machining, longitudinal** Single longitudinal cut	135
	**Linear machining, transverse** Single transverse cut	136
	**Linear machining at angle** Single oblique cut	137
	**Circular machining** Single circular cut	139
	**Circular machining** Single circular cut	139
	**Chamfer** For machining a chamfer	141
	**Rounding arc** For machining a rounding arc	143
	**M function** For entering an M function	145

10.2 Turning cycles

Turning cycles		Page
	**Overview**	146
	**Cut longitudinal** Roughing and finishing cycle for simple contours	149
	**Cut transverse** Roughing and finishing cycle for simple contours	151
	**Cut with longitudinal plunge Roughing and finishing cycle for simple contours**	163
	**Plunge, transverse** Roughing and finishing cycle for simple contours	165
	**ICP contour-parallel, longitudinal** Roughing and finishing cycle for any type of contour	178
	**ICP contour-parallel, transverse** Roughing and finishing cycle for any type of contour	180
	**ICP cut longitudinal** Roughing and finishing cycle for any type of contour	186
	**ICP cut transverse** Roughing and finishing cycle for any type of contour	188

10.3 Recessing and recess-turning cycles

Recessing cycles		Page
	**Overview**	198
	**Recessing, radial** Recessing and finishing cycles for simple contours	200
	**Recessing, axial** Recessing and finishing cycles for simple contours	202
	**ICP recessing, radial** Recessing and finishing cycles for any contour	216
	**ICP recessing, axial** Recessing and finishing cycles for any contour	218
	**Undercut H**	248
	**Undercut K**	250
	**Undercut U**	251
	**Parting** Cycle for parting the workpiece	253

Recess-turning cycles		Page
 	**Overview**	224
	**Recess turning, radial** Recess-turning and finishing cycles for simple contours	225
	**Recess-turning, axial** Recess-turning and finishing cycles for simple contours	226
	**ICP recess turning, radial** Recess-turning and finishing cycles for any type of contour	240
	**ICP recess turning, axial** Recess-turning and finishing cycles for any type of contour	242

10.4 Thread cycles

Thread cycles		Page
	**Overview**	257
	**Thread cycle** Longitudinal single or multi-start thread	261
	**Tapered thread** Tapered single or multi-start thread	265
	**API thread** Single or multi-start API thread (API: American Petroleum Institute)	267
	**Thread recutting** Recut longitudinal single or multi-start thread	269
	**Tapered thread, recutting** Recut tapered single or multi-start thread	273
	**API thread, recutting** Recut single or multi-start API thread	275
	**Undercut DIN 76** Thread undercut and thread chamfer	277
	**Undercut DIN 509 E** Undercut and cylinder chamfer	279
	**Undercut DIN 509 F** Undercut and cylinder chamfer	281

10.5 Drilling cycles

Drilling cycles		Page
	**Overview**	285
	**Axial drilling cycle** For drilling single holes and patterns	286
	**Radial drilling cycle** For drilling single holes and patterns	288
	**Axial deep-hole drilling cycle** For drilling single holes and patterns	290
	**Radial deep-hole drilling cycle** For drilling single holes and patterns	293
	**Axial tapping cycle** For drilling single holes and patterns	295
	**Radial tapping cycle** For drilling single holes and patterns	297
	**Thread milling** For milling threads in existing holes	299

10.6 Milling cycles

Milling cycles		Page
	**Overview**	303
	**Rapid traverse positioning** Activate C axis; position tool and spindle	304
	**Slot, axial** For milling single slots or slot patterns	305
	**Figure, axial** For milling a single figure	307
	**Axial ICP contour** For milling single ICP contours or contour patterns	311
	**Face milling** For milling surfaces or polygons	314
	**Slot, radial** For milling single slots or slot patterns	317
	**Figure, radial** For milling a single figure	319
	**Radial ICP contour** For milling single ICP contours or contour patterns	323
	**Helical-slot milling, radial** For milling a helical slot	326
	**Thread milling** For milling threads in existing holes	299

A
Absolute coordinates ... 44
Additive compensation ... 112
Additive compensation for cycle programming ... 128
Alphanumeric keyboard ... 55
API thread ... 267
API thread, recutting ... 275
Axis designations ... 43
Axis values, setting ... 91, 92

B
Backup ... 546
Backup name ... 557
Bar/tube blank ... 130
Basic-block mode
  Display during program execution ... 110
Block number
  Cycle programming ... 106

C
C axis, fundamentals ... 35
Calculator ... 56
C-axis values, setting ... 94
Chamfer ... 141
Character set ... 335
Circular drilling pattern, axial ... 341
Circular drilling pattern, radial ... 349
Circular machining ... 139
Circular milling pattern, axial ... 343
CNC PILOT 620 Ethernet interface
CNC PILOT 640 Ethernet interface
Comments
  Comment blocks in cycle programs ... 125
Comments in cycles ... 125
Compatibility in DIN programs ... 591
Compensation, additive ... 112
Compensations ... 111
Context-sensitive help ... 64
Continuous run
  Program execution ... 110
Contour follow-up in Teach-in mode ... 124
Conversion into DIN format ... 119
Coordinate system ... 44
Coordinates, absolute ... 44
Coordinates, incremental ... 45
Coordinates, polar ... 45

C
Copy
  Circular ... 376
  Linear ... 375
  Mirror ... 376
Cut, ICP contour-parallel, longitudinal ... 178
Cut, ICP contour-parallel, longitudinal finishing ... 182
Cut, ICP contour-parallel, transverse ... 180
Cut, ICP contour-parallel, transverse finishing ... 184
Cut, ICP longitudinal ... 186
Cut, ICP longitudinal finishing ... 190
Cut, ICP transverse ... 188
Cut, ICP transverse finishing ... 192
Cut, longitudinal ... 149
Cut, longitudinal finishing ... 157
Cut, longitudinal finishing plunge ... 171
Cut, longitudinal finishing plunge—expanded ... 174
Cut, longitudinal finishing—expanded ... 159
Cut, longitudinal plunge ... 163
Cut, longitudinal plunging—expanded ... 167
Cut, longitudinal—expanded ... 153
Cut, transverse ... 151
Cut, transverse finishing ... 158
Cut, transverse finishing plunge ... 172
Cut, transverse finishing plunge—expanded ... 176
Cut, transverse finishing—expanded ... 161
Cut, transverse plunge ... 165
Cut, transverse plunging—expanded ... 169
Cut, transverse—expanded ... 155
Cutting direction (cycle programming) ... 328, 329
Cutting limits SX, SZ ... 128
Cycle keys ... 124
Cycle menu ... 126
Cycle programming
  Cycle keys ... 124
Cycle programs, converting ... 565
Cycle starting point ... 122
Cycle status ... 81
Cycles in Manual mode ... 105
Cycles, addresses used ... 128

D
Data backup ... 40
Data entry—fundamentals ... 54
Data transfer ... 546
DATAPILOT 546
Deep-hole drilling, axial ... 290
Deep-hole drilling, radial ... 293
DIN cycle ... 356
DIN cycle (cycle programming) ... 356
DIN macros ... 123
DIN programs, converting ... 566
Direction of rotation (tool parameters) ... 498
Display operating times ... 97
Drilling cycles, cycle programming ... 285
Drilling pattern linear, axial ... 337
Drilling, axial ... 286
Drilling, radial ... 288
Drilling/milling patterns ... 336
Driven tool ... 499
Driven tools ... 87
Dry run mode ... 114
DXF contours ... 465

E
End point of ICP contour ... 368
Engraving, character set ... 335
Engraving, face ... 331
Engraving, lateral surface ... 333
Equidistant line (MCRC) ... 48
Equidistant line (TRC) ... 48
Error log file ... 62
Error messages ... 60
Ethernet ... 547
Ethernet interface ... 547
  Configuring ... 550
  Connection possibilities ... 549
  Introduction ... 549
Example of drilling cycles ... 301
Example of milling cycle ... 330
Example of pattern machining ... 353
Example of thread and undercut cycles ... 283
Examples of recessing cycles ... 255
Examples of turning cycles ... 194
External access ... 546

F

Feed angle ... 258
Feed rate ... 82
Feed rate reduction for drilling
  Cycle programming
    Deep-hole drilling ... 291, 294
    Drilling cycle ... 287, 289
File organization ... 116
Fit calculation ... 370
Fits ... 370
Form elements (ICP)
  Fundamentals ... 361
Form elements, ICP ... 361
Full-surface machining
  Fundamentals ... 37

G

Geometry calculations (ICP) ... 362

H

Handwheel operation ... 104
Handwheel resolution ... 120
Help files, downloading ... 69
Help graphics ... 123
Help system ... 64

I

ICP absolute or incremental dimensioning ... 369
ICP angle input ... 371
ICP axial, recess turning, finishing ... 246
ICP basic elements, turning contour ... 387
ICP centric polygon in XY plane ... 447
ICP centric polygons in YZ plane ... 463
ICP chamfer in XY plane ... 437
ICP chamfer in YZ plane ... 454
ICP chamfer of turning contour ... 391
ICP chamfer on face ... 403
ICP chamfer on lateral surface ... 409
ICP circle in XY plane ... 438
ICP circle in YZ plane ... 455
ICP circle on face ... 414
ICP circular arc in XY plane ... 436
ICP circular arc in YZ plane ... 453, 454
ICP circular arc on face ... 402
ICP circular arc on lateral surface ... 408
ICP circular arc, turning contour ... 390
ICP circular on lateral surface ... 423
ICP circular pattern in XY plane ... 445

I

ICP circular pattern in YZ plane ... 461
ICP circular pattern on face ... 420
ICP circular pattern on lateral surface ... 430
ICP circular slot in XY plane ... 442
ICP circular slot in YZ plane ... 458
ICP circular slot on face ... 417
ICP circular slot on lateral surface ... 427
ICP contour direction ... 377
ICP contour element, deleting ... 379
ICP contour elements
  Face ... 398, 413
ICP contour elements on the lateral surface ... 404
ICP contour elements, adding ... 378
ICP contour elements, editing ... 380
ICP contour elements, face ... 398
ICP contour elements, turning contour ... 387
ICP contour form elements ... 391
ICP contour graphics ... 372
ICP contour, creating ... 368
ICP contours with C-axis machining ... 410
ICP contours with Y-axis machining ... 410
ICP contours, editing ... 368, 378
ICP contours, fundamentals ... 360
ICP editor in cycle mode ... 363
ICP editor in smart.Turn ... 365
ICP face contours in smart.Turn ... 413
ICP form elements, superimposing ... 378
ICP geometry calculations ... 362
ICP hole in XY plane ... 443
ICP hole in YZ plane ... 459
ICP hole on face ... 418
ICP hole on lateral surface ... 428
ICP horizontal lines in XY plane ... 434
ICP horizontal lines in YZ plane ... 451
ICP horizontal lines of a turning contour ... 388
ICP horizontal lines on face ... 400
ICP horizontal lines on lateral surface ... 406
ICP last contour element, editing or deleting ... 379
ICP lateral surface contours in smart.Turn ... 421

I

ICP line at angle in XY plane ... 435
ICP line at angle in YZ plane ... 452
ICP line at angle on face ... 401
ICP line at angle on lateral surface ... 407
ICP line at angle on turning contour ... 389
ICP linear pattern in XY plane ... 444
ICP linear pattern in YZ plane ... 460
ICP linear pattern on face ... 419
ICP linear pattern on lateral surface ... 429
ICP linear slot in XY plane ... 441
ICP linear slot in YZ plane ... 458
ICP linear slot on face ... 417
ICP linear slot on lateral surface ... 426
ICP machining attributes ... 361
ICP nested contours and holes ... 411
ICP polar coordinates ... 371
ICP polygon in XY plane ... 440
ICP polygon in YZ plane ... 457
ICP polygon on face ... 416
ICP polygon on lateral surface ... 425
ICP programming
  Absolute or incremental dimensioning ... 369
  Contour direction ... 377
  Contour elements on face ... 398, 413
ICP recess turning, axial ... 242
ICP recess turning, radial ... 240
ICP recess turning, radial finishing ... 244
ICP recessing axial, finishing ... 222
ICP recessing cycles, axial ... 218
ICP recessing radial ... 216
ICP recessing radial, finishing ... 220
ICP rectangle in XY plane ... 439
ICP rectangle in YZ plane ... 456
ICP rectangle on face ... 415
ICP rectangle on lateral surface ... 424
ICP reference data in XY plane ... 432
ICP reference data in YZ plane ... 448
ICP reference data: ... 411
ICP rounding arc in XY plane ... 437
ICP rounding arc of turning contour ... 391

I

ICP rounding arc on face ... 403
ICP rounding arc on lateral surface ... 409
ICP selection functions ... 374
ICP selection of solutions ... 373
ICP single hole in XY plane ... 446
ICP single surface in YZ plane ... 462
ICP starting point of contour in XY plane ... 433
ICP starting point of contour in YZ plane ... 450
ICP starting point of face contour ... 398
ICP starting point of lateral surface contour ... 404
ICP starting point of turning contour ... 387
ICP transitions between contour elements ... 369
ICP undercut DIN 509 E ... 393
ICP undercut DIN 509 F ... 394
ICP undercut DIN 76 ... 392
ICP undercut type H ... 396
ICP undercut type K ... 397
ICP undercut type U ... 395
ICP vertical lines in XY plane ... 433
ICP vertical lines in YZ plane ... 450
ICP vertical lines of a turning contour ... 388
ICP vertical lines on face ... 399
ICP vertical lines on lateral surface ... 406
ICP workpiece blank contour ... 131
ICP workpiece blank, "bar" ... 386
ICP workpiece blank, "tube" ... 386
ICP zoom ... 385
Inch, units of measure ... 46
Incremental coordinates ... 45
Input box ... 51
Input fields ... 54
Inside thread calculation ... 370
Invert ... 376

K

Keystroke log file ... 63

L

Last cut in thread cycles ... 260
Linear dimension ... 404
Linear drilling pattern, radial ... 345
Linear machining at angle ... 137
Linear machining, longitudinal ... 135
Linear machining, transverse ... 136
Linear milling pattern, axial ... 339
List operations ... 54
Log file, error log file ... 62
Log file, keystroke log file ... 63

M

M functions ... 145
M functions in cycles ... 125
Machine data display ... 78
Machine data, entering ... 76
Machine mode of operation ... 72
Machine setup ... 89
Machine with multifix ... 83
Machine with turret ... 83
Machine zero point ... 45
Machining attributes, ICP ... 361
Manual Operation ... 104
Manual Operation operating mode ... 104
Marking (program transfer) ... 559
Menu selection ... 53
Metric, units of measure ... 46
Mid-program startup ... 109
Milling cutter radius compensation (MCRC) ... 48
Milling cycles, cycle programming ... 303
Milling direction for contour milling ... 328
Milling direction for pocket milling ... 329
Milling pattern
  Cycle programming
    Notes ... 336

M

Milling pattern circular, radial ... 351
Milling pattern linear, radial ... 347
Milling, axial figure ... 307
Milling, axial slot ... 305
Milling, face milling ... 314
Milling, ICP contour, axial ... 311
Milling, ICP contour, radial ... 323
Milling, radial figure ... 319
Milling, radial helical slot ... 326
Milling, radial slot ... 317
Mirroring
  Copying a contour section by mirroring ... 376
Monitoring EnDat encoders ... 73
Multipoint tools, editing ... 489

N

NC programs, importing from predecessor controls ... 564
Network connections ... 547
Network settings ... 550

O

Operating modes ... 38, 52
Operating modes, tool editor ... 484
Operation, the basics ... 52
Optical gauge ... 102
Organization mode of operation ... 520

P

Parameters ... 521
  Machining parameters ... 527
Parting ... 253
Parting tools ... 484
Pattern, circular drilling pattern, axial ... 341
Pattern, circular drilling pattern, radial ... 349
Pattern, circular milling pattern, axial ... 343
Pattern, circular milling pattern, radial ... 351
Pattern, linear drilling pattern, axial ... 337

P

Pattern, linear drilling pattern, radial ... 345
Pattern, linear milling pattern, axial ... 339
Pattern, linear milling pattern, radial ... 347
Polar coordinates ... 45
Position encoders ... 43
Positioning
  Spindle positioning in cycle mode ... 76
Program execution ... 110
Program information ... 116
Program Run ... 107
Program Run mode ... 107
Program selection ... 116
Program types ... 59
Proportioning of cuts ... 258
Protection zone
  Display of protection zone status ... 92
Protection zone, setting ... 92

R

Rapid positioning milling ... 304
Rapid traverse positioning ... 133
Recess turning with ICP, axial ... 242
Recess turning with ICP, radial ... 240
Recess turning with ICP, radial finishing ... 244
Recess turning, axial ... 226
Recess turning, axial finishing ... 234
Recess turning, axial finishing—expanded ... 238
Recess turning, ICP axial, finishing ... 246
Recess turning, radial ... 225
Recess turning, radial finishing ... 232
Recess turning, radial finishing—expanded ... 236
Recess turning, radial—expanded ... 228
Recess turning—fundamentals of cycle programming ... 224
Recessing axial, finishing ... 210
Recessing axial, finishing—expanded ... 214
Recessing cycles ... 198
Recessing cycles, contour forms ... 199
Recessing cycles, direction of cutting and infeed ... 198

R

Recessing cycles, undercut position ... 199
Recessing radial, finishing ... 208
Recessing radial, finishing—expanded ... 212
Recessing tools ... 484
Recessing turning, axial—expanded ... 230
Recessing, axial ... 202
Recessing, axial—expanded ... 206
Recessing, radial ... 200
Recessing, radial—expanded ... 204
Recess-turning tools ... 484
Reference mark ... 43
Reference run ... 74, 91
Rounding arc ... 143

S

Safety clearance ... 146
Safety clearance G47 ... 128
Safety clearances SCI and SCK ... 128
Screen ... 51
Service files, saving ... 63
Set axis values ... 90
Setting up machine dimensions ... 95
Simulation ... 115, 468
  Contour generation during simulation ... 481
Simulation window ... 471
Simulation with mid-program startup ... 478
Simulation, material-removal graphic ... 474, 475
Simulation, miscellaneous functions ... 470
Simulation, operating the ... 469
Simulation, setting up the views ... 471
Simulation, tool display ... 474
Simulation, traverse path ... 473
Simulation, zoom ... 476
Single cut cycles 132
Single-block mode
  Program execution ... 110
Slide position ... 35
smart.Turn dialogs ... 54
Soft ... 53
Soft keys ... 53
Sorting functions ... 116
Special compensation (recessing tools) ... 500, 501

S

Speed limiting
  Definition in cycle mode ... 76
Spindle ... 82
Spindle utilization ... 78
Starting point of ICP contour ... 368
Stopping angle (cycle mode) ... 76
Surface roughness
  Machining parameters ... 529
Switching functions for cycles ... 125
Switch-off ... 75
Switch-on ... 73
System time, setting ... 98

T

Tapered thread ... 265
Tapered thread, recutting ... 273
Tapping, axial ... 295
Tapping, radial ... 297
Teach-in ... 106
Teach-in mode ... 106
Teach-in operating mode ... 106
Technical characteristics ... 582
Technology editor ... 514
Thread cycle (longitudinal) ... 261
Thread cycle (longitudinal)—expanded ... 263
Thread cycles ... 257
Thread depth ... 258
Thread milling, axial ... 299
Thread parameters ... 572
Thread pitch ... 573
Thread position, cycle programming ... 257
Thread run-in ... 259
Thread run-out ... 259
Thread, recut (longitudinal) ... 269
Thread, recut expanded, (longitudinal) ... 271
Threads
  Cycle programming
    API thread ... 267
    Tapered thread ... 265
Time calculation (simulation) ... 480
TNCguide ... 64
Tool call ... 87
Tool carriers for multifix ... 83
Tool carriers, turret ... 83
Tool change position G14 ... 128
Tool change position, defining ... 93
Tool change position, moving to the 134

T
Tool compensation ... 103, 111
Tool dimensions, fundamentals ... 47
Tool editor ... 486
Tool life monitoring ... 88
Tool list ... 486
Tool list, comparing ... 108
Tool list, setting up ... 83
Tool measurement ... 99
Tool measurement by touching the workpiece ... 100
Tool measurement with a touch probe ... 101
Tool position in turning cycles ... 147
Tool touch probe, calibrating ... 96
Tool types ... 484
Tool-life data, editing ... 491
Tools
  Driven tools ... 87
  Tool administration (tool management) ... 484
  Tool compensation, entering ... 103
  Tool list ... 486
  Tools in different quadrants ... 84
Tools in different quadrants ... 84
Tools measurement with an optical gauge ... 102
Tool-tip radius compensation (TRC) ... 48
Touch probe ... 101
Touch-off ... 100
Transfer ... 546
Transformations
  Mirroring ... 384
  Rotating ... 383
  Shifting ... 382
Turning cycles ... 146
Turning cycles, example ... 194
Turret list, filling the ... 86
Turret list, filling with the tool list ... 85

U
Undercut
  Parameters, undercut DIN 509 E, DIN 509 F ... 581
  Parameters, undercut DIN 76 ... 579
Undercut cycles ... 257
Undercut DIN 509 E ... 279
Undercut DIN 509 F ... 281
Undercut DIN 76 ... 277
Undercut position, cycle programming ... 257
Undercutting type H ... 248
Undercutting type K ... 250
Undercutting type U ... 251
Units of measure ... 46
Unresolved contour elements (ICP) ... 362
USB interface ... 547

W
Wear compensation ... 484
Working with cycles ... 122
Workpiece blank contour, ICP ... 131
Workpiece blank cycles ... 129
Workpiece blank definition ICP ... 386
Workpiece zero point ... 46
Workpiece zero point, defining ... 90

Y
Y axis, fundamentals ... 36

Z
Zero point shift ... 375

Index

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 8669 31-0
FAX +49 8669 5061
E-mail: info@heidenhain.de

Technical support FAX +49 8669 32-1000
Measuring systems ☏ +49 8669 31-3104
E-mail: service.ms-support@heidenhain.de
TNC support ☏ +49 8669 31-3101
E-mail: service.nc-support@heidenhain.de
NC programming ☏ +49 8669 31-3103
E-mail: service.nc-pgm@heidenhain.de
PLC programming ☏ +49 8669 31-3102
E-mail: service.plc@heidenhain.de
Lathe controls ☏ +49 8669 31-3105
E-mail: service.lathe-support@heidenhain.de

www.heidenhain.de

1079662-21 · Ver01 · SW02 · H · 1/2014