

HEIDENHAIN



User's Manual

MANUALplus 620

NC Software 548430-04 548431-04

Controls and displays of the MANUALplus

Keys on visual display unit

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

Operating mode keys

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

Navigation keys

Key		Function
t	+	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

Кеу	Function block
0 9	Number keys 0-9:
	Numeric input keys
	Menu operation
	Decimal point
_/+	Switchover between positive and negative values
ESC	Escape key: Cancellation 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
X	Backspace: Deletes the character to the left of the cursor
CE	CE key: Deletes the error messages in the machine operating mode
>>	Continue: Enables input fields for additional entries in dialog boxes
ENT	Enter: Confirms the input

smart.Turn keys

Кеу	Function
	Go to the next form
	Next/previous group

Special keys

Кеу	Function
ERR	Error key: Opens the error window
CALC	Starts the integrated calculator
Ĺ	Info key: Shows additional information in the parameter editor
бото П	Activates special functions, such as input options or input of characters as on an alphabetic keyboard
PRT SC	Print screen: Generates a screen saver
	DIADUR key

Machine operating panel

Кеу	Function
	Cycle start / cycle stop
₩ ○	Feed rate stop
° Å	Spindle stop
	Spindle on – M3/M4 direction
6	Spindle jog – M3/M4 direction. The spindle rotates as long as you press the key.
↑ ↓	Manual direction keys +X/-X

Operating panel of the MANUALplus

8									0	
	Teach-in	smartTurn	Tool e	ditor	8				8	
	X 100.000	۵X	X ° <u>50 100 150</u>	02 T x 0.00	001 00 Z 0.000		-	ERR CALC	i	
	Z 2.000	△Z	Z ° <u>50 100 150</u> S ° <u>50 100 150</u>	02 02		Mx3		7 8	9 🖂	
6	1002 0.0nn/nin	1 1 0 0.00/1	002 1 2 0 0.0	02 U/nin S100%	F100%	Mx4		4 5	6 DEL	
	H-B		X 11 X1 80	00 Z [0 Z1 [2 0	Mx19		1 2	3	
	H-1		- ^{©I} P 2	8 Z2 [H [-20 0: With e	-		CE 0	• =/+	
				W		Mx5		ESC EN	TINS	
	H=2	- <u>P</u>		G14	0: simult <u>-</u>	1				
			S 8 Feed d	00 F	0.8					
õ	Expanded Finishin run	g Tool Talist p	ake over Proposed position technolog	Constant y speed	Input finished	Back	$\overline{\mathbb{C}}$	HOME	PG UP	
4						-			PG DN	
0									0	

MANUALplus 620, software and features

This manual describes functions that are available in the MANUALplus with NC software number 548430-04 and 548431-04.

The programming of smart.Turn and DIN PLUS is 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 MANUALplus on your machine tool.

Some of the MANUALplus 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 MANUALplus 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 offlocation 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 MANUALplus complies with the limits for 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

- Organization mode of operation
- ▶ LICENSE INFO SOFT KEY

5

New functions of software 548328-04

- In the simulation submode, the current contour description (of workpiece blank and finished part) can be mirrored and saved. In the smart.Turn mode of operation, these contours can be reinserted (see page 500)
- On machines with opposing spindle, the workpiece spindle can now be selected in the TSF menu (see page 107)
- On machines with opposing spindle, its zero point can be shifted (see Page 107)
- The user documentation is now also available in the context-sensitive help system TURNguide (see Page 66)
- You can make your own project folder in the project management, so that you can centrally manage associated files (see Page 142)
- With a manual tool change system it is possible to insert tools that are not in the turret during a program run (see Page 517)
- Engraving cycles are now available in the Teach-In submode (see Page 350)
- During tool data backup, you can now select in a dialog window the data to be saved or restored (see Page 598)
- 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 548328-05 and 54843x-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 548)
- 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 58)
- Tool touch probes can now be calibrated in the Machine Setup menu (see also "Calibrating the tool touch probe" on page 109)
- The workpiece zero point can now also be set in the direction of the Z axis using a touch probe (see also "Machine setup" on page 100)
- In Teach-in submode, 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 256)
- The oversizes RI and RK for the workpiece blank were introduced for finishing in the recess-turning units and in Cycle G869 (see smart.Turn and DIN Programming User's Manual).
- 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 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).

7

New functions of software 54843x-02

- The "Zero point shift" miscellaneous function was introduced in ICP (see also "Zero point shift" on page 394)
- In ICP contours, fit dimensions and inside threads can now be calculated using an input form (see also "Fits and inside threads" on page 389)
- The miscellaneous functions "Copy in linear/circular series, and by mirroring" were introduced in ICP (see also "Copying a contour section in linear series" on page 394)
- The system time can now be set using an input form (see also "Displaying operating times" on page 110)
- The parting cycle G859 was expanded by the parameters K, SD and U (see also "Parting" on page 272)
- The angle of approach and departure can now be defined for ICP recess turning (see also "ICP recess turning, radial finishing" on page 264)
- 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)

New functions of software 54843x-03

- In the Teach-In submode, the parameter RB was added to the cycles "Figure, axial", "Figure, radial", "ICP contour, axial" and "ICP contour, radial" (see "Milling cycles" on page 322)
- In the Teach-In submode, the parameters SP and SI were added to all tapping cycles (see "Drilling cycles" on page 305)
- In the Simulation submode, the 3-D view provides additional features (see "3-D view" on page 494)
- Tool control graphics were introduced in the Tool Editor mode of operation (see "Tool control graphics" on page 511)
- An ID number can be entered directly in the turret list (see "Editing the turret list" on page 92)
- The tool list provides additional filter options (see "Sorting and filtering the tool list" on page 508)
- The Transfer submode provides enhanced tool backup functionality (see "Transferring tool data" on page 598)
- The Transfer submode provides enhanced tool import functionality (see "Importing tool data of the CNC PILOT 4290" on page 606)
- The Set Axis Values menu item now also enables you to define offset values for shifts using G53, G54 and G55 (see "Defining offsets" on page 102)
- Load monitoring was introduced in the Program Run submode (see "Load monitoring (option)" on page 131)
- The definition of skip levels was introduced in the Program Run submode (see "Program execution" on page 125)
- A function was introduced to query information on the tool status (see "Tool life monitoring", page 97), (see "Editing tool-life data" on page 515)
- A user parameter was introduced to enable you to activate and deactivate the software limit switches for the Simulation submode (see "List of machine parameters" on page 551)
- A user parameter was introduced to enable you to suppress the error message for the software limit switches (see "List of machine parameters" on page 551)
- A user parameter was introduced to enable you to use NC Start for executing a tool change programmed in the T,S,F dialog (see "List of machine parameters" on page 551)
- A user parameter was introduced to divide the T,S,F dialog into separate dialogs (see "List of machine parameters" on page 551)
- The parameter WE was added to G32 (see the smart.Turn and DIN Programming User's Manual)
- The parameters U, V and W were added to G51, G56 and G59 (see the smart.Turn and DIN Programming User's Manual)
- Parameters ensuring maximum compatibility with the ICP contour description were added to G0, G1, G12/G13, G101, G102/G103, G110, G111, G112/G113, G170, G171, G172/G173, G180, G181 and G182/G183 (see the smart.Turn and DIN Programming User's Manual)
- The parameter C was added to G808 (see the smart.Turn and DIN Programming User's Manual)

- The parameter U was added to G810 and G820 (see the smart.Turn and DIN Programming User's Manual)
- The parameter D was added to G4 and G860 (see the smart.Turn and DIN Programming User's Manual)
- The parameter B was added to G890 (see the smart.Turn and DIN Programming User's Manual)
- The parameter RB was added to the units G840 "Contour milling, figures" and G84X "Pocket milling, figures" (see the smart.Turn and DIN Programming User's Manual)
- The parameters SP and SI were added to all tapping units (see the smart.Turn and DIN Programming User's Manual)
- G48 was introduced to allow limiting the rapid traverse rate for rotary and linear axes (see the smart.Turn and DIN Programming User's Manual)
- G53, G54 and G55 were introduced for zero point shifts using offset values (see the smart.Turn and DIN Programming User's Manual)
- The functions for superimposing axis movements G725 "Eccentric turning", G726 "Transition to eccentric" and G727 "Eccentric X" were introduced (see the smart.Turn and DIN Programming User's Manual)
- The load monitoring functions G995 "Monitoring zone definition" and G996 "Type of load monitoring" were introduced (see the smart.Turn and DIN Programming User's Manual)
- The AWG submode now also supports tools with quick-change holders (see the smart.Turn and DIN Programming User's Manual)
- A tree view is available in the smart.Turn operating mode (see the smart.Turn and DIN Programming User's Manual)
- Skip levels can be defined in the smart.Turn operating mode (see the smart.Turn and DIN Programming User's Manual)
- A function was introduced to query information on the tool status (see the smart.Turn and DIN Programming User's Manual)

New functions of software 54843x-04

- In the Simulation submode, the "Contour measurement" function was introduced (see "Dimensioning" on page 501)
- In the Simulation submode, the "Save contours" function was enhanced (see "Saving the contour" on page 500)
- In the Simulation submode, display of the B head is supported (see "Depicting the tool holder in simulation submode" on page 493)
- In the Teach-In submode, the workpiece blank is also redrawn with centric drilling (see "Contour follow-up in Teach-in submode" on page 148)
- In the Teach-In submode, the GK parameter can now also be negatively programmed with taper threads (see "Thread and undercut cycles" on page 276)
- In the ICP Editor submode, contour groups are supported. The number of the contour group is displayed at top left in the graphic window (see "Contour groups" on page 484)
- Machine parameter 602414 is now evaluated in the Teach-In submode, so that the possibilities "Part a horizontal element" and "Move through and lift-off" are also available here (see "Recessing cycles" on page 218)
- New machine parameter 602023 for converting ICP contours (see "List of machine parameters" on page 551)
- The machining parameters for approach and depart have been adapted (see "List of machine parameters" on page 551)
- The reamer tool type (type 43 from CNC PILOT 4290) is supported (see "Tool types" on page 506)
- The navigation and tool parameters view has been improved in the tool list (see "Navigating in the tool list" on page 508)
- The pocket type tool parameter was introduced (see "General tool parameters" on page 521)
- Magazine pocket systems are supported, (see "Editing the magazine list" on page 93)
- Tool compensations can now be entered with the handwheel or in a dialog (see "Tool compensation" on page 116), (see "Tool compensation in Teach-in submode" on page 153)
- When configuring the C axis, you can set a defined value on the current position (see "Setting C-axis values" on page 106)
- It is now possible to sequentially run several main programs automatically. A program list is generated for this purpose. With each program, you can now specify how often it is run before the next program is started (see "Automatic job" on page 126)
- The continuous run status is also kept if the control is restarted in the Program Run submode (see "Program run submode" on page 122)
- Programs can be deleted in the file manager even though they are selected in the Program Run submode, after program block display is deselected (see "Program run submode" on page 122)
- For systems with a C axis, the position display in the machine data display (axis letter and index) can be configured by the machine manufacturer

- The functions G0, G1 and G701 have been enhanced with parameters for auxiliary axes
- Programming with variables in smart.Turn mode is now possible with soft keys (see the smart.Turn and DIN Programming User's Manual)
- The number of local variables was increased from 30 to 99 (see the smart.Turn and DIN Programming User's Manual)
- In the NC program, the status of shifts G920/G921 can now be queried with the variables #n920(G) (see the smart.Turn and DIN Programming User's Manual)
- In smart.Turn the number of an M function can now be defined with a variable (see the smart.Turn and DIN Programming User's Manual)
- In smart.Turn, up to four contour groups are supported (see the smart.Turn and DIN Programming User's Manual)
- In a program generated by the AWG submode, the tool traverses to the tool change position after the parting working step (see the smart.Turn and DIN Programming User's Manual)
- In a program generated by the AWG submode, machining is now also possible with simplified geometry programming (see the smart.Turn and DIN Programming User's Manual)
- The TURNPLUS function can now also be used in INCH mode (see the smart.Turn and DIN Programming User's Manual)
- The CW parameter has been changed to a Reverse Tool yes/no query (see the smart.Turn and DIN Programming User's Manual)
- The Q parameter in G99 is supported (see the smart.Turn and DIN Programming User's Manual)
- The G860 ICP Contour Recessing and Direct Contour Recessing cycles have been enhanced with the DO program run parameter (see the smart.Turn and DIN Programming User's Manual)
- The tool contact type parameter can now also be modified via the machining parameters in smart.Turn (see the smart.Turn and DIN Programming User's Manual)
- Die G function "G154 Short Path in C" was introduced (see the smart.Turn and DIN Programming User's Manual)
- The G function G741 has been enhanced with the parameter O program run (see the smart.Turn and DIN Programming User's Manual)
- Parameter A of the G function G845 has been enhanced with the input option pilot drilling in the reference point of the figure (see the smart.Turn and DIN Programming User's Manual)
- The input range was expanded for drilling depth in the G74 drilling cycle (see the smart.Turn and DIN Programming User's Manual)
- No error message is displayed anymore in the paraxial turning cycles if the task is to be carried out with the secondary cutting edge of the tool (see the smart.Turn and DIN Programming User's Manual)
- Machining parameters are displayed in millimeters or inches according to the CfgUnitOfMeasure parameter

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.

Would you like 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**.

About this manual

Contents

Introduction and fundamentals

Basics of operation

Machine mode of operation

Cycle Programming

ICP programming

Submode simulation

Tool and technology database

Organization mode of operation

Tables and overviews

Overview of cycles



```
1.1 The MANUALplus ..... 36
       The MANUALplus for cycle lathes ..... 36
       MANUALplus for CNC lathes ..... 36
1.2 Configuration ..... 37
       Slide position ..... 37
       Tool carrier systems ..... 37
       The C axis ..... 38
       The Y axis ..... 38
       Full-surface machining ..... 39
1.3 Features ..... 40
       Configuration ..... 40
       Modes of operation ..... 40
1.4 Data backup ..... 42
1.5 Explanation of terms ..... 43
1.6 MANUALplus design ..... 44
1.7 Fundamentals ..... 45
       Position encoders and reference marks ..... 45
       Axis designations ..... 45
       Coordinate system ..... 46
       Absolute coordinates ..... 46
       Incremental coordinates ..... 47
       Polar coordinates ..... 47
       Machine zero point ..... 47
       Workpiece zero point ..... 48
       Units of measure ..... 48
1.8 Tool dimensions ..... 49
       Tool length ..... 49
       Tool compensation ..... 49
       Tool-tip radius compensation (TRC) ..... 50
       Milling cutter radius compensation (MCRC) ..... 50
```

2.1 General information on operation 52 Operation 52 Setup 52 Programming - Teach-in 52 Programming - smart.Turn 52 2.2 The MANUALplus screen 53 2.3 Operation and data input 54 Operating modes 54 Menu selection 55 Soft keys 55 Data input 56 smart.Turn dialogs 56 List operations 57 Alphanumeric keyboard 57 2.4 Integrated calculator 58 Calculator functions 58 Adjusting the position of the calculator 60 2.5 Types of programs 61 2.6 The error messages 62 Display of errors 62 Opening the error window 62 Closing the error window 62 Detailed error messages 63 Soft key INTERNAL INFO 63 Clearing errors 64 Error log 64 KEYSTROKE LOG 65 Save service files 65 2.7 TURNguide context-sensitive help system 66 Application 66 Working with the TURNguide 67 Downloading current help files 71 2.8 The DataPilot programming station 73 Application 73 Operation 73

3 Machine mode of operation 75

3.1 Machine mode of operation 76 3.2 Switch-on / Switch-off 77 Switch-on 77 Monitoring EnDat encoders 77 Traversing the reference marks 78 Switch-off 79 3.3 Machine data 80 Input of machine data 80 Machine data display 84 Cycle statuses 88 Axis feed rate 88 Spindle 88 3.4 Pocket table setup 89 Machine with a tool holder (multifix) 89 Machine with turret 90 Machine with magazine 90 Filling the turret list with the tool list 91 Editing the turret list 92 Editing the magazine list 93 Tool call 95 Driven tools 96 Tools in different quadrants 96 Tool life monitoring 97 3.5 Machine setup 100 Defining the workpiece zero point 101 Defining offsets 102 Homing the axes 103 Setting the protection zone 104 Defining the tool change position 105 Setting C-axis values 106 Setting up machine dimensions 108 Calibrating the tool touch probe 109 Displaying operating times 110 Setting the system time 111 3.6 Tool measurement 112 Touch off 113 Touch probe (tool touch probe) 114 Optical gauge 115 Tool compensation 116

3.7 Manual Operation 118 Tool change 118 Spindle 118 Handwheel operation 118 Manual direction keys 119 Teach-in cycles in the Machine operating mode 119 3.8 Teach-in submode 120 Teach-In mode 120 Programming Teach-in cycles 121 3.9 Program run submode 122 Loading a program 122 Comparing a tool list 123 Before executing a program 123 Finding a start block 124 Program execution 125 Automatic job 126 Entering compensation values during program run 127 Program execution in "dry run" mode 130 3.10 Load monitoring (option) 131 Reference machining 133 Checking the reference values 134 Adapt limit values 136 Using load monitoring during production 137 3.11 Graphic simulation 138 3.12 Program management 139 Program selection 139 File manager 141 Project management 142 3.13 Conversion into DIN format 143 Making a conversion 143 3.14 Units of measure 144

4 Cycle Programming 145

4.1 Working with cycles 146	
Cycle starting point 146	
Help graphics 147	
DIN macros 147	
Graphical test run (simulation) 147	
Contour follow-up in Teach-in submode 148	
Cycle keys 148	
Switching functions (M functions) 149	
Comments 149	
Cycle menu 150	
Tool compensation in Teach-in submode 15	3
Addresses used in many cycles 154	
4.2 Workpiece blank cycles 155	
Bar/tube blank 156	
ICP workpiece blank contour 157	
4.3 Single cut cycles 158	
Rapid traverse positioning 159	
Move to the tool change position 160	
Linear machining, longitudinal 161	
Linear machining, transverse 162	
Linear machining at angle 163	
Circular machining 165	
Chamfer 167	
Rounding arc 169	
M functions 171	

i

4.4 Turning cycles 172 Tool position 173 Cut longitudinal 175 Cut transverse 176 Roughing, longitudinal—expanded 177 Roughing, transverse—expanded 179 Finishing cut, longitudinal 181 Finishing cut, transverse 182 Finishing cut, longitudinal—expanded 183 Finishing cut, transverse—expanded 185 Cut, longitudinal plunge 187 Cut, transverse plunge 189 Cut, longitudinal plunging-expanded 191 Cut, transverse plunging-expanded 193 Cut, longitudinal finishing plunge 195 Cut, transverse finishing plunge 196 Cut, longitudinal finishing plunge-expanded 198 Cut, transverse finishing plunge-expanded 200 Cut, ICP contour-parallel, longitudinal 202 Cut, ICP contour-parallel, transverse 204 Cut, ICP contour-parallel, longitudinal finishing 206 Cut, ICP contour-parallel, transverse finishing 207 ICP cutting, longitudinal 208 ICP cut transverse 210 ICP longitudinal finishing cut 212 ICP transverse finishing cut 213 Examples of turning cycles 214

4.5 Recessing cycles 218 Cutting and infeed directions for recessing cycles 218 Undercut position 219 Contour forms 219 Recessing, radial 220 Recessing, axial 222 Recessing, radial-expanded 224 Recessing, axial-expanded 226 Recessing radial, finishing 228 Recessing axial, finishing 230 Recessing radial, finishing-expanded 232 Recessing axial, finishing-expanded 234 ICP recessing radial 236 ICP recessing cycles, axial 238 ICP recessing, radial finishing 240 ICP recessing, axial finishing 242 Recess turning 244 Recess turning, radial 245 Recess turning, axial 246 Recess turning, radial-expanded 248 Recess turning, axial-expanded 250 Recess turning, radial finishing 252 Recess turning, axial finishing 254 Recess turning, radial finishing-expanded 256 Recess turning, axial finishing-expanded 258 ICP recess turning, radial 260 ICP recess turning, axial 262 ICP recess turning, radial finishing 264 ICP recess turning, axial finishing 266 Undercutting type H 268 Undercutting type K 269 Undercutting type U 270 Parting 272 Examples of recessing cycles 274

4.6 Thread and undercut cycles 276 Thread position, undercut position 276 Handwheel superimposition 277 Feed angle, thread depth, proportioning of cuts 278 Thread run-in / thread run-out 278 Last cut 279 Thread cycle (longitudinal) 280 Thread cycle (longitudinal)-expanded 282 Tapered thread 284 API thread 287 Recut (longitudinal) thread 289 Recut (longitudinal) thread—expanded 291 Tapered thread, recutting 293 API thread, recutting 295 Undercut DIN 76 297 Undercut DIN 509 E 299 Undercut DIN 509 F 301 Examples of thread and undercut cycles 303 4.7 Drilling cycles 305 Drilling, axial 306 Drilling, radial 308 Deep-hole drilling, axial 310 Deep-hole drilling, radial 312 Tapping, axial 314 Tapping, radial 316 Thread milling, axial 318 Examples of drilling cycles 320 4.8 Milling cycles 322 Rapid positioning milling 323 Slot, axial 324 Figure, axial 326 ICP contour, axial 330 Face milling 333 Slot, radial 336 Figure, radial 338 ICP contour, radial 342 Helical-slot milling, radial 345 Milling direction for contour milling 347 Milling direction for pocket milling 348 Example of milling cycle 349 Engraving, axial 350 Engraving, radial 352 Engraving, axial/radial 354

4.9 Drilling and milling patterns 355
Drilling pattern linear, axial 356
Milling pattern linear, axial 358
Drilling pattern circular, axial 360
Milling pattern circular, axial 362
Drilling pattern linear, radial 364
Milling pattern linear, radial 366
Drilling pattern circular, radial 367
Milling pattern circular, radial 369
Examples of pattern machining 371

4.10 DIN cycles 374

DIN cycle 374

5.1 ICP contours 378 Loading contours 379 Form elements 379 Machining attributes 380 Calculation of contour geometry 380 5.2 ICP editor submode in cycle mode 381 Editing contours for cycles 381 File organization with the ICP editor submode 382 5.3 ICP editor submode in smart. Turn mode of operation 383 Machining a contour in smart.Turn mode 385 5.4 Creating an ICP contour 387 Entering an ICP contour 388 Absolute or incremental dimensioning 389 Transitions between contour elements 389 Fits and inside threads 389 Polar coordinates 390 Angular input 390 Contour graphics 391 Selection of solutions 392 Colors in contour graphics 392 Selection functions 393 Zero point shift 394 Copying a contour section in linear series 394 Copying a contour section in circular series 395 Copying a contour section by mirroring 395 Inverting 395 Contour direction (cycle programming) 396 5.5 Editing ICP contours 397 Superimposing form elements 397 Adding contour elements 397 Editing or deleting the last contour element 398 Deleting a contour element 398 Editing contour elements 399 5.6 Zoom functions in ICP editor submode 404 Changing the view 404 5.7 Defining the workpiece blank 405 "Bar" blank 405 "Tube" blank 405 "Cast part" blank 405 5.8 Contour elements of a turning contour 406 Basic elements of a turning contour 406 Contour form elements 410

5.9 Contour elements on face 416 Starting point of face contour 416 Vertical lines on face 418 Horizontal lines on face 418 Line at angle on face 419 Circular arc on face 420 Chamfer/rounding arc on face 421 5.10 Contour elements on lateral surface 422 Starting point of lateral surface contour 422 Vertical lines on lateral surface 424 Horizontal lines on lateral surface 424 Line at angle on lateral surface 425 Circular arc on lateral surface 426 Chamfer/rounding arc on lateral surface 427 5.11 C and Y axis machining in smart. Turn mode 428 Reference data, nested contours 429 Representation of the ICP elements in the smart. Turn program 430 5.12 Face contours in smart.Turn mode 431 Reference data for complex face contours 431 TURN PLUS attributes 432 Circle on face 432 Rectangle on face 433 Polygon on face 434 Linear slot on face 435 Circular slot on face 435 Hole on face 436 Linear pattern on face 437 Circular pattern on face 438 5.13 Lateral surface contours in smart. Turn mode 439 Reference data of lateral surface 439 TURN PLUS attributes 440 Circle on lateral surface 441 Rectangle on lateral surface 442 Polygon on lateral surface 443 Linear slot on lateral surface 444 Circular slot on lateral surface 445 Hole on lateral surface 446 Linear pattern on lateral surface 447 Circular pattern on lateral surface 448

5.14 Contours in the XY plane 449 Reference data in XY plane 449 Starting point of contour in XY plane 450 Vertical lines in XY plane 450 Horizontal lines in XY plane 451 Line at angle in XY plane 452 Circular arc in XY plane 453 Chamfer/rounding arc in XY plane 454 Circle in XY plane 455 Rectangle in XY plane 456 Polygon in XY plane 457 Linear slot in XY plane 458 Circular slot in XY plane 459 Hole in XY plane 460 Linear pattern in XY plane 461 Circular pattern in XY plane 462 Single surface in XY plane 463 Centric polygon in XY plane 464 5.15 Contours in the YZ plane 465 Reference data in YZ plane 465 TURN PLUS attributes 466 Starting point of contour in YZ plane 467 Vertical lines in YZ plane 467 Horizontal lines in YZ plane 468 Line at angle in YZ plane 469 Circular arc in YZ plane 470 Chamfer/rounding arc in YZ plane 471 Circle in YZ plane 472 Rectangle in YZ plane 472 Polygon in YZ plane 473 Linear slot in YZ plane 474 Circular slot in YZ plane 475 Hole in YZ plane 476 Linear pattern in YZ plane 477 Circular pattern in YZ plane 478 Single surface in YZ plane 479 Centric polygons in YZ plane 480 5.16 Loading existing contours 481 Integrating cycle contours in smart.Turn mode 481 DXF contours (option) 482 5.17 Contour groups 484 Contour groups in smart. Turn mode 484

6 Submode simulation 485

6.1 The submode Simulation 486
Using the simulation submode 487
The miscellaneous functions 489
6.2 Simulation window 490
Setting up the views 490
Single-window view 491
Multiple window view 491
6.3 Views 492
Traverse path display 492
Tool depiction 493
Material-removal graphic 493
3-D view 494
6.4 The zoom function 496
Adjusting the visible section 496
6.5 Simulation with mid-program startup 497
Startup block with smart.Turn programs 497
Mid-program startup in cycle programs 498
6.6 Time calculation 499
Showing the machining times 499
6.7 Saving the contour 500
Saving the generated contour in simulation submode 500
6.8 Dimensioning 501
Measuring contours in simulation submode 501
6.9 3-D simulation 503
3-D simulation in simulation submode 503

i

7 Tool and technology database 505

7.1 Tool database 506 Tool types 506 Multipoint tools 507 Tool life management 507 7.2 Tool editor mode of operation 508 Navigating in the tool list 508 Sorting and filtering the tool list 508 Editing the tool data 510 Tool control graphics 511 Tool texts 512 Editing multipoint tools 513 Editing tool-life data 515 Manual change systems 517 7.3 Tool data 521 General tool parameters 521 Standard turning tools 524 Recessing tools 525 Thread-cutting tools 526 Twist drills and indexable-insert drills 527 NC center drill 528 Centering tool 529 Counterbore 530 Countersink 531 Reamer 532 Tap 533 Standard milling tools 534 Thread milling tools 535 Angle cutters 536 Milling pins 537 Knurling tool 538 Touch probes 539 Stopper tool 540 Gripper 541 7.4 Technology database 542 Technology editor submode 543 Editing a workpiece material or cutting material list 544 Displaying/editing cutting data 545

8 Organization mode of operation 547

8.1 Organization mode of operation 548 8.2 Parameters 549 Parameter editor 549 List of machine parameters 551 Descriptions of the most important machining parameters (processing) 565 General settings 565 Thread cutting 580 8.3 Submode Transfer 583 Data backup 583 Data exchange with TNCremo 583 External access 583 Connections 584 Ethernet interface (with software 548328-xx) 585 Ethernet interface (with software 54843x-xx) 586 USB connection 593 Data transfer options 594 Transferring programs (files) 595 Transferring parameters 597 Transferring tool data 598 Service files 600 Creating a data backup file 601 Importing NC programs from predecessor controls 602 Importing tool data of the CNC PILOT 4290 606 8.4 Service pack 607 Installing a service pack 607

9 Tables and overviews 609

9.1 Thread pitch 610

Thread parameters 610
Thread pitch 611

9.2 Undercut parameters 617

DIN 76—undercut parameters 617
DIN 509 E – undercut parameters 619
DIN 509 F – undercut parameters 619

9.3 Technical information 620
9.4 Compatibility in DIN programs 629

Syntax elements of the MANUALplus 620 631

10 Overview of cycles 643

10.1 Workpiece blank cycles, single cut cycles 644

10.2 Turning cycles 645

10.3 Recessing and recess-turning cycles 646

10.4 Thread cycles 647

10.5 Drilling cycles 648

10.6 Milling cycles 649





Introduction and fundamentals

1.1 The MANUALplus

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

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

The MANUALplus for cycle lathes

On the MANUALplus, rework or simple tasks can be done in the same way as on a conventional lathe. You move the axes in the normal manner by turning the handwheels. The cycles of the MANUALplus are available for difficult cuts like tapers, undercuts or threads. For small and medium-size production runs, cycle programming will increase your revenue. When machining the first workpiece, you can store the machining cycles, and already save valuable time when machining the second workpiece. For increased requirements and complex machining tasks, you will profit from the new programming mode smart.Turn.



MANUALplus for CNC lathes

Regardless of whether you are turning simple parts or complex workpieces, the MANUALplus 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: Simply switch to DINplus. This programming mode helps you solve all your special tasks.

The MANUALplus supports operations with the C axis in cycle, smart.Turn and DIN programming. In the Y axis, the MANUALplus 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 MANUALplus. 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 control supports the following systems as tool carriers:

- Multifix holder with one tool mount
- Turret with **n** tool mounts
- Turret with **n** tool mounts and **one** multifix holder with one tool mount. Here it is possible for one of the two tool carriers to be arranged mirrored on the workpiece side opposite from the standard tool carrier.
- Two multifix holders, each with **one** tool mount. The tool carriers are arranged opposite to each other. One of them is then mirrored.
- Magazine with m tool mounts and one tool carrier in the working space with one tool mount

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 MANUALplus 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 MANUALplus supports program creation with the Y axis in:

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



Full-surface machining

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

The MANUALplus 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

Modes of operation

Machine mode of 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 submode

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

Program run submode

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

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

Setup functions

- Setting the workpiece zero point
- Defining the tool-change point
- Defining the 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
- Optional tool magazine

Technology database

- Cutting data is 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)
- Caxis: Interpolation in the linear axes X and Z with the Caxis
- 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

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 items also appear dimmed.
- Menu, menu item: The MANUALplus 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 item.
- **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 MANUALplus 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 functions united into a dialog in smart.Turn.

1.6 MANUALplus 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 keys 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 MANUALplus does not need an alphanumeric keyboard. Tool descriptions, program descriptions or comments in an NC program are entered with an on-screen alphanumeric keyboard. 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 MANUALplus .

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 MANUALplus 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 ${\bf X}$ axis and the saddle as the ${\bf Z}$ 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 coordinates entered for the **principal axes** X, Y 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 MANUALplus 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 to 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 MANUALplus 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

i

1.8 Tool dimensions

The MANUALplus 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 MANUALplus 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 MANUALplus 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 MANUALplus 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 MANUALplus 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 MCRC function compensates for this error by calculating a new path of traverse **equidistant line**.







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

- Select the Teach-in submode 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 MANUALplus screen

2.2 The MANUALplus screen

The MANUALplus 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) show 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 **Machine** 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 **triple-arrow key** (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.





3

2.3 Operation and data input

Operating modes

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

- Machine—with the submodes:
 - Teach-in
 - Program run
 - ICP editor
 - Reference
 - Simulation
- smart.Turn—with the submodes:
 - ICP editor
 - Automatic working plan generation (AWG)
 - Simulation
- Tool editor—with the submodes:
 - Technology editor
- Organization—with the submodes:
 - Machine para. Programming and editing
 - Transfer

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 MANUALplus 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 operating mode).



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, in the **Teach-In** submode 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.

➡ Program r	un)	🚯 smart.Turn		A Tool editor		8	
							Q0 V0 D1 00
X [®] 300).000 Δ Χ		W	0.000 ID		001	
Z 450).000 △Z		🖞 C 1	Т	1	X 0.0000 Z 0.0000	
Y ().000 ∆ Y		C 4	R	i •	0.000	
🗄 1 🔃 📲	0 mn/nin 1	1 0 0.0 116.0	Si Si	3000 U/min 03 1	F 100%	S ₁ 100%	-
Prog 📕 Se	quence 👖 Di	isplay					
TNC:\Project\	SHB_KAPITEL2	\ncps\bar.nc					
N 8G1Z-40 N 9G1X0						1	
MACHTNING							
N 50 UNITID N 51 [<uni< td=""><td>"START" [Pro t ID="START"</td><td>gram beginningj S0="3000" S1=""</td><td>\$2="" \$3="</td><td>" S4="" S5="" Z="</td><td>" WT1="6</td><td>9" WX1="</td><td></td></uni<>	"START" [Pro t ID="START"	gram beginningj S0="3000" S1=""	\$2="" \$3="	" S4="" S5="" Z="	" WT1="6	9" WX1="	
N 52 L"Ini N 53 G26 S	tStart" V1 3000						
N 54 G14 Q)					_	
N 55 END_OF_	JNTI 2585322	9239					201
N 180 UNIT ID	'G820_G80" [TD="G820 G	G820 Roughing, c 80" YS="52" 75="	lirect trans 2" T="1" TT	verse] D="" F="0 3" S="3	!AA" ¥1="	58" 71=	
N INTE ESTIN		M2				··· ·· ··	10.22
N 101 [Cunii N 102 T1			-				10.32
N 101 [CUN1 N 102 T1 N 103 C96 S	-						
N 101 [Cull N 102 T1 N 103 C98 S Program list	Contin.	Single To block c	ool/Add.	Bill Bill	ase	Start blck	Back

Data input

Input windows comprise several **input fields.** You can move the cursor to the desired input field with the vertical arrow keys. The MANUALplus 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 **Cance1** 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**.



ICP	cut longi	tud.	
Х	23.405	Z	31.7025
FK	Huelse		
Ρ	5	Н	0: With €▼
I		K	
E		0	0: No 💌
SX		SZ	-27
G47	2		
Т	1	G14	0: Simult <u>-</u>
ID	001		
S	200	F	0.35
Star	ting poin	t [mm]	1/2

Overva. Toor cont	our oye	Te	GIUDAL
Appro XS <mark>52</mark>	Appro	ZS	2
ID number	TID 5		
> Feed		F	0.3
Cutting speed		5	300
Start point contour		X1	50
Start point contour		Z1	0
End point contour		X2[0
End point contour		Z2	1
Maximum infeed			P 2
Oversize X			I <mark>0.500</mark>
Oversize Z			K <mark>0.200</mark>

2.3 Operation and data input

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.

Entering 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 MANUALplus opens the Text Input 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 CONFIRM VALUE soft key enables you to load the current value from the calculator to the active input field.

Using the calculator

 \triangleright

Use the arrow keys to select the input field.



 \triangleleft

▶ 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 MANUALplus 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.

V#

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

View

Calculate function	Shortcut (soft key)
Addition	+
Subtraction	-
Multiplication	*
Division	/
Calculations in parentheses	()
Arc	ARC
Sine	SIN



Standa	ard					×
Vi	.ew					0.
+	-	*	1	7	8	9
()	CE	=	4	5	6
ARC	SIN	COS	TAN	1	2	3
х^у	SQRT	1/x	PI	0		±

Scien	tific					X
Vi	.ew					0.
+	-	*	1	7	8	9
()	CE	=	4	5	6
ARC	SIN	COS	TAN	1	2	3
х^у	SQRT	1/x	PI	0		±
M+	MS	MR	MC	D	Е	F
LN	LOG	e^x	SGN	A	В	C
ABS	INT	FRAC	MOD			
DEG	RAD	MM	INCHE	DEC	•	

Equat:	ion ed	itor				×
Vi	.ew					
+	-	*	/	7	8	9
()	CE	=	4	5	6
ARC	SIN	COS	TAN	1	2	3
х^у	SQRT	1/x	PI	0		±

Calculate function	Shortcut (soft key)
Cosine	COS
Tangent	TAN
Powers of values	Х^Ү
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
Exponent 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)

 \bigcirc

The arc calculate function only has a function in connection with SIN, COS or TAN.

The calculator writes the inverse function as ASIN, ACOS or ATAN.



Adjusting the position of the calculator

You can move the calculator as follows:



Move the calculator with the arrow keys

You can also shift the calculator with a connected mouse.

➡ Teach-in		Smart.Tu	Jrn	🍴 Tool ed	itor	8	
							Q0 V0 D1 00
X [®] 30	0.000 🗠	X	W	0.0	90 ID	001	-
Z 45	0.000 🛆	Z	🛛 🔀 C 1		T	1 × 0.0000 z 0.0000)
Y	0.000 🗠	Y	C 4		₩ ⊕	0.000	
🔄 1 🙋	0. Standard		×	3080 U/min	et 1 F 100 R 100	ξ S 1 100%	
	View		0.	ICP cut	longitud.		
H-8	+	- * /	/ 8 9	X 23.	405 Z	31.7025	
			4 5 6	FK Hue	lse		
	ARC S.			P 5	— н	0: With €	
H-1	<u>x.</u> x.	1/X PI	±		К		
			1	E	0	U: No Y	
			0X	76	52	-2/	
H-2			1	G47 2		0.01.11	
				TD 001	014	-	-
	-			5 200		0.35	
				0 1200		10.00	
_impbedienion1/b_c024_1.png				Startin	g point [mm]	1/2	18:31
e^x	SGN	ABS	INT	FRAC	MOD	View	RAD

1

2.5 Types of programs

The MANUALplus supports the following programs/contours:

- **Teach-In programs** (cycle programs) are used in the **Teach-In** submode.
- 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 in **Machine** mode or during the Teach-In submode. The extension depends on the contour described.

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

Program type	Folder	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"

ſ

2.6 The error messages

Display of errors

The MANUALplus 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 MANUALplus automatically opens the error window. You cannot remove such an error. Shut down the system and restart the MANUALplus.

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 MANUALplus opens the error window and displays all accumulated error messages.

Closing the error window



ERR

Press the END soft key—or

Press the ERR key. The MANUALplus closes the error window.

Detailed error messages

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

Information on error causes and remedies:

▶ Open the error window.



Position the cursor on the error message and press the soft key. The MANUALplus opens the window with information on the error cause and corrective action.

Leave Info: Press the **MORE INFO** soft key again

lumber	Type Text					A	
303-0182	Residual A edge geo <==> N 1	. material due metry! .57 G810 NS4	to cutting NE8 P4 I0.5 H	(8.2 H0 Q0 V0	D1 00		
							=

Soft key INTERNAL INFO

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

▶ Open the error window.

INTERNAL INFO Position the cursor on the error message and press the soft key. The MANUALplus opens the window with internal information about the error.

Leave Details: Press the INTERNAL INFO 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.



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



▶ 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

The MANUALplus stores errors and important events (e.g. system startup) in an error log file. The capacity of error logs 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. 5 log files are available.

Open the error window.

LOG FILES	Press the LOG FILES soft key.
ERROR LOG	▶ Open protocol
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

The MANUALplus stores keystrokes and important events (e.g. system startup) in the keystroke log file. The capacity of keystroke logs 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 error window.



The MANUALplus 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.

Save service files

If necessary, you can save the "Current status of the MANUALplus," 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 600.

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 MANUALplus 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 FILES soft key.



Press the SAVE 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 71).

The **TURNguide** context-sensitive help system includes the user documentation in HTML format. The TURNguide is called with the Info key, and the control 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 DIN (ISO) programming (BHBsmartturn.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 to the right 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 	ENT
 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	È Ŷ
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.	1700 - 100 -
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 arrow 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 all of 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 pressing the keys), 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 languages are at:

- Documentation and information
- User documentation
- Product, e.g. MANUALplus620 CNC PILOT 620/640MANUALplus
- NC software number, e.g. 68894x-03
- Download the compressed CHM file in the desired language and extract
- Move the extracted 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 TNCremo to transfer the .chm files to the control, in the connection configuration on the mode form you must select the third option in the transfer area in binary mode.

Language	Directory in TNCremo
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\p1
Hungarian	TNC:\tncguide\hu
Russian	TNC:\tncguide\ru
Chinese (simplified)	TNC:\tncguide\zh
Chinese (traditional)	TNC:\tncguide\zh-tw
Slovenian	TNC:\tncguide\sl

Language	Directory in TNCremo
Norwegian	TNC:\tncguide\no
Slovak	TNC:\tncguide\sk
Korean	TNC:\tncguide\kr
Turkish	TNC:\tncguide\tr
Romanian	TNC:\tncguide\ro

i
2.8 The DataPilot programming station

Application

Intended for use with the CNC PILOT 640 and

MANUALplus 620 controls, respectively, the PC programs DataPilot CP 640 and DataPilot MP 620 enable you to create NC programs on a PC, test them before execution, transfer them to the control, and archive them after production is concluded.

DataPilot can be used for shop-floor programming as well as for office and off-location program creation and testing. Its practicality and wide range of functions also make DataPilot ideal for training purposes.

Operation

Keyboard

DataPilot is operated through function keys and numerical keys of the PC keyboard.



For more information about installation and operation, please refer to the DataPilot installation and operating instructions.



i





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 semiautomatically.
- **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.

With MANUALplus, you can produce a part in the usual manner by moving the axes with the handwheels and jog controls, just like on a conventional lathe. In most cases, however, it is much more convenient to use the cycles offered by MANUALplus.

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 the **Machine** mode of operation, cycles are **not saved**. 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 **Teach-in 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 378).

You write **smart.Turn** and **DIN programs** in the **smart.Turn** mode of operation. The DIN programming feature provides you 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

DIN subprograms 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 optimize or complete the NC program following conversion into DIN format

NC program following conversion into DIN format.

3.2 Switch-on / Switch-off

Switch-on

The MANUALplus 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 62).



After system start, the MANUALplus 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 MANUALplus 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.

REFEREN	CE RUN
z	Press the Z reference soft key
X	Press the X reference soft key
All	Or press the A11 soft key
	Press Cycle start for the control to traverse the reference marks

The MANUALplus activates the position display and switches to the $\ensuremath{\textbf{main menu}}$.



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



→ Mach	ine	sm:	art.Turn	Tool edito	r	B	
X®	23,405	∆X	W	0.000	ID		
Z	31,703	ΔZ	🔀 C 1		T	0 X 0.0000 Z 0.0000	₽
Y	0.000	ΔY	C 4		₩ ⊕	0.000	
🛱 1 🔯	0.0 mn/nin 0.0 mn/nin	I) 1 0	300.0 m/nin S 1 D 30	00 150 990 U/min 03	1 F 100	S ₁ 100%	
					s ₽⊤ ⊐⊅ ////F	-	END PGM
				⇒ ^^	MDI- cycles		MOB
						DIN	
				Single cut	s		07:30
OFF	Teach-i	.n	Cyc.prog. -> DIN		Program run	Machine reference	

ſ

3.3 Machine data

Input of machine data

In **Machine** 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 the TSF dialog box you also define the "maximum speed", the "stopping angle" and the workpiece material.

Cycle parameters

- T Tool pocket
- ID Tool ID number
- F Feed per revolution/feed per minute
- S Cutting speed/constant speed
- D Max. shaft speed
- A Stopping angle
- WS Material

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.

With activated **Feed per minute** soft key the F value in [mm/min] is evaluated.

With activated $\ensuremath{\textbf{Constant speed}}$ soft key the S value in [rev/min] is evaluated.

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.





→ Machine	smart.Ti	urn	🏷 Tool edi	tor		
X 300.000	B O	.000 🛒 👁	0.000 f	-⊈T	1 × 0.0000 z 0.0000	
2 450,000 V 0.000	ω U			л.	0.000	• •
S1 1 0 0.000 mm/1 0.0 mm/nin	I) 1 - 1500	.0 n/nin 20.0 1 2	0 1500.0 m/r 0.0 U/r		S ₁ 100%	
			റട	07	m.F	
				0Ľ		(FL.202202)
			0F	0F		T
	1		Tool cha	inge		12:54
						Back

3.3 Machine data

Machine-dependent variants of the TSF dialog

Machine operating panel with spindle change key:

If the manufacturer has equipped your machine operating panel with a spindle change key, specify per key for which spindles the entries for S, D and A apply. The SP field displays the number of the selected spindle in the TSF dialog.



In the machine parameter **Separate dialogs for tool change, speed and feed rate** (604906) you can define how you would like the TSF dialog to be displayed on machines with turrets:

- TSF dialog box with input of all cutting data
- Separate dialogs for T, S and F

For machines with tool magazines, separate dialogs are automatically available in the TSF menu.

Machine operating panel without spindle change key:

For machines with only one main spindle the entries S, D and A always apply to the main spindle.

For machines with one main spindle and one tool spindle, the entries, depending on the inserted tool, apply to the main or tool spindle:

- Without driven tool: The parameters S, D and A apply to the main spindle
- With driven tool: The parameters S, D and A apply to the selected spindle

Machine with opposing spindle and/or B axis:

Depending on the machine construction, the TSF dialog may contain further information for controlling an opposing spindle and/or B axis.

Additional cycle parameter with opposing spindle:

WP: Number of spindle

Additional cycle parameters with B axis:

- BW: Angle of B axis
- CW: Reverse the tool
- HC: Block brake
- DF: Auxiliary function

TSF entry via a form



For machines with tool magazines, separate dialogs are automatically available.

Soft keys for "Set T, S, F"

Tool correct.	See "Tool compensation" on page 116.
Measure tool	See "Touch off" on page 113.
Tool list	Call the tool list. Transfer of T number from the tool list: See "Pocket table setup" on page 89.
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)

ENTER THE TOOL DATA AND TECHNOLOGY DATA

of operation)

Ľ,
g
σ
e
•
0
ā
2
Ω.
~
(7)

ß

Conclude data input



Save

s 🖬 T

⊐¢ M\F

Define the parameters

Caution. Depending on the machine, the input of data in the T dialog might cause the turret to turn.

Select Set TSF (can only be selected in Machine mode

TSF entry via separate forms

ENTER THE T	TOOL DATA OR TECHNOLOGY DATA
s ₽⊤ ⊐p mrF	Select Set TSF (can only be selected in Machine mode of operation)
₽	Select T for tool change
ດ s ⊐	Select S for setting the spindle speed
mF ⊟	Select F for setting the feed rate
Enter the para	imeters in the submenu
Save	Conclude data input



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

i

Machine with opposing spindle

Selecting the workpiece spindle (machine-dependent)

If your machine is equipped with a workpiece spindle, the WP parameter is shown in the TSF dialog.

Cycle parameter:

■ WP: Number of spindle

The WP parameter allows you to select the workpiece spindle for machining in **Teach-in** submode and with the MDI cycles from **Machine** mode.

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**

Machine with B axis

Machines with B axes allow the tool carrier to be tilted for flexible use of tools during turning and milling. By tilting the B axis and rotating the tool you can bring it into positions that enable you to use one and the same tool to machine in the longitudinal and transverse (or radial and axial) directions on the main and opposing spindles. In this way, you need fewer tools and fewer tool changes.

Tool data: All tools are described in the tool database by specifying the X, Z and Y dimensions as well as the compensation values. These dimensions and tool orientation must be entered referenced to the **tilt angle B=0°** (reference position).

Cycle parameter:

- BW: Angle of B axis
- CW: Reverse the tool
- HC: Block brake
- DF: Auxiliary function

The BW and CW parameter settings are saved in the Teach-In and MDI cycles and displayed in the corresponding cycle form.



Caution. Depending on the machine, this operation might cause the tool carrier to move, tilting of the B axis or rotation of the tool.

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.	Χ	57.496
Handwheel active	Clamping active	X**
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. 	C	21.296
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.	Х А	11.085
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.	C 2	352.080
Distance-to-go display X, Y, Z, W: The distance remaining from the current position to the target position of the active traversing command	۵X	-14.012
Distance-to-go and protection zone status: Distance-to-go display and display of status of protective zone monitoring	ΔZ	×
Protection zone monitoring active	Protection zone not active	e monitoring
Position display for four axes: Display of position values for up to four axes. The displayed axes depend on the machine configuration.	X 30.00 Z 18.50	00 C 00

i

Elements of machine data display	
T number display	- X 0.5500
T number of the inserted tool	5 ₂ 0.6600
lool compensation values	
For all of the T displays:	
 T highlighted in color indicates a driven tool T number or ID highlighted in color indicates a mirrored tool holder. 	
T number of 1D 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	N 01000 E 01000
T ID display without compensation values	
■ ID of the inserted tool	Stechwerkzeug222
Tool compensation	- X 0.2200 V 0.0000
Special compensation only for recessing tools or button tools	D Z 5,1000 S 5,1000
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 correction	► X 0,5000
Compensation values in gray means D compensation is not active	U_{201}^{901} Z 0.3000
Compensation values in black means D compensation is active	
Tool life information	- MT BT
■ "T": Black=global tool life monitoring on; white=global tool life monitoring off	MZ BZ
MT, RT active: Monitoring according to tool life	
MZ, RZ active: Monitoring according to part quantity	
All fields empty: 1001 Without tool-life monitoring	
Slide display and cycle status	100%
Upper field: Setting of the override control	🖂 📕 💶 10394.1 mm/min
Lower field with white background: Actual feed rate	
Slide display and cycle status	→ 1 → 6.789 mm/1
Upper field: Programmed feed rate	5,779 mm/1

(

1

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 88.
- 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.
- During synchronous operation, the spindle symbol is highlighted in color for both the master spindle and the slave spindle

 Spindle display with spindle number, gear range and spindle status Upper field: Programmed speed Lower field: Actual speed or spindle position 	107.0 1/min 107.0 1/min
 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 	100% 150.0 m/min 107.0 1/min
Override display of the active spindle	4 F 100% c 4 400%
 F: Feed rate R: Rapid traverse S: Spindle 	I R 100% SI 100%









Elements of machine data display	
Display of quantity: The quantity is incremented after each M30, M99 or M18 programmed counter pulse.	50 P 2
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 50 t 00:00:28
MP: Default unit quantity	✓ P 2 2t 00:06:57
P: Number of finished parts	
t: Run time of the current program	
Sum t: Total time	
Display of skip levels and M01 conditional stop	
Defined skip levels (upper row) and set/activated skip levels (lower row)	2)123456789 🜔
Setting for M01: M01 is not executed in Continuous Run mode (yellow display)	
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	+ + + 0.000 ++-
Active zero point shift of the configured RSM axis	
	┣ 1000.000
B-axis display: The information displayed about the status of the tilted plane varies depending on the setting of the machine parameters.	D.000
Programmed angular value of the B axis	
Display of the current values I, K, U and W	
■ I: Plane reference in X	■ I 0.000 U 0.000
■ K: Plane reference in Z	K 0.000 W 0.000
U: Shift in X	
W: Shift in Z	



HEIDENHAIN MANUALplus 620

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

Cycle symbols

Status "Cycle ON" Cycle or program execution is active. ŢŢ]

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 MANUALplus 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	МЗ
Direction of spindle rotation M4	M4
Spindle stopped	0
Spindle position-controlled (M19)	M 19
C axis on spindle motor is active	С
Spindle designations	

Spindle designations			
Main spindle	Н	0	1
Driven tool	1	1	2

3.4 Pocket table setup

Tool data such as length and radius, as well as further tool-specific information required by the control for executing diverse functions, are saved in the **"toolturn.htt" tool table** (in the TNC:\table\ directory). In the control this tool table is designated a **tool list**.

Tools equipped in your tool carriers are saved in the **"ToolAllo.tch" pocket table** (in the TNC:\table\ directory). This pocket table is made available according to the machine and correspondingly designated a **turret list** or **magazine list**.

Machine with a tool holder (multifix)

You do not need to manage a pocket table for machines with multifix tool holders because the tool carrier only has one pocket.

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

Machine with turret

Press the turret list soft key to open the list of the momentary turret assignment. There is a place in the table for every tool holder of the turret (and possibly multifix). During setup, each tool (ID number) is assigned to a tool holder. In the turret list, multipoint tools are displayed with all cutting edges.

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

T turret pocket number

Tool ID (name) is entered automatically.

Cursor in the TSF menu in the T input field:

Turret list Open the Turret list. After opening, the turret list can be edited.

Cursor in the TSF menu in the ID input field:



Open the **Tool list**. The tool list is also opened in addition to the turret list. The turret can be populated with tools from the list.

In the Teach-in cycle, you program the turret pocket as a **T number** The **tool ID number** is automatically entered under "ID" with a populated pocket.



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

Machine with magazine

Press the magazine list soft key to open the list of the momentary tool carrier assignment. There is a place in the table for every tool holder. During setup, each tool (ID number) is assigned to a tool holder.

On machines with a magazine the tool is also loaded via the **TSF** menu.

T turret pocket number: Always T1

Tool ID (name) is entered automatically.



Open the magazine list

The tools used are listed in the magazine list. The magazine can be loaded and unloaded via the **TSF menu**.



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

Filling the turret list with the tool list

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 submode.

Look at the entries in the tool list in order to move entries from the list into the turret assignment list. The MANUALplus displays the tool list in the lower area of the screen. The cursor keys are active in this list. You can move the cursor directly to a tool ID number by entering the first few letters or digits of the ID number.



Initiate the cycle dialog

		Selecting and sorting entries in the tool database					
Tool list	Press the Tool list soft key to activate the turret assignment list and the tool list.	Tool type	The MANUALplus opens the soft-key menu to select the desired tool type.				
Adapt the	turret assignment	Internet to select the desired toor type. Int Immenu to select the desired toor type. Int Immenu to select the desired toor type. Immenu to select the soft the test toor type. Immenu to select the desired toor type. Immenu to select the toor type. Immenu to select the desired toor type. Immenu to selec					
TRANSFE	RRING TOOLS FROM THE DATABASE	View	The MANUALplus opens the soft-key menu containing various sorting options.				
Pocket down	Select the position in the turret assignment list.	Sort by ID / Type	Sorts the tools in the displayed list as desired according to:				
Pocket up			Tool typeTool IDTool orientation				
Select and	I sort the entries in the tool database (see soft-key table at		Each time the soft key is pressed it changes to the next sorting mode.				
right)		Reverse sorting	Switches between ascending and descending sorting.				
Use the cu	ursor keys to select the entry in the tool database.	Edit the tool	Not active here				
Load tool	Load the selected tool into the turret assignment list.	Back	Closes the tool list.				

→ Machine

Turret assign

T # ID number

1 801

3

4 5 6

10 11 Tool sel

Filter ID number 601 601-capto 601AP0 601AP1 601AP1 602

Pocket

Filter All types

ID-number clipboard

📀 smart.Turn

TO Designation

0.80

0.80 93.0

0.80 93.0

0.80

0.80

9.80

Tool type

93.0

93.0 95 A

More filters

1 Sch

TO Designation
 J Schruppen Aussen
 J Schruppen Aussen

💐 1 Schruppen Aussen

J_ 1 Schruppen Aussen

🖉 1 Schrunnen Aussei

Editing

Pocket down

Load tool

Back

View

3.4 Pocket ta<mark>ble</mark> setup

Editing 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 submode. 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 520).

SET UP TURRE	T LIST
s ₽⊤ ⊐⊅ MF	Select Set TSF (can only be selected in Machine mode of operation)



Paste entry from clipboard

Switch to next menu

Reset tool life

Cut out entry and save it in the clipboard

Delete the complete turret list

Back by one menu level

Show entries in the tool database

Soft keys in turret list

⊁

25

5×

Tool list

Special Functions

DELETE

ALL

Schneiden neu setzen

Delete entry

Initiate the cycle dialog



3.4 Pocket t<mark>able</mark> setup

Press the **Turret list** soft key to activate the turret assignment list

Use the arrow keys to select a turret pocket

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

Enter the tool ID number directly

ENTER THE TOOL ID NUMBER DIRECTLY



Press the **ENT** key to activate direct input.

Enter the tool ID number



Editing the magazine list

The magazine list for machines with tool magazines notates the momentary filling of the existing magazines and tool carriers in the working space. The magazine list can be edited via the **TSF menu**.



Select the tool

Load tool	Select the tool with the Load tool soft key
Save	Transfer the tool to the magazine list with the Save soft key
TOOL CHANG	F

TOOL CHANGE	
s H⊤ ⇒m⊧ ⇔	Select TSF (can only be selected in the Machine mode of operation)
0 ²	Select Tool change
Magazine list	Select the tool with the Magazine list soft key
Enter the tool II	D

Save Change the tool with the Save soft key

Ŧ	Mach	ine	J	🔶 sı	art.Turn		A (Tool ed	itor		8		
х		23.420	B		0.000	ľ	•	0.000	rafi ∎		O z	0.0000	
Z		31.710	W		0.000	È	6		ID			Θ	0 0
Y		0.000	C 1			C 4			Q 1		0	.000	-
<u>\$1</u>	1 🙋	0.200 mm/1 0.0 mm/nin	1	1 0	1500.0 m/min 360.0 °	1	2 0	1500.0 m/ 0.0	ron 1	F 100 R 100	ξ [*] S ₁	100%	-
Sel	ect m	agazine pocke	t fo	r load	ing	_	-					100	
		ID number		TO	Designation	_	RS/DV	EW/BW/AZ	SW/SB/	s U HG Cut	ting m	1138 nat. –	
12	1												
12	3												
12	4												
12 12	6 7												
12	8												
12	10												
12 12	11 12												
12	13	1	- 1		1	_	1		1			-	12:58
								Switch			Loa	d the	Cancel

HEIDENHAIN MANUALplus 620



RETURN TOOL	TO MAGAZINE
s ⊒t⊤ ⇒ mF	Select Set TSF (can only be selected in Machine mode of operation)
Q <i>2</i>	Select Return tool to the magazine
Save	Return the tool to the magazine with the Save soft key
UNLOAD MAGAZ	INE
s ₩T ⇒ MrF	Select Set TSF (can only be selected in Machine mode of operation)
0 ^µ	Select Unload magazine
Select the tool	
Unload	Press the Unload soft key
Save	Remove the tool from the magazine list with the Save soft key

i



Tool call



This function is also available on machines with a tool magazine. The control uses the magazine list instead of the turret list.

Parameters for calling the tool

T is the identification letter for the tool holder.



The designations of the tool pockets are fixed by the machine tool builder. Each tool holder of a tool carrier in the working space has a unique **T number**.

ID designates the tool ID number.



The ID number of a tool is defined when you create a tool in the tool editor. Each tool has a unique **ID**.

Variants of the tool call

- One tool holder (e.g. multifix): The tool is called by "ID". The pocket number "T" is always 1. No turret list is kept.
- More than one tool holder (e.g. turret): 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 the turret list, multipoint tools are displayed with all cutting edges.

You enter the parameters for calling the tool in the TSF dialog in the **Machine** mode of operation. In the **Teach-in** submode and **smart.Turn** mode of operation, "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. The existing turret list will be overwritten.



Driven tools

- 3.4 Pocket t<mark>able</mark> setup
- 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.

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 the control 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** submode 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.



Tool life monitoring

If desired, you can have the MANUALplus 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 MANUALplus sets the diagnostic bit 1. This causes an error message to be issued the next time the tool is called. If no replacement tool is available, the program will be stopped.

- For Teach-in programs, the **simple tool life monitoring** is available. Here the MANUALplus 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 and the tool life monitoring with replacement tools option. If you use replacement tools, the MANUALplus automatically inserts the "sister tool" as soon as the tool is worn-out. The MANUALplus 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/Settings for automatic operation/Tool life management."

The MANUALplus manages the type of monitoring, the "tool life/ remaining tool life" and the "maximum number of pieces/remaining number of pieces" in the diagnostic bits of the tool data. You can edit and display the diagnostic bits and the tool life in the **Tool editor** (see "Editing tool-life data" on page 515).

You can define replacement tools when setting up the tool carrier in **smart.Turn**. 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 editor** mode when you replace the insert of a tool.

→ Progra	m run	smart.T	urn	Tool	editor				
The servic	e life of tool	('001') has	expired!						
X S	300.000 B	0	000 脂 🗢	0.0	00 <u>⊢</u> ∏ 00		1 [×]	0.0000	
Z Z	450.000 W	0	.000 🛋		ID			001	5
2	0.000 0		C 4		Q ₁		0	.000	
🛱 1 🙋	0.000 nn/1 0.0 nn/nin 1	10-	0.0 rpn 1 2 0	θ.	8.8 rpm 1	F 100 R 100]% S₁	100%	
Prog	Sequence 👖 D	isplay		ICP o	out longitu	.bu			
TNC:\nc_pr	og\gtz\Huelse			X	23.405	Z	31.76	925	
%Huelse.gm	z			FK [Huelse				
N1 T1 ID"0	01" ICP-Zerspa	nen längs %Hu	uelse G14 Q0	P	5	н	0: Wi	th €	
				I		К		_	
				Εſ		0	0: No		
				SX		SZ	-27		[],acceq
				G47	2				
				т [1	G14	1 0: Si	imult-	
				ID [901		-		
				s [200	F	0.35		
								1/3	12:56
	Contin.	Single	Tool/Add.		Ba	se			Back

Resetting the	tool life in the turret list
RESET TOOL	LIFE
s ₽⊤ ⊐ĵ MF	Select Set TSF (can only be selected in Machine mode of operation)
Turret list	Open the turret list
Special Functions	Press the Special Functions soft key
Schneiden neu setzen	Press the Set new cutting edge soft key
YES	Answer Yes to the confirmation prompt
Back	Press the Back soft key

1

Resetting the tool life in the magazine list

RESET TOOL L	.IFE
s ₽т ⇒ MrF	Select Set TSF (can only be selected in Machine mode of operation)
07	Select Tool change
Magazine list	Open the magazine list
Select the tool	
Edit the tool	Select the Edit tool soft key
Nev tooth	Press the New cutting edge soft key
Back	Press the Back soft key



3.5 Machine setup

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

- Setting the axis values (defining workpiece zero point)
 - Machine reference (axis reference run)
- Set the protection zone
- Defining the tool change position
- Setting C-axis values
- Defining machine dimensions
- Display 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 zero point can also be set in 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.

WORKPIECE DATUM SETTING



Select Setting up



Select Set axis values

Touch the workpiece zero point (end face)



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 MANUALplus 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



Defining offsets

Before using zero point shifts with G53, G54 and G55, you need to define the offset values in setup mode.





The MANUALplus saves the values to a table. In this way, you can activate the offsets in the program by entering the respective G codes.

i

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
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 the All soft key
	Press Cycle start for the control to traverse the reference marks
The MANUAL	plus refreshes the position display.





Setting the protection zone

With active protection zone monitoring, the MANUALplus checks for every movement whether the **protection zone in –Z direction** would be violated. 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).



Select Setting up



Protection zone status

Protection zone monitoring active



Protection zone monitoring not active



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

Select Set 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")



Protectn off Switch off protection zone monitoring



Protection zone monitoring is not active if the Set protection zone dialog box is open.

In DIN programming, you deactivate protection zone monitoring with G60 Q1 and reactivate it with G60.

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" function enables you to define a zero point shift for the workpiece spindle:

- CN: Position value of the workpiece spindle (display)
- C: Datum shift of the C-axis
- CM: Measuring point coordinate (set momentary position to defined value)

DEFINING THE ZERO POINT OF THE C AXIS



Select Setting up



Select Set C axis values

Positioning the C axis



Define the position as the ${\bf C}$ axis zero point

Set momentary position to defined value

Absolute	
Apsolute	
offset	

Press the **Absolute offset** soft key and enter the value in the **CM** input field

Enter 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)

3.5 Machine setup

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 n	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



Machine mode of operation

i
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

₽ L 	Select Setting up
()) 0+1	Select touch probe
i I I I I I	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.
Back	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



Displaying operating times

3.5 Mac<mark>hin</mark>e setup

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

i

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 $\ensuremath{\textit{Adjust system time}}$ 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.

SYSTEM TIME,	SETTING
	Select Setting up
	Select Service
	Select Adjust system time
Select Synchron	ize the time over NTP server (if available)
Select Set the	time manually
Select Date	
Select Time	
Select Time zon	e
Press the 0K sof	it key

3.6 Tool measurement

The MANUALplus 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 ₽́т the TSF dialog box 🗐 MAF M 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 🗐 /// F Activate Measure tool Measure tool Touch the face with the tool Enter the value 0 for the **measuring point coordinate** Transfer Z (workpiece zero point) and save it Ζ Turn a measuring diameter Enter the diameter value as **measuring point** Transfer coordinate X and save it Х For turning tools, enter the cutting edge radius and Save load it into the tool table radius







1

Touch probe (tool touch probe)



Optical gauge









1

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.

You can set tool compensations with the handwheel or enter them into a dialog field.

SETTING	TOOL COMPENSATION WITH THE HANDWHEEL
s ₽́⊤ ⊐́р № F	Select Set TSF (can only be selected in Machine mode of operation)
Tool correct.	Press the Tool correct. soft key

Set by handwheel	Press the Handwheel correct. soft key if required
X compns. of tool	Press the X offset for tool (or Z offset) soft key 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

Machine mode of operation



DEFINING	TOOL COMPENSATION
s ∎⊤ ⊐⊅ mF	Select Set TSF (can only be selected in Machine mode of operation)
Tool correct.	Press the Tool correct. soft key
Set comp- ensation	Press the Set correct. soft key if required
Enter the co	mpensation value dx (or dz, dy)

SaveTransfer the compensation value to the tool tableThe T display shows the new compensation valueThe distance-to-go display is canceled

DELETING	TOOL COMPENSATION VALUES
s ₽⊤ ⊐D MF	Select Set TSF (can only be selected in Machine mode of operation)
Tool correct.	Press the Tool correct. soft key
DELETE	Press the Delete soft key
Delete X compens.	Delete compensation value in X (or Z)

1

3.7 Manual Operation

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 MANUALplus is in **Machine** mode. This mode remains active until you select **Teachin** or the **Program Run** submode. **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 ${\bf T}$ 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 your machine manual

Machine mode of operation



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 the Machine operating 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 submode

Teach-In mode

In the **Teach-in mode** you machine a workpiece step by step with the help of Teach-in cycles. The MANUALplus "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** submode can be switched on with the **Teach-in** 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, MANUALplus displays the cycle parameters.

The cycle block contains:

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

➡ Teach-in	🚯 smart.Tu	urn	🍴 Tool edi	itor		
X [®] 23.40	5 △X	W	0.00	00 ID		
Z 31.70	3 △Z	🛛 🔀 C 1		T	0 X 0.0000 Z 0.0000	₹ ↓
Y 0.00) ∆Y 1]) 1 0	C ₄ .0 m/nin 116.8 S ₁	50 100 150 3090 U/min	₩ ♥ •* 1 F 100 R 100	0.000 \$ S1 100%	F.
TNC:\Project\BHB_KA %pgm02.gmz [] N1 Bar/tube blank	PITEL3\gtz\pgm02		X 52 FK 754	longitud. Z	2	END PGM → ··· OFF OFF ON
N2 T3 TD*020" Cut transv. G14 00 N3 T1 TD*081" TCP cut longitud. %754 G14 00 N4 T4 TD*081AP1" TCP finish longit. %754 G14 00 I 0.5					0: With er 0.1 0: No r	M87
			SX G47 1	SZ		/ M98
			ID 001 S 200	- G14	0.35	
					1/2	07:32
Program list Renum	change text	Erase cycle	Copy cycle	Edit cycle	Add cycle	Back



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** submode or switch off the machine, the teach-in program is kept.

When you call an ICP cycle, the control displays a soft key for switching to the ICP contour editor (see "ICP editor submode in cycle mode" on page 381).

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 submode

Loading a program

In the **Program run** submode you use teach-in programs, DIN programs or automatic jobs for parts production. You cannot change the programs in this mode. The **Simulation** submode, however, allows you to check the programs **before** you run them. The MANUALplus 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 saved as DIN programs (*.nc). Automatic jobs (*.job) are also generated in the **smart.Turn** mode of operation.

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



🎦 Tool editor

🚯 smart.Turn

→ Program run

LOAD THE TEACH-IN PROGRAM OR NC 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 $\ensuremath{\text{Program}}$ $\ensuremath{\text{Run}}$ submode can be switched on with the soft key and is displayed in the header.

If you press **Program Run**, the MANUALplus 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 139).



Programs selected in the $\ensuremath{\text{Program}}$ $\ensuremath{\text{Run}}$ submode are protected from deletion.

To release the file for deletion, exit program block display by pressing the Back soft key.



Comparing a tool list

While a program is being loaded, the MANUALplus 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.



This function is also available on machines with a tool magazine. The control uses the magazine list instead of the turret list.

Before executing a program

Faulty programs

The MANUALplus 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 MANUALplus displays the Teach-in/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 the program run with the **Simulation** submode (see "The submode Simulation" on page 486).



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 MANUALplus 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 MANUALplus 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 MANUALplus 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 menu Program Run > No. of pieces, 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.

In the menu Program Run > Skip levels, the skip levels defined in the program can be set/activated with the NR parameter. Before you can set/activate skip levels, you need to define them in the program (see the smart.Turn and DIN Programming User's Manual).

If you enter the value 2 in the NR parameter and press the Save soft key, the control sets/activates the skip level 2 and refreshes the display field accordingly (see "Machine data display" on page 84). The next time the program is run, the NC blocks defined by the set/active skip level will not be executed by the control.

You can set/activate more than one skip level at a time by entering a string of numerals in the NR parameter. The entry "159" sets/activates the skip levels 1, 5 and 9.

To deactivate the skip levels, program the NR parameter without a value and press Save.

When setting/activating skip levels during program run, please remember that the control will respond with a delay due to the block scan.

Soft kevs Select the Teach-in program or Program smart.Turn program list Teach-in program: Contin. run 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. **On:** The simulation stops after each Single path of traverse (basic block). Start the block next path: Cvcle start. (Recommendation: Single block should be used together with the basic-block display.) Off: Cycles / DIN commands are executed without any interruption. Input of tool compensation values or Tool/Add. additive correction values, see "Entering correct. compensation values during program run" on page 127 Activate the Simulation submode **On**: The traversing and switching Base commands are shown in DIN format blocks (basis blocks).

- Off: Teach-in or DIN program is
displayed.Program
startThe cursor returns to the first block of the
Teach-in program or DIN program.
- Start blck search Enables entering into an NC program at a selected point.

Automatic job

In the **Program Run** submode the control can execute several main programs in succession without you needing to reselect and start these programs. For this purpose, create a program list (automatic job) in **smart.Turn** mode that is then processed in the **Program Run** submode.

Specify a quantity for each main program in the list that defines how often this program is carried out before the next NC program is started. With job selection, you can specify an NC program from which the job is to be processed.

specify an NC program from which the job is to be processed.



If the program list should be processed without intervention, the processed main programs must be exited with M99.

 $\ensuremath{\mathsf{M30}}$ stops the automatic job. The NC Start key continues the automatic job.

Selecting job

- Select the "Prog" menu item
- Select the "Job selection" menu item
- Select "Automatic job"

Press the Open soft key

0pen

- Select the start program with the cursor if necessary
- Confirm with the "Load job" soft key



If you have already machined workpieces you can correct the number of repeats in: Process menu > Quantity menu item.

Changing job display

The programs of the selected job are displayed and the cursor is positioned at the selected start program.

In the menu item Display > Job list off, you can switch from job display to NC program display to work with the mid-program startup block scan.

You can also activate basic block display by soft key.

3.9 Program run <mark>su</mark>bmode

Entering compensation values during program run

Tool compensation



To delete a compensation, enter the current compensation value with the opposite algebraic sign.

→ Progr	ram run	🔶 sm	art.Turn		A Tool edito	r)	8	
x	300.000	B	0.000	*	0.000	T :	L X 8.0000 Z 8.0000	-
Z	450.000	W	0.000			ID	001	
Y	0.000	C ₁		C 4		Q ₁	0.000	
출1 🙋	0.000 mn/1 0.0 mn/nin	I) 1 0	0.0 rpm 20.0 °	I) 2 🤇	0.0 rpm 0.0 U/nin	1 F 1008	S ₁ 100%	
Prog	Sequence	Display			Set tool c	orrection		
					T 1]		
					ID 001			
					DX 0	dx		
					DZ 0	dz		
					DY 0	dy		B aaraa
					BW 0	CW	9:No 💌	100000
					KM Werks	ück in Haup	tspindel	
					G16			T
					ID number		1/1	12:55
Tool	Additiv	e Magaz	zine		Delete	Delete	Sava	Rack

Additive compensation

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

ENTERIN	G ADDITIVE COMPENSATION			
Tool/Add. correct.	Activate the additive compensation			
Additive correct.				
Enter the number of the additive compensation				
Enter the	compensation values			

➡ Program	run	sm:	art.Turn		A (Tool edi	tor	8	
X [®] 2	23.405	△X		U		0.00	0 10		
Z 3	31,703	ΔZ	×	C ₁			T	0 X 0.0000 Z 0.0000	ŦŢ
Y ∰1 🔯	0.000 0.0 mm/min 0.0 mm/min	∆Y ⊐) 1 ○	0.0 rpm 115.8 °		. so 	100 150 0 U/min	₩ ♥ • 1 F 100 R 100	0.000 ∰ \$ S₁ 100%	
Prog Project	Gequence	Display	n02			Set addi D 901 X 0.4	tive correct	ion	END PGM → ····································
N1 Bar/tube N2 T3 ID"626 N3 T1 ID"601 N4 T4 ID"601	blank)" Cut tran L" ICP cut LAP1" ICP (nsv. G14 C longitud. finish long	0 %754 G14 C mit. %754 G	10		Z 0.2 Y 0	dz dy		M07
		-							MOB
ļ		_				Add. cor	rection numb	er 1/1	07:33
Tool correct.	Additiv correct	e Turr lis	et t		D	elete all	Delete	Save	Back

Save

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

READING ADDITIVE COMPENSATION



Activate the additive compensation

Additive correct.

Enter the number of the additive compensation

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



DELETING ADDITIVE COMPENSATION



Enter the number of the additive compensation

DELETE	
	_
DELETE	

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

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

- Entered values are **added** to the existing compensation values and are immediately effective in the display. The control moves in the compensation direction by the compensation value in the following traverse block.
- 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 MANUALplus 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** submode 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 MANUALplus returns to the programmed feed rates and spindle speeds.



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

3.10 Load monitoring (option)



The control must be specially prepared by the machine tool builder for the use of the Load Monitoring option.

The following steps are required before you can use the load monitoring feature in the Program Run submode:

- Define the respective machine parameters in the System section (see "List of machine parameters", page 551)
- In the smart. Turn operating mode, define the type of load monitoring with G996 and the monitoring zone with G995 in your program (see smart. Turn and DIN Programming User's Manual)

During a machining operation with active load monitoring, the control compares the current utilization of the drives specified by G995 with the respective limit values. To calculate the limit values for a monitoring zone defined with G995, the control uses reference values determined during reference machining as well as predefined factors specified in the machine parameters.

If the utilization limit 1 or the total utilization limit is exceeded, the control issues a warning and sets the corresponding diagnostic bits in the **Tool editor** mode to identify the active tool as worn-out.

If the utilization limit 2 is exceeded, the control issues an error message, stops the machining operation and sets the corresponding diagnostic bits in the **Tool editor** mode to identify the active tool as broken.



When tool life monitoring is active, the control will automatically insert a predefined sister tool during the next tool call in order to replace the tool that has been identified as worn-out or broken by the diagnostic bits of the **Tool editor** mode. As an alternative to the automatic evaluation of the diagnostic bits by the tool life monitoring function, you can also evaluate the diagnostic bits in your program.



Keep in mind that load monitoring is not possible for hanging axes without counterweight!



Note that load monitoring only works to a limited extent if the load changes are small. Therefore, monitor the drives that are subjected to significant loads, such as the main spindle. 3.10 Load monitorin<mark>g (</mark>option)

When face turning with a constant surface speed, remember that the load monitoring feature will monitor the spindle up to a maximum of 15 % of the nominal acceleration defined in the machine parameters. Since acceleration increases as a result of the change in rotational speed, the control only monitors the period after the first cut!

The load monitoring function compares current utilization values with maximum limit values. For proper comparison, the utilization values must not be too low. Since the utilization depends on the cutting conditions, the following example is provided for your guidance. It illustrates the required values for the machining of steel:

- Longitudinal turning: Cutting depth > 1 mm
- Recessing: Cutting depth > 1 mm
- Drilling into solid material: Hole diameter > 10 mm

Reference machining

During reference machining, the control determines the maximum utilization and the total utilization for each monitoring zone. The determined values are used as reference values. To calculate the limit values for a monitoring zone, the control uses the determined reference values and the predefined factors specified in the machine parameters.



Perform reference machining under the conditions in which the parts will later be produced, for example, with respect to feed rates, spindle speeds, and the type and quality of the tools you will be using.

REFERENCE MACHINING

Select the Program Run submode and open the NC program

Activate load monitoring: Menu **Program Run** > select menu item **Activate load monitoring**

Select reference machining: Menu **Program Run** > select menu item **Reference machining**—the control shows the title bar with a green background

Start reference machining: Press **NC Start**—the control executes the machining operation and stores the reference data in a separate file. After the successful completion of reference machining, the control displays an information message.



Program M30 or M99 to conclude reference machining. If the program is aborted during machining, no reference data are stored. In this case you need to execute reference machining again.



Reference machining must be repeated if you make any changes to your program; for example:

- Define new zones
- Delete existing zones
- Change zone numbers
- Change, add or remove axes within a zone
- Change feed rates or spindle speeds
- Change tools
- Change cutting depth values



Checking the reference values

After the successful completion of reference machining, check the determined reference values.



The load monitoring function compares current utilization values with limit values. For proper comparison, the reference values for utilization must not be too low. Check the determined values and, if required, remove monitored axes with a utilization rate of less than 5 % from the monitoring zone.

Meanings of the values:

- Utilization: Determined drive torque relative to the rated torque of the drive in [%]
- Total utilization: Sum of the utilization values in the monitoring zone in [%*ms]

OPEN REFERENCE VALUES

Display reference values: Menu **Display** > select menu item **Edit load data**—the control opens the "Set load data" form containing the parameters listed below. In addition, the determined values are displayed as a bar graph.

Parameters

- ZO Number of the monitoring zone
- AX Monitored axis
- CH Selected channel
- T Tool pocket of the active tool in the monitoring zone
- ID Name of the active tool in the monitoring zone
- P Maximum utilization during reference machining
- PA Maximum utilization during the current machining operation
- PG1 Utilization limit 1
- PG2 Utilization limit 2
- W Total utilization during reference machining
- WA Total utilization during the current machining operation
- WGF Factor for the total utilization limit



Bar graph

Thick upper bar (display in %)

Green	Range up to maximum utilization during reference machining (P)

- Yellow Range up to utilization limit 1 (PG1)
- Red Range up to utilization limit 2 (PG2)
- Magenta Maximum utilization during last machining operation (PA)

Thin lower bar (display standardized to reference value 1):

- Green Range up to maximum total utilization during reference machining (W)
- Yellow Range up to total utilization limit (WGF)
- Magenta Maximum total utilization during last machining operation (WA)



After reference machining, the value W equals WA and the value P equals PA. They are used as the reference values for calculating the limit values.

Adapt limit values

After successful reference machining, the control uses the reference values and the predefined factors specified in the machine parameters to calculate the limit values.

You can adapt the calculated limit values as required for the subsequent production.

ADAPT LIMIT VALUES

Display limit values: Menu **Display** > select menu item **Edit load data**—the control opens the "Set load data" form

Check the limit values

If required, adapt the parameters PG1, PG2 or WGF

with the adapted limit values.



Be sure to adapt the correct limit values. First, press the **Next zone** and **Next axis** soft keys to select the form containing the limit values you want to change! As an alternative, you can use the selection lists of the **ZO** and **AX** parameters to select the correct form. Save the changes separately for each axis by pressing the **Save** soft key!

Reference machining does not have to be repeated after adapting the limit values. You can continue production



🚯 smart.Turn → Machine 🏷 Tool editor ## Prog ## Head ## ICP ## Units» ## Go to ## Config ## Misc ## Extras ## Grap /1 N 111 END_OF_UNIT \$787866785 G810 Longitudinal roughing in ICP Overvw. Tool Contour Cycle Global /12 N 158 UNIT ID"G818_ICP" [G818 Longitud /12 N 151 [-(unit ID="G818_ICP" APP="0" X /12 N 152 G8 S380 G95 F0.3 N3 /12 N 154 G8 S320 G95 F0.3 N3 /12 N 155 G8 X52 Z2 /12 N 155 G47 P2 Type of approach APP 0: Simultaneour Approach position X XS 52 Approach position Z 🚽 Tool number T 1 END PGM ···· Feed F 0.3 OF OFF ON Cutting speed S 300 M07 Auxiliary c... FK OFF NS 4 Contour start block no Contour end block no NE 8 MOS 0/ P4 Maximum infeed 10.5 Oversize X K0.2 Oversize Z 1/7 Type of approach TNC:\Project\01_SCREEN\ncps\bar.nc 11:35 Help graphic Proposed technology Turret list Save Cancel



136

Using load monitoring during production



Keep in mind that you cannot adapt the limit values during machining. Adapt the limit values before starting a machining operation.

During **Program Run**, the control monitors the utilization and the total utilization in each interpolator cycle. In parallel with machining, you can display a graph of the current utilization values for all monitored axes of the active zone.

OPEN DIAGRAM DURING MACHINING

Display utilization values: Menu **Display** > select menu item **Edit load data**—the control opens the "Set load data" form and additionally displays the determined values as a bar graph

Display current utilization values: Press the **Display active zone** soft key—the control automatically switches to the current monitoring zone and displays the current utilization values as a bar graph

Bar graph

Thick upper bar (display in %)GreenCurrent utilization (PA)Mark in thick upper bar:GreenCurrent peak value between 0 and limit value 1 (P)YellowCurrent peak value between P and limit value 1 (PG1)RedCurrent peak value between PG1 and limit value 2 (PG1)Thin lower bar (display standardized to reference value 1):GreenCurrent total utilization (WA)YellowCurrent total utilization up to limit value (WGF)



3.11 Graphic simulation

The submode **Simulation** enables you to check the machining sequence, the proportioning of cuts and the finished contour **before** actual machining.

In the **Machine** mode and **Teach-in** submode you check the sequence of a single Teach-in cycle—in the **Program Run** submode you control a complete Teach-in program or DIN program.

A programmed workpiece blank is displayed in the **Simulation** submode. The MANUALplus 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 the **Machine** mode and **Teach-in** submode, the Teach-in cycle you are currently processing is simulated. In the **Program Run** submode the simulation starts from the cursor position. smart.Turn and DIN programs are simulated from the beginning of the program.

For more details on use and operation of the **Simulation** submode refer to the chapter "The submode Simulation" on page 486.



3.12 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.

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

 Soft keys in the program selection dialog

 INTERNAL
 Display the file attributes: size, date, time

INFO	
DIN	Switching between Teach-in and DIN/smart.Turn programs
File manager	Open the program organization soft-key menu (see page 141)
Sorting	Open the sorting functions soft-key menu (see following table)
Project -	Open the project organization soft key menu (see "Project management" on page 142)
Alphabetic keyboard	Open the alphabetic keyboard (see "Alphanumeric keyboard" on page 57)
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.



Program list

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

i

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.

File mana	File manager soft keys						
Paths / files	Alternate between directory and file window						
Cut out	Cut marked file						
Сору	Copy marked file						
Insert	Insert the file stored in the buffer memory						
Rename	Rename marked files						
DELETE	Delete the marked file following confirmation prompt; program block display must not be opened in any operating mode						
Back	Return to program selection dialog						

Other soft keys

	•
INTERNAL INFO	Display details
Mark all	Mark all files
Сору	Update marked program
Write protection	Activate/deactivate write protection for the marked program
Alphabetic keyboard	Open alphabetic keyboard
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				
Selection of standard	Open standard directory				
Selection of PLC:	Open OEM programs				
Select project	Select marked project				
Select std.director y	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.

3.13 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

CONVERS	ION 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 the **Cycle program --> DIN** soft key (program selection menu)

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

Should the MANUALplus 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 must be confirmed with the **Overwrite** soft key. The MANUALplus overwrites the program that is open in the smart.Turn editor.

3.14 Units of measure

The MANUALplus 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			

➡ Machine	•	🚯 smart.Tu	rn	Tool ed	litor	8	
X®	0.9215 4	۲	ω	0.00	00 ID		
z	1.2481 4	2	🔀 C 1		T	0 x 0.00000 z 0.00000	F
Y	0.0000 4	2	C 4		hi ⊕	0.0000	
😫 1 🚺	0.00 in/nin 1 0.00 in/nin 1	10 300.	0 n/nin S 1	50 100 150 D 3000 U/nin	et 1 F 100 R 100	S 100%	
Tool list	1	_			001	01 Have 000	END PGM
Filter A.	LI types	l	DC (D)/ EU	(PLL (A7 CLL (CP (L	JJZI OF	MILMD IA A	🔶 🔶 OFF
A01	10 Des	action	0.016	93 0 80	A Hartmetall	3	OFF ON
001-capto	J 1 TO	ahina	0.032	93.0 80	0 Hartmetall	3	13 M97
001AP0	J 1 roi	Jahina	0.032	93.0 80	0 Hartmetall	3	2
001AP1	J 1 101	Jahina	0.032	93.0 80.	0 Hartmetall	3	OFF
002	J 1 rou	Jahina	0.032	95.0 55.	0 Hartmetall	3	
002-capto	J 1 rot	ghing	0.032	93.0 80.	0 Hartmetall	3	/ / ///
002AP1	J. 1 rou	ghing	0.032	95.0 55.	0 Hartmetall	3	E OFF
003	🖉 1 fir	nishing	0.032	95.0 35.	0 Hartmetall	3	
003-capto	J. 1 rou	ghing	0.032	93.0 80.	0 Hartmetall	0 3	
003-capto_1	🤳 1 rou	ghing	0.032	93.0 80.	0 Hartmetall	1 3	
003-capto 2	J 1 rol	Jahina	0.032	93.0 88	0 Hartmetall	23 1	07:34
New tool	Editing	Сору	Tool type	More filters	Viev	⊁	Other tables

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 551). 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.


Cycle Programming

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 the **Machine** 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 Machine mode)
- Select and program a cycle
- Graphically test the cycle
- Execute the cycle
- Save the cycle (only in Teach-in submode)

Cycle starting point

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

In the **Teach-in** submode you enter the **starting point** as one of the parameters. The MANUALplus 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 circular arrow 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: "Directional dimension"—the algebraic sign has no effect

DIN macros

DIN macros (DIN cycles) are DIN subprograms (see "DIN cycle" on page 374). 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 "The submode Simulation" on page 486).

Contour follow-up in Teach-in submode

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 in the **RG** parameter select "With contour follow-up" (see also "Workpiece blank cycles" on page 155).



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 and centric drilling.

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

- Pressing Cycle start initiates a search for the selected cycle for midprogram 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 cycle. 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 MANUALplus 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

 Press the Goto key to show the alphabetic keyboard

 Press the Goto key to show the alphabetic keyboard

 Enter the comment with the on-screen alphanumeric keyboard



Transfer the comment

Cycle menu

The main menu shows the cycle groups (see table below). Once a cycle group has been selected, the menu items 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 used in the **Teach-in** submode:

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

Cycle groups	Menu item
Blank Defining standard workpiece blanks or workpiece blanks with ICP.	·
Single cuts Positioning in rapid traverse, linear and circular single cuts, chamfers, and rounding arcs.	
Turning cycles, longitudinal/transverse Roughing 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	DIN



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 starting 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

If you have confirmed the cycle with the **Input finished** soft key, a further soft-key row is displayed.

Soft keys	s in cycle programming
Tool correct.	Tool compensation(see page 153)
Single block	Initiate cycle in Program Run, Single Block
Base blocks	Display basis blocks
	Display simulation
Save	Save the cycle (only with Add cycle)
Over- write	Overwrite the cycle (only with Modify cycle)
Back	Return to cycle description

i

Tool compensation in Teach-in submode

You can set tool compensations with the handwheel or enter them into a dialog field.

DEFINING TOOL COMPENSATION

Define the cycle

Input finished	Press the Input finished soft key
Tool correct.	Press the Tool correct. soft key
Set by handwheel	Press the Handwheel correct. soft key
Set comp- ensation	Press the Set correct. soft key
X compns. of tool	Press the X offset for tool (or Z offset) soft key The compensation values that you determine per handwheel are now shown in the distance-to-go display
Enter the comp	pensation value dx (or dz, dy)

Press the **Save** soft key or press **Overwrite**

Т

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 551.

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 551.

Tool change position 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 105). 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 \mathbf{SX} and \mathbf{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 again at the end of the cycle.



4.2 Workpiece blank cycles

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.

➡ Teach-in	smart.Turn) A I	fool editor	B	
X® 300.000	۵X	W	0.000 ID	001	
Z 450.000	△Z 🔀	C 1	T	1 × 0.0000 z 0.0000	; <u></u>
Y 0.000	ΔY	C 4	™ ⊕	0.000	•
51 1 🔯 0.0 m/nin 0.0 m/nin	1) 1 0 300.0 m/nin 116.8 *	S 1 0 3000	U/min 02 1 F 100)% S ₁ 100%	
TNC:\Project\BHB_KAP1 %1.gmz	[TEL4\gtz\1		efine workpiece bla	ink	
Ĺ 1					
			- u		
		E	Bar/tube blank		13:09
					Back

Workpiece blankSymbolBar/tube blankImage: Comparison of the standard blanks.

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 ${\bf Simulation}$ submode.

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 the **Teach-in** submode (see also "Contour follow-up in Teach-in submode" on page 148):
 - 0: Without contour follow-up
 - 1: With contour follow-up



ICP workpiece blank contour



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

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



4.3 Single cut cycles



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

➡ Teach-in		smart.Tu	rn	🍴 Tool edi	tor	8	
X® 300	0.000 🛆	X	W	0.00	00 ID	001	. . 8
Z 450),000 🛆	Z	🔣 C 1		Т	1 × 0.0000 z 0.0000	0 0
Y C	.000 △	Y	C.4		™ ●	0.000	-
§1 1 💽 💡	.θ mm/nin .θ mm/nin 1	010-300.	0 m/min S 1	3000 U/min	et 1 F 100	S 1 100%	
TNC:\Project\E	BHB_KAPITEL	.4\gtz\1		Single o	uts		•
[]					' = *		
					•	<i>q</i>	
					_ }⊔		
						⇒ [^]	T
				Rapid po	sitioning		13:09
	With return						Back

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 item)	
Machine a chamfer	
Machine a rounding	
Call an M function	⇒ ^м

i

Rapid traverse positioning



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

Cycle parameters

Χ, Ζ	Starting point
X2, Z2	Target point
Т	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 driveOpposing spindle for rear-face machining
BW	Angle in the B axis (machine-dependent function)
\bigcirc	If more axes are available on your machine, additional

input parameters will be displayed.





Move to the tool change position



The tool moves at rapid traverse from the current position to the tool change position (see page 154).

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
- Turret pocket number
- ID Tool ID number

Т

- 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



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





ſ

Linear machining, transverse



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



Linear machining at angle

The MANUALplus 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 MANUALplus 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 Start angle (range: $-180^{\circ} < A < 180^{\circ}$)
- 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 on a linear path 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





Circular machining

The tool moves on a circular path from the **starting point X**, **Z** to the contour **end point X2**, **Z2** at the programmed feed rate and remains at the cycle end position.



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 arc
- 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.







i

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

Cycle p	alameters
X, Z	Starting point
X1, Z1	Corner point of contour
A	Start angle (range of chamfer: 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)
Т	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



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







i

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
- RRadius of rounding arcJElement position (default: 1)—the algebraic sign
- determines the cutting direction (see help graphic)G47 Safety clearance (if "With return" is active)
- G4/ Satety clearance (If "VVith return"
- 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
_p M	Select M function
Enter the nun	nber of the M function
Input finished	Conclude entry
	Press Cycle start
SPINDLE ST	OP M19 (SPINDLE POSITIONING)
	Call the single-cut menu
⇒ ^M	Select M function
Spindle Stop M19	Switch M19 on
Enter the stop	pping angle
Input finished	Conclude entry



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 378.

Proportioning of cuts: The MANUALplus calculates an infeed that is <= infeed depth P. An "abrasive cut" is avoided.</p>

- Oversizes are considered in "expanded" mode.
- **Cutter radius compensation:** Active
- Safety clearance after each cut:
 - Basic mode: 1 mm
 - Expanded mode: The safety clearance is set separately for internal and external machining (see "List of machine parameters" on page 551).

Cutting and infeed directions for turning cycles

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

- Basic mode: The parameters for starting point X, Z (In Machine 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 starting point X, Z (In Machine mode: current tool position) and contour starting point of the ICP contour determine these directions.



Turning cycles

Turning, longitudinal/transverse Roughing and finishing cycle for simple contours



Symbol

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



Cycle Programming

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	
Expanded mode Chamfer (or rounding) 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) at contour start	
Evenended mede	

Expanded mode Chamfer (or rounding) at contour end



i

Cut longitudinal

Select cut, longitudinal/transverse
Select cut longitudinal

The cycle roughs the rectangle described by the starting point and the 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 154)
- G14 Tool change point (see page 154)
- 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 calculates the proportioning of cuts (infeed)
- 2 Approach the workpiece from starting point for first pass
- 3 Move at the feed rate to 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

HEIDENHAIN MANUALplus 620







4.4 Turning cycles

Cut transverse

7



Select cut transverse

The cycle roughs the rectangle described by the starting point and the 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 154)
- G14 Tool change point (see page 154)
- 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 calculates 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





	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 the contour starting point X1/contour end point Z2.

Cycle parameters

Χ, Ζ	Starting point
X1, Z1	Contour starting point
X2, Z2	Contour end point
	Lefere de la contra Marchan

- P Infeed depth: Maximum infeed depth
- A Start angle (range: $0^{\circ} \le A \le 90^{\circ}$)
- W End angle (range: $0^{\circ} \le W \le 90^{\circ}$)
- 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 154)
- G14 Tool change point (see page 154)
- 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: Width of chamfer</p>
- BP Break duration: Time span for interruption of the feed. The chip is broken by the (intermittent) interruption of the feed.
- BF Break 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.







B>0: Radius of rounding

- 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 calculates 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

	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

Cycle pa	liameters
Χ, Ζ	Starting point
X1, Z1	Contour starting point
X2, Z2	Contour end point
Р	Infeed depth: Maximum infeed depth
А	Start angle (range: 0° <= A < 90°)
W	End angle (range: 0° <= W < 90°)
R	Rounding arc
I, K	Oversize X, Z
Н	Contour smoothing
	■ 0: With every cut
	1: With the last cut
	2: No finishing cut
G47	Safety clearance (see page 154)
G14	Tool change point (see page 154)
Т	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: Radius of rounding
	B<0: Width of chamfer
BP	Break duration: Time span for interruption of the feed. The chip is broken by the (intermittent) interruption of the feed.
BF	Break 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

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





1

- 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 calculates 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 154)
- 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

- 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



The cycle finishes the contour area from ${\tt contour}\ {\tt starting}\ {\tt point}\ {\tt Z1}$ to ${\tt contour}\ {\tt end}\ {\tt point}\ {\tt X2}.$



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 154)
- 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

- 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
	Prass the Exnanded soft key
Expanded	Tress 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.

Х,	Ζ	Starting	point
	_	-	

- X1, Z1 Contour starting point
- X2, Z2 Contour end point
- А Start angle (range: 0° <= A < 90°)
- W End angle (range: $0^{\circ} \le W \le 90^{\circ}$)
- R Rounding arc
- DXX Additive compensation number: 1-16 (see page 154)
- G58 Contour-parallel oversize
- G47 Safety clearance (see page 154)
- B1, B2 Chamfer/rounding arc (B1 contour start; B2 contour end)
 - B>0: Radius of rounding
 - B<0: Width of chamfer</p>
- G14 Tool change point (see page 154)
- Т 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.





- 4.4 Turning cycles
- 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)

- 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 point according to the G14 setting

	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 ${\bf contour}\ {\bf starting}\ {\bf point}$ to the ${\bf contour}\ {\bf end}\ {\bf point}.$



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

Cycle parameters

Χ, Ζ	Starting point	
X1 71	Contour startin	

- X1, Z1 Contour starting point
- X2, Z2 Contour end point
- A Start angle (range: $0^\circ \le A < 90^\circ$)
- W End angle (range: $0^{\circ} \le W \le 90^{\circ}$)
- R Rounding arc
- DXX Additive compensation number: 1-16 (see page 154)
- G58 Contour-parallel oversize
- G47 Safety clearance (see page 154)
- G14 Tool change point (see page 154)
- 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: Radius of rounding

■ B<0: Width of chamfer

- 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.





- 4.4 Turning cycles
- 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)

- 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 plunge, longitudinal

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

Select cut, longitudinal/transverse



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: $0^{\circ} \le A \le 90^{\circ}$; default: 0°)
- W End angle—oblique cut at contour end (range: $0^{\circ} \le W \le 90^{\circ}$)
- G47 Safety clearance (see page 154)
- G14 Tool change point (see page 154)
- 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



H=0

H=1

H=2

*

G47 øX2

øx1

—Z2



- Z1-

øх

4.4 T<mark>urn</mark>ing cycles

- 1 calculates the proportioning of cuts (infeed)
- **2** Approach the workpiece from starting point for first pass on paraxial path
- 3 Plunge-cut at plunge 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**
- ${\bf 5}$ $\,$ 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	H=0	- Z
A	Select plunge, transverse	H=1	
The cycle contour	e roughs the area described by the contour starting point ,, end point and plunge angle.	H=2	
	 leaving material remaining. The steeper the tool plunges into the material, the greater the feed rate decrease (max. 50%). 		- 22
Cycle pa X, Z X1, Z1	Starting point Contour starting point	H=0	
X2, Z2 P H	Contour end point Infeed depth: Maximum infeed depth Contour smoothing	H=1	A
А	 0: With every cut 1: With the last cut 2: No finishing cut Plunge angle (range: 0° <= A < 90°: default: 0°) 	H=2	
W	End angle—oblique cut at contour end (range: $0^{\circ} \le W \le 90^{\circ}$)		
G47 G14	Safety clearance (see page 154) Tool change point (see page 154)		
I ID S	l urret pocket number Tool ID number Spindle speed/cutting speed		
F	Feed per revolution		
MT MFS	M after T: M function that is executed after the tool call T. 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

- 1 calculates the proportioning of cuts (infeed)
- **2** Approach the workpiece from starting point for first pass on paraxial path
- 3 Plunge-cut at plunge 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**
- ${\bf 5}$ $\,$ 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 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 plunging-expanded



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%).

- 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: $0^{\circ} \le A \le 90^{\circ}$; default: 0°)
- W End angle—oblique cut at contour end (range: $0^{\circ} \le W \le 90^{\circ}$)
- G14 Tool change point (see page 154)
- 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 Break duration: Time interval until the next break. The chip is broken by the (intermittent) interruption of the feed.
- G47 Safety clearance (see page 154)
- 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

- 1 calculates the proportioning of cuts (infeed)
- **2** Approach the workpiece from starting point for first pass on paraxial path
- 3 Plunge-cut at plunge 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
- ${\bf 5}$ $\,$ The contour is machined depending on the ${\bf contour}\,$ 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
A	Select plunge, transverse
Expanded	Press the Expanded soft key

H=0 H=0 -Z -Z +B1 0X 0X



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: $0^{\circ} \le A \le 90^{\circ}$; default: 0°)
- W End angle—oblique cut at contour end (range: 0° <= W < 90°)
- G14 Tool change point (see page 154)
- 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 Break duration: Time interval until the next break. The chip is broken by the (intermittent) interruption of the feed.
- G47 Safety clearance (see page 154)
- MT M after T: M function that is executed after the tool call T.

4.4 T<mark>urn</mark>ing cycles

- 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

- 1 calculates the proportioning of cuts (infeed)
- **2** Approach the workpiece from starting point for first pass on paraxial path
- 3 Plunge-cut at plunge 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
- ${\bf 5}$ $\,$ The contour is machined depending on the ${\bf contour}\,$ 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



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 154)
- G14 Tool change point (see page 154)
- 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





1

.4 Turning cycles

Cut, transverse finishing plunge



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 par	ameters
Χ, Ζ	Starting point
X1, Z1	Contour starting point
X2, Z2	Contour end point
А	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 154)
G14	Tool change point (see page 154)
Т	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

- 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





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%).

- X, Z Starting point
- X1, Z1 Contour starting point
- X2, Z2 Contour end point
- DXX Additive compensation number: 1-16 (see page 154)
- G58 Contour-parallel oversize
- A Plunge angle (range: $0^{\circ} \le A \le 90^{\circ}$; default: 0°)
- W End angle—oblique cut at contour end (range: 0° <= W < 90°)
- R Rounding arc
- G14 Tool change point (see page 154)
- 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: Radius of rounding
 - B<0: Width of chamfer
- G47 Safety clearance (see page 154)
- 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

- 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





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%).

- X, Z Starting point
- X1, Z1 Contour starting point
- X2, Z2 Contour end point
- DXX Additive compensation number: 1-16 (see page 154)
- G58 Contour-parallel oversize
- A Plunge angle (range: $0^{\circ} \le A \le 90^{\circ}$; default: 0°)
- W End angle—oblique cut at contour end (range: $0^{\circ} \le W \le 90^{\circ}$)
- R Rounding arc
- G14 Tool change point (see page 154)
- 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: Radius of rounding
 - B<0: Width of chamfer
- G47 Safety clearance (see page 154)
- 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

- 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



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!

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.

- 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 154)







- G47 Safety clearance (see page 154)
- G14 Tool change point (see page 154)
- 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 Break 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: perpendicular 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

- 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 on paraxial path
- **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



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!

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.

- 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 154)









- G47 Safety clearance (see page 154)
- G14 Tool change point (see page 154)
- 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 Break 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 Approach angle (reference: Z axis)—(default: perpendicular 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

- 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 on paraxial path
- **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



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 154)
- G58 Contour-parallel oversize
- DI Axis-parallel oversize X
- DK Axis-parallel oversize Z
- SX, SZ Cutting limits (see page 154)
- G47 Safety clearance (see page 154)
- G14 Tool change point (see page 154)
- 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

- **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



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 154)
- G58 Contour-parallel oversize
- DI Axis-parallel oversize X
- DK Axis-parallel oversize Z
- SX, SZ Cutting limits (see page 154)
- G47 Safety clearance (see page 154)
- G14 Tool change point (see page 154)
- 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

HEIDENHAIN MANUALplus 620





207

4.4 Turning cycles

ICP cutting, longitudinal



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%).

- X, Z Starting point
- FΚ ICP finished part: Name of the contour to be machined
- Ρ Infeed depth: Maximum infeed depth
- Н Contour smoothing
 - 0: With every cut
 - 1: With the last cut
 - 2: No finishing cut
- I, K Oversize X, Z
- Е 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 154)
- G47 Safety clearance (see page 154)
- G14 Tool change point (see page 154)
- Т Turret pocket number
- ID Tool ID number
- S Spindle speed/cutting speed
- F Feed per revolution
- ΒP Break duration: Time span for interruption of the feed. The chip is broken by the (intermittent) interruption of the feed.
- ΒF Break duration: Time interval until the next break. The chip is broken by the (intermittent) interruption of the feed.
- А Approach angle (reference: Z axis)-(default: parallel to Z axis)
- W Departure angle (reference: Z axis)-(default: perpendicular 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

- 1 calculates the proportioning of cuts (infeed)
- **2** Approach the workpiece from starting point for first pass on paraxial path
- 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 ${\rm H}$
- 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 cut transverse



Select ICP cutting, transverse

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



-77

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%).

- 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 154)
- G47 Safety clearance (see page 154)
- G14 Tool change point (see page 154)
- 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 Break 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 Approach angle (reference: Z axis)—(default: perpendicular 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

- 1 calculates the proportioning of cuts (infeed)
- **2** Approach the workpiece from starting point for first pass on paraxial path
- **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 ${\rm H}$
- 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



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 154)
- G58 Contour-parallel oversize
- DI Axis-parallel oversize X
- DK Axis-parallel oversize Z
- SX, SZ Cutting limits (see page 154)
- G47 Safety clearance (see page 154)
- G14 Tool change point (see page 154)
- 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

- 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



DXX XX=01..16 4.4 Turning cycles

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 154)
- G58 Contour-parallel oversize
- DI Axis-parallel oversize X
- DK Axis-parallel oversize Z
- SX, SZ Cutting limits (see page 154)
- G47 Safety clearance (see page 154)
- G14 Tool change point (see page 154)
- 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

HEIDENHAIN MANUALplus 620





Examples of turning cycles

Roughing and finishing an outside contour



The shaded area from **AP** (contour starting point) to **EP** (contour end point) is rough-machined with the cycle **Cut 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)
- TO = 1 (tool orientation)
- A = 93° (tool angle)
- B = 55° (point angle)





Roughing and finishing an inside contour



The shaded area from **AP** (contour starting point) to **EP** (contour end point) is rough-machined with the cycle **Cut 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 positive X-axis direction.

Tool data

- Turning tool (for internal machining)
- TO = 7 (tool orientation)
- \blacksquare A = 93° (tool angle)
- \blacksquare 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.

1st step:

The shaded area from **AP** (contour starting point) 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 MANUALplus 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)
- TO = 1 (tool orientation)
- $\blacksquare A = 93^{\circ}$ (tool angle)
- \blacksquare B = 55° (point angle)
2nd 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)
- TO = 3 (tool orientation)
- A = 93° (tool angle)
- B = 55° (point angle)

4.4 T<mark>urn</mark>ing cycles

217

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 378).



Proportioning of cuts: The MANUALplus calculates a constant recessing width that is <= P.</p>

- **Oversizes** are considered in "expanded" mode.
- **Cutter radius compensation** is conducted (exception: Undercut type K).



Cutting and infeed directions for recessing cycles

The MANUALplus automatically determines the cutting and infeed directions from the cycle parameters. The decisive ones are:

- Basic mode: Parameters for starting point X, Z (In Machine 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: Parameters for starting point X, Z (In Machine mode: current tool position) and contour starting point of the ICP contour

Recessing cycles	Symbo	
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		P
Undercut K Undercut type K		20
Undercut U Undercut type U		
Parting Cycle for parting the workpiece		

Undercut position

The MANUALplus determines the position of an undercut from the cycle parameters for **starting point X**, **Z** (Machine mode: current tool position) and **corner point of contour X1**, **Z1**.



Undercuts can only be executed in orthogonal, paraxial contour corners along the longitudinal axis.

Contour forms





Recessing, radial

┉┟

Ш́



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 154)
- G14 Tool change point (see page 154)
- 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



In this cycle you can specify how the floor element is machined in sectional view.

For this purpose the control evaluates the machining parameter Processing/recessing/finishing(602414). If this parameter is not defined the floor element is divided in the center.

- **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

f



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 154)
- G14 Tool change point (see page 154)
- 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







In this cycle you can specify how the floor element is machined in sectional view.

For this purpose the control evaluates the machining parameter Processing/recessing/finishing(602414). If this parameter is not defined the floor element is divided in the center.

- 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



The cycle machines the number of recesses defined in **number Qn**. The parameters for the **contour starting point** and **contour end point** define the first recess (position, recess depth and recess width).

Cycle parameters

X, Z X1, Z1 X2, Z2 B1, B2	Starting point Contour starting point Contour end point Chamfer/rounding arc (B1 contour start; B2 contour end)
	 B>0: Radius of rounding B<0: Width of chamfer
А	Start angle (range: $0^\circ \le A \le 90^\circ$)
W	End angle (range: $0^\circ \le W \le 90^\circ$)
R	Rounding arc
I, K	Oversize X, Z
G14	Tool change point (see page 154)
Т	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed
F	Feed per revolution
Ρ	Recessing width: Infeeds $\leq P$ (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 154)
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.







i



In this cycle you can specify how the floor element is machined in sectional view.

For this purpose the control evaluates the machining parameter Processing/recessing/finishing(602414). If this parameter is not defined the floor element is divided in the center.

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

- 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



The cycle machines the number of recesses defined in **number Qn**. The parameters for the **contour starting point** and **contour end point** define the first recess (position, recess depth and recess width).

Cycle parameters

- X, ZStarting pointX1, Z1Contour starting point
- X2, Z2 Contour end point
- B1, B2 Chamfer/rounding arc (B1 contour start; B2 contour end)
 - B>0: Radius of rounding
 - B<0: Width of chamfer
- A Start angle (range: $0^{\circ} \le A \le 90^{\circ}$)
- W End angle (range: $0^{\circ} \le W \le 90^{\circ}$)
- R Rounding arc
- I, K Oversize X, Z
- G14 Tool change point (see page 154)
- 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 154)
- 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



In this cycle you can specify how the floor element is machined in sectional view.

For this purpose the control evaluates the machining parameter Processing/recessing/finishing(602414). If this parameter is not defined the floor element is divided in the center.

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

- 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





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

Χ, Ζ	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 154)
G14	Tool change point (see page 154)
Т	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.
MES	M at beginning. M function that is executed at the

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

i

- 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



In this cycle you can specify how the floor element is machined in sectional view.

For this purpose the control evaluates the machining parameter Processing/recessing/finishing(602414). If this parameter is not defined the floor element is divided in the center.

- 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





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

Χ, Ζ	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 154)
G14	Tool change point (see page 154)
Т	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

i



In this cycle you can specify how the floor element is machined in sectional view.

For this purpose the control evaluates the machining parameter Processing/recessing/finishing(602414). If this parameter is not defined the floor element is divided in the center.

- 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





The cycle machines the number of recesses defined in **number Qn.** The parameters for the **contour starting point** and **contour end point** 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: Radius of rounding
 - B<0: Width of chamfer</p>
 - Start angle (range: 0° <= A < 90°)
- W End angle (range: $0^{\circ} \le W \le 90^{\circ}$)
- R Rounding arc

А

- G14 Tool change point (see page 154)
- 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 154)
- 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





In this cycle you can specify how the floor element is machined in sectional view.

For this purpose the control evaluates the machining parameter Processing/recessing/finishing(602414). If this parameter is not defined the floor element is divided in the center.

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

- 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





The cycle machines the number of recesses defined in **number Qn.** The parameters for the **contour starting point** and **contour end point** 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: Radius of rounding
 - B<0: Width of chamfer</p>
 - Start angle (range: 0° <= A < 90°)
- W End angle (range: $0^{\circ} \le W \le 90^{\circ}$)
- R Rounding arc

А

- G14 Tool change point (see page 154)
- 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 154)
- 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





In this cycle you can specify how the floor element is machined in sectional view.

For this purpose the control evaluates the machining parameter Processing/recessing/finishing(602414). If this parameter is not defined the floor element is divided in the center.

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

- 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 radial



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 154)
- T Turret pocket number
- ID Tool ID number
- S Spindle speed/cutting speed
- F Feed per revolution
- SX, SZ Cutting limits (see page 154)
- G47 Safety clearance (see page 154)
- 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









In this cycle you can specify how the floor element is machined in sectional view.

For this purpose the control evaluates the machining parameter Processing/recessing/finishing(602414). If this parameter is not defined the floor element is divided in the center.

- 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



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
- FKICP finished part: Name of the contour to be machinedPRecessing 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 154)
- T Turret pocket number
- ID Tool ID number
- S Spindle speed/cutting speed
- F Feed per revolution
- SX, SZ Cutting limits (see page 154)
- G47 Safety clearance (see page 154)
- 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









In this cycle you can specify how the floor element is machined in sectional view.

For this purpose the control evaluates the machining parameter Processing/recessing/finishing(602414). If this parameter is not defined the floor element is divided in the center.

- 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







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
- DX, DZ Distance to subsequent recess with respect to the preceding recess
- G14 Tool change point (see page 154)
- T Turret pocket number
- ID Tool ID number
- S Spindle speed/cutting speed
- F Feed per revolution
- SX, SZ Cutting limits (see page 154)
- G47 Safety clearance (see page 154)
- 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







In this cycle you can specify how the floor element is machined in sectional view.

For this purpose the control evaluates the machining parameter Processing/recessing/finishing(602414). If this parameter is not defined the floor element is divided in the center.

- 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





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, ZStarting pointFKICP finished part: Name of the contour to be machinedQnNumber of recess cycles (default: 1)DX, DZDistance to subsequent recess with respect to the
preceding recess
- G14 Tool change point (see page 154)
- T Turret pocket number
- ID Tool ID number
- S Spindle speed/cutting speed
- F Feed per revolution
- SX, SZ Cutting limits (see page 154)
- G47 Safety clearance (see page 154)
- 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





In this cycle you can specify how the floor element is machined in sectional view.

For this purpose the control evaluates the machining parameter Processing/recessing/finishing(602414). If this parameter is not defined the floor element is divided in the center.

- 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 0: Feed rate for recessing movement
- **Turning operation, unidirectional/bidirectional U:** You can perform a unidirectional or bidirectional turning operation.
- 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 MANUALplus 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.

~
2

These cycles require the use of **recess-turning tools**.

Recess turning, radial

	Call the recessing cycles	
	Select recess turning	
Ľ	Select recess turning, radial	



The cycle clears the rectangle described by the **starting point** and contour end point.

Cycle narameters

Cycle par	ameters
Χ, Ζ	Starting point
X2, Z2	Contour end point
Р	Infeed depth: Maximum infeed depth
0	Recessing feed rate (default: active feed rate)
В	Offset width (default: 0)
U	Unidirectional turning (default: 0)
	■ 0: Bidirectional
	1: Unidirectional
G47	Safety clearance (see page 154)
G14	Tool change point (see page 154)
Т	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



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 154)
- G14 Tool change point (see page 154)
- 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

- **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



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 244).

Cycle parameters

-,	
Χ, Ζ	Starting point
X1, Z1	Contour starting point
X2, Z2	Contour end point
Р	Infeed depth: Maximum infeed depth
0	Recessing feed rate (default: active feed rate)
I, K	Oversize X, Z
А	Start angle (range: 0° <= A < 90°)
W	End angle (range: 0° <= W < 90°)
R	Rounding arc
G14	Tool change point (see page 154)
Т	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: Radius of rounding
	■ B<0: Width of chamfer
В	Offset width (default: 0)
U	Unidirectional turning (default: 0)
	0: Bidirectional
	1: Unidirectional
G47	Safety clearance (see page 154)
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.





i

- 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

- 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

Recess turning, axial-expanded





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 244).

Cycle parameters

-, p	
Χ, Ζ	Starting point
X1, Z1	Contour starting point
X2, Z2	Contour end point
Р	Infeed depth: Maximum infeed depth
0	Recessing feed rate (default: active feed rate)
I, K	Oversize X, Z
А	Start angle (range: $0^{\circ} \le A \le 90^{\circ}$)
W	End angle (range: 0° <= W < 90°)
R	Rounding arc
G14	Tool change point (see page 154)
Т	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: Radius of rounding
	■ B<0: Width of chamfer
В	Offset width (default: 0)
U	Unidirectional turning (default: 0)
	O: Pidiroctional
C 47	
	Mafter T: M function that is executed after the tool call T
	M at beginning: M function that is executed after the tool call 1.
IVIF3	beginning of the machining step
MFF	M at end: M function that is executed at the end of the
	machining step.





1

- 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

- 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

Recess turning, radial finishing



The cycle finishes the contour area defined by the **starting point** and **contour end point** (see also "Recess turning" on page 244).



With **oversizes I**, **K**, you define the material left remaining after the finishing cycle.

Cycle parameters

- X, Z Starting point
- X2, Z2 Contour end point
- I, K Oversize X, Z
- G14 Tool change point (see page 154)
- T Turret pocket number
- ID Tool ID number
- S Spindle speed/cutting speed
- F Feed per revolution
- G47 Safety clearance (see page 154)
- 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




- **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**
- $\textbf{3} \quad \text{Move paraxially to the starting point X / end point Z}$
- 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



The cycle finishes the contour area defined by the **starting point** and **contour end point** (see also "Recess turning" on page 244).



With **oversizes I**, **K**, you define the material left remaining after the finishing cycle.

Cycle parameters

- X, Z Starting point
- X2, Z2 Contour end point
- I, K Oversize X, Z
- G14 Tool change point (see page 154)
- T Turret pocket number
- ID Tool ID number
- S Spindle speed/cutting speed
- F Feed per revolution
- G47 Safety clearance (see page 154)
- 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





- **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**
- $\textbf{3} \quad \text{Move paraxially to the starting point } \textbf{Z} \,/\, \textbf{end point } \textbf{X2}$
- 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



The cycle finishes the contour area defined by the **contour starting point** and **contour end point** (see also "Recess turning" on page 244).

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.

With **oversizes I**, **K**, you define the material left remaining after the finishing cycle.

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
- RI, RK Workpiece blank oversizes in X and Z
- A Start angle (range: $0^{\circ} \le A \le 90^{\circ}$)
- W End angle (range: $0^{\circ} \le W \le 90^{\circ}$)
- R Rounding arc
- G14 Tool change point (see page 154)
- 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: Radius of rounding
 - B<0: Width of chamfer





- 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 154)
- 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

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

- **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





The cycle finishes the contour area defined by the **contour starting point** and **contour end point** (see also "Recess turning" on page 244).



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.

With **oversizes I**, **K**, you define the material left remaining after the finishing cycle.

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
- RI, RK Workpiece blank oversizes in X and Z
- A Start angle (range: $0^{\circ} \le A \le 90^{\circ}$)
- W End angle (range: $0^{\circ} \le W \le 90^{\circ}$)
- R Rounding arc
- G14 Tool change point (see page 154)
- 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: Radius of rounding
 - B<0: Width of chamfer



- 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 154)
- 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

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

- **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



The cycle clears the defined area (see also "Recess turning" on page 244).



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 154)
- B Offset width (default: 0)
- U Unidirectional turning (default: 0)
 - 0: Bidirectional
 - 1: Unidirectional (direction: see help graphic)
- G14 Tool change point (see page 154)
- 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 154)
- 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

- 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





The cycle clears the defined area (see also "Recess turning" on page 244).



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
Р	Infeed depth: Maximum infeed depth
ET	Recessing depth by which one cut is fed.
0	Recessing feed rate (default: active feed rate)
I, K	Oversize X, Z
SX, SZ	Cutting limits (see page 154)
В	Offset width (default: 0)
U	Unidirectional turning (default: 0)
	0: Bidirectional
	1: Unidirectional (direction: see help graphic)
G14	Tool change point (see page 154)

- 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 154)
- 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

- 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





The cycle finishes the contour area defined by the ICP contour (see also "Recess turning" on page 244). 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.

With **oversizes I, K,** you define the material left remaining after the finishing cycle.



- 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
- RI, RK Workpiece blank oversizes in X and Z
- SX, SZ Cutting limits (see page 154)
- G14 Tool change point (see page 154)
- 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 154)
- 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

- 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





The cycle finishes the contour area defined by the ICP contour (see also "Recess turning" on page 244). 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.

With **oversizes I, K,** you define the material left remaining after the finishing cycle.

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
- RI, RK Workpiece blank oversizes in X and Z
- SX, SZ Cutting limits (see page 154)
- G14 Tool change point (see page 154)
- 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 154)
- 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

- 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

шĽ

6



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 **plunge 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 plunge 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 Plunge angle (default: W is calculated)
- G47 Safety clearance (see page 154)
- G14 Tool change point (see page 154)
- 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

- 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 $% \left({{{\mathbf{r}}_{\mathbf{r}}}_{\mathbf{r}}} \right)$

Select undercut K

This cycle performs only one linear 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 154)
- G14 Tool change point (see page 154)
- 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

- 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: Radius of rounding
 - B<0: Width of chamfer
- G47 Safety clearance (see page 154)
- G14 Tool change point (see page 154)
- 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

- **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

шĽ

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: Radius of rounding
 - B<0: Width of chamfer</p>
- E Reduced feed rate
- D Maximum speed
- K Retraction distance after parting: Lift off the tool laterally from the plane surface before retraction
- 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 154)
- G14 Tool change point (see page 154)
- 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 MANUALplus switches to the reduced feed ${\bf E}$ as soon as the tool reaches the diameter for feed-rate reduction ${\bf I}.$

- 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

4.5 Rec<mark>ess</mark>ing cycles



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)
- TO = 1 (tool orientation)
- SB = 4 (cutting width 4 mm)







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 **recessing width P** is not entered, the MANUALplus plungecuts with 80 % of the recessing 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)
- TO = 7 (tool orientation)
- \blacksquare SB = 2 (cutting width 2 mm)





b (

4.6 Thread and undercut cycles



4.6 Thread and un<mark>de</mark>rcut cycles

The thread and undercut cycles machine single or multistart 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.
 Feed rate override is not effective during cycle execution.





The machine manufacturer specifies whether the tool is retracted at cycle stop before the movement is stopped or if the thread is finished.

Thread position, undercut position

Thread position

The MANUALplus determines the direction of the thread from the parameters for **starting point Z** (**Machine** mode: Current tool position and **end point Z2**. You select internal or external thread by soft key.

Undercut position

The MANUALplus determines the position of an undercut from the parameters starting point X, Z (Machine mode: current tool position) 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	lcon
Thread cycle Longitudinal single or multi-start thread	[]
Tapered thread Tapered single or multi-start thread	Limm,
API thread Single or multi-start API thread (API: American Petroleum Institute)	API
Undercut DIN 76 Thread undercut and thread chamfer	
Undercut DIN 509 E Undercut and cylinder chamfer	DIN 509 E
Undercut DIN 509 F Undercut and cylinder chamfer	DIN 509 F

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 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 MANUALplus 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 MANUALplus displays a warning.

.

Last cut

After the cycle is finished, the MANUALplus 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



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)



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	param	eters

- X, Z Starting point of thread
- Z2 End point of thread
- F1 Thread pitch (= feed rate)
- U Thread depth No input:
 - Outside thread: U=0.6134*F1
 - Inside 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: I is calculated from U and F1
- G47 Safety clearance (see page 154)
- G14 Tool change point (see page 154)
- 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 Infeed angle (range: $-60^{\circ} \le A \le 60^{\circ}$; default: 30°)
 - A<0: Infeed on left thread flank
 - A>0: Infeed on left right 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

- 1 Calculate the proportioning of cuts
- 2 Start first pass at starting point Z
- 3 Move at the feed rate to 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



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 Threads per unit (default: 1 single-start thread)
- U Thread depth No input:
 - Outside thread: U=0.6134*F1
 - Inside 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: I is calculated from U and F1
- GK Run-out length
- G47 Safety clearance (see page 154)
- G14 Tool change point (see page 154)
- 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)
А	Infeed angle (range: -60° <= A < 60°; default: 30°)
	■ A<0: Infeed on left thread flank
	A>0: Infeed on left right 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

- **1** Calculate the proportioning of cuts
- 2 Start the first thread groove at starting point Z
- $\textbf{3} \quad \text{Move at the feed rate to end point } \textbf{Z2}$
- 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 thread depth U are reached
- 8 Move to the tool change point according to the G14 setting

Tapered thread



This cycle cuts a single or multi-start tapered external or internal thread.

Cycle parameters

• , • • • • •	
Χ, Ζ	Starting point
X1, Z1	Starting point of thread
X2, Z2	End point of thread
F1	Thread pitch (= feed rate)
D	Threads per unit (default: 1 single-start thread)
U	Thread depth – No input:
	Outside thread: U=0.6134*F1
	■ Inside thread: U=-0.5413*F1
I	Maximum infeed
	I <u: cut="" cuts:="" cutting="" depth="" depth<="" first="" further="" i;="" of="" p="" reduction="" with=""></u:>
	■ I=U: One cut
	No input: I is calculated from U and F1
W	Taper angle (range: –60° < A < 60°)
GK	Run-out length
	■ GK<0: Run-out at start of thread
	GK>0: Run-out at end of thread
G47	Safety clearance (see page 154)
G14	Tool change point (see page 154)
Т	Turret pocket number
ID	Tool ID number
S	Spindle speed/cutting speed







1

- 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 Infeed angle (range: $-60^{\circ} \le A \le 60^{\circ}$; default: 30°)
 - A<0: Infeed on left thread flank
 - A>0: Infeed on left right 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



4.6 Thread and un<mark>de</mark>rcut cycles

Parameter combinations for the taper angle:

- X1/Z1, X2/Z2
- X1/Z1, Z2, W
- Z1, X2/Z2, W

- **1** Calculate the proportioning of cuts
- 2 Move to thread starting point X1, Z1
- 3 Move at the feed rate to 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 thread depth U are reached
- 8 Move to the tool change point according to the G14 setting

API thread

	Call the thread-cutting menu
API	Select API thread
Inner thread	On: Inside threadOff: Outside 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 X1, Z1 X2, Z2 F1 D U	Starting point Starting point of thread End point of thread Thread pitch (= feed rate) Threads per unit (default: 1 single-start thread) Thread depth – No input:
I	 Outside thread: U=0.6134*F1 Inside thread: U=-0.5413*F1 1st cutting depth
WE W G47 G14 T ID S GV	 I<u: cut="" cuts:="" cutting="" depth="" down="" first="" i—further="" j<="" li="" of="" reduction="" to="" with=""> I=U: One cut No input: Calculation from U and F1 Run-out angle (range: 0° <= WE < 90°) Taper angle (range: -60° < A < 60°) Safety clearance (see page 154) Tool change point (see page 154) Turret pocket number Tool ID number Spindle speed/cutting speed Type of infeed </u:>
	 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)







i

4.6 Thread and undercut cycles

А

R

- GH Type of offset
 - 0: Without offset
 - 1: From left
 - 2: From right
 - 3: Alternately left/right
 - Infeed angle (range: -60° <= A < 60°; default: 30°)
 - A<0: Infeed on left thread flank
 - A>0: Infeed on left right flank
 - 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

- 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 runout 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 ${\bf D}$ and thread depth ${\bf U}$ 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: Inside threadOff: Outside thread



This optional cycle reworks a single-start thread. Since you have already unclamped the workpiece, the MANUALplus 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 angle** 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:
 - Outside thread: U=0.6134*F1
 - Inside 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 Infeed angle (range: $-60^{\circ} \le A \le 60^{\circ}$; default: 30°)
 - A<0: Infeed on left thread flank
 - A>0: Infeed on left right 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.

HEIDENHAIN MANUALplus 620



- 4.6 Thread and un<mark>de</mark>rcut cycles
- 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

- 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 position 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

	Call the thread-cutting menu
	Select thread cycle
Expanded	Press the Expanded soft key
Re- cut	Press the Recut soft key
Inner thread	 On: Inside thread Off: Outside thread

This optional cycle recuts a single or multi-start external or internal thread. Since you have already unclamped the workpiece, the MANUALplus 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 angle** 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.

- 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:
 - Outside thread: U=0.6134*F1
 - Inside 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 Infeed angle (range: $-60^{\circ} \le A \le 60^{\circ}$; default: 30°)
 - A<0: Infeed on left thread flank
 - A>0: Infeed on left right 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

- 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 position ZC and measured angle C
- 3 Move the tool manually out of the thread groove
- Position the tool to the starting point 4
- Start cycle with the Input finished soft key, then press Cycle 5 Start

Tapered thread, recutting





This optional cycle recuts a single or multi-start external or internal taper thread. Since you have already unclamped the workpiece, the MANUALplus 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 angle** 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:
 - Outside thread: U=0.6134*F1
 - Inside 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
 - Taper angle (range: –60° < A < 60°)
- GK Run-out length

W

- GK<0: Run-out at beginning of thread
- GK>0: Run-out at end of thread
- C Measured angle
- ZC Measured position







А

- Infeed angle (range: -60° <= A < 60°; default: 30°)
 - A<0: Infeed on left thread flank
 - A>0: Infeed on left right 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

- 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 position 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

API thread, recutting



This optional cycle recuts a single or multi-start external or internal API thread. Since you have already unclamped the workpiece, the MANUALplus 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 angle** 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:
 - Outside thread: U=0.6134*F1
 - Inside 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: $0^{\circ} \le WE \le 90^{\circ}$)
- W Taper angle (range: $-60^{\circ} < A < 60^{\circ}$)
- C Measured angle
- ZC Measured position
- A Infeed angle (range: $-60^{\circ} \le A \le 60^{\circ}$; default: 30°)
 - A<0: Infeed on left thread flank
 - A>0: Infeed on left right 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

- 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 position 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



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.

- 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 154)
- 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 154)—evaluated only if "With return" 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 MANUALplus determines these parameters from "FP" in the standard table (see "DIN 76—undercut parameters" on page 617).

- **1** Approach workpiece from starting point
 - to the cylinder starting point X1, or
 - \blacksquare 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: the tool remains at end point on plane surface
 - **With return:** Retract and return diagonally back to starting point
- 8 Move to the tool change point according to the G14 setting

Undercut DIN 509 E



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 start 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 154)
- 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 154)—evaluated only if "With return" 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.



4.6 Thread and un<mark>de</mark>rcut cycles





- 4.6 Thread and un<mark>de</mark>rcut cycles
- 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 MANUALplus determines these parameters from the cylinder diameter in the standard table (see "DIN 509 E – undercut parameters" on page 619).

- 1 Approach workpiece from starting point
 - \blacksquare 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: the tool remains at end point on plane surface
 - **With return:** Retract and return diagonally back to starting point
- 7 Move to the tool change point according to the G14 setting

Undercut DIN 509 F



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.

- 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 start 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 154)
- 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 154)—evaluated only if "With return" 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 MANUALplus determines these parameters from the cylinder diameter in the standard table (see "DIN 509 F – undercut parameters" on page 619).

- 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: the tool remains at end point on plane surface
 - **With return:** Retract and return diagonally back 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.

1st step

The parameters for the undercut and thread chamfer are programmed in two superimposed input windows.

Tool data

Turning tool (for external machining)

- \blacksquare TO = 1 (tool orientation)
- A = 93° (tool angle)
- B = 55° (point angle)

2nd 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)
- \blacksquare TO = 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.

1st step

The parameters for the undercut and thread chamfer are programmed in two superimposed input windows.

The MANUALplus 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)
- TO = 7 (tool orientation)
- A = 93° (tool angle)
- \blacksquare B = 55° (point angle)

2nd step

The **thread cycle (longitudinal)** cuts the thread. The thread pitch is defined. The MANUALplus 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)

■ TO = 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 355.



Drilling cycles	lcon	
Axial/radial drilling cycle For drilling single holes and patterns		
Axial/radial deep-hole drilling cycle For drilling single holes and patterns	<u> </u>	U U U
Axial/radial tapping cycle For drilling single holes and patterns	<	
Thread milling For milling threads in existing holes		R



Drilling, axial

Select drilling

Select drilling, axial

This cycle drills a hole on the face of the workpiece.

- X, Z Starting point
- C Spindle angle (C-axis position)
- Z1 Start point drill (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 154)
- G60 Deactivate the protective zone for the drilling operation
 - 0: Active
 - 1: Inactive
- G14 Tool change point (see page 154)
- 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 Break 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: Drilling
- Indexable insert drill: Predrilling

If "AB" and "V" are programmed, the feed rate is reduced by 50% during both pre-drilling and through-boring.

The MANUALplus uses the tool parameter driven tool to determine whether the programmed spindle speed and feed rate apply to the spindle or the driven tool.

- 1 Position spindle to **spindle angle C** (in **Machine** mode: machining starts from the current spindle angle)
- 2 If defined, move at rapid traverse to hole starting point Z1
- 3 If defined, spot drill at reduced feed rate
- 4 Depending on through drilling variants V:
 - Through-drilling reduction:
 - Drill at programmed feed rate to position Z2 AB
 - Drill at reduced feed rate to hole end point $\ensuremath{\textbf{Z2}}$
 - 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

Drilling, radial

Select drilling

Select drilling, radial

This cycle drills a hole on the lateral surface of the workpiece.

Cycle parameters

4

- X, Z Starting point
- C Spindle angle (C-axis position)
- X1 Start point drill (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 154)
- G14 Tool change point (see page 154)
- 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 Break 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: Drilling
- Indexable insert drill: **Predrilling**



If "AB" and "V" are programmed, the feed rate is reduced by 50% during both pre-drilling and through-boring.

- 1 Position spindle to **spindle angle C** (in **Machine** 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 through drilling variants 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 $\ensuremath{\textbf{hole}}\xspace$ end $\ensuremath{\textbf{point}}\xspace$ 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 **reduction value**, however, without falling below the **minimum drilling depth**.

- Χ, Ζ Starting point С Spindle angle (C-axis position) Ζ1 Start point drill (starting point of hole; default: drilling starts from Z) Ζ2 End point drill (end point of hole) Ρ 1. drilling depth (default: hole will be drilled in one pass) IΒ Hole depth reduction value (default: 0) JB Minimum hole depth (default: 1/10 of P) В Retraction length (default: retract to starting point of hole) Е 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 154)
 - T Turret pocket number
 - ID Tool ID number
 - S Spindle speed/cutting speed
 - F Feed per revolution
 - SCK Safety clearance (see page 154)
 - 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 Break 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: Drilling

Indexable insert drill: **Predrilling**



If "AB" and "V" are programmed, the feed rate is reduced by 50 % during both pre-drilling and through-boring.

The MANUALplus uses the tool parameter driven tool to determine whether the programmed spindle speed and feed rate apply to the spindle or the driven tool.

- **1** Position spindle to **spindle angle C** (in **Machine** mode: machining starts from the current spindle angle)
- 2 If defined, move at rapid traverse to hole starting point 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 through drilling variants 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 drilling, 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 **reduction value**, however, without falling below the minimum drilling depth.

- Χ, Ζ Starting point С Spindle angle (C-axis position) X1 Start point drill (starting point of hole; default: drilling starts from X) Х2 End point drill (end point of hole) Ρ 1. drilling depth (default: hole will be drilled in one pass) IΒ Hole depth reduction value (default: 0) JB Minimum hole depth (default: 1/10 of P) В
- Retraction length (default: retract to starting point of hole)
- Е 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 154)
- Т Turret pocket number
- ID Tool ID number
- S Spindle speed/cutting speed
- F Feed per revolution
- SCK Safety clearance (see page 154)
- Break duration: Time span for interruption of the feed. The ΒP chip is broken by the (intermittent) interruption of the feed.
- BF Break 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: Drilling

Indexable insert drill: **Predrilling**



If "AB" and "V" are programmed, the feed rate is reduced by 50 % during both pre-drilling and through-boring.

- **1** Position spindle to **spindle angle C** (in **Machine** 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 through drilling variants V:
 - Through-drilling reduction:
 - Drill at programmed feed rate to position X2 AB
 - Drill at reduced feed rate to $\ensuremath{\text{hole}}$ end $\ensuremath{\text{point}}$ 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	dri1	1 i	ng
--------	------	-----	----

Select tapping, axial

This cycle is used to tap a thread on the face of a workpiece.

Meaning of the **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.

- X, Z Starting point
- C Spindle angle (C-axis position)—(default: current spindle angle)
- Z1 Start point drill (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 154)
- G60 Deactivate the protective zone for the drilling operation
 - 0: Active
 - 1: Inactive
- G14 Tool change point (see page 154)
- T Turret pocket number
- ID Tool ID number
- S Spindle speed/cutting speed
- SP Chip breaking depth
- SI Retraction distance
- 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 MANUALplus uses the tool parameter **driven tool** to determine whether the programmed spindle speed and feed rate apply to the spindle or the driven tool.

- 1 Position spindle to **spindle angle C** (in **Machine** mode: machining starts from the current spindle angle)
- 2 If defined, move at rapid traverse to hole starting point Z1
- $\textbf{3} \quad \text{Tap thread to hole end point } \textbf{Z2}$
- 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	dri1	1	ing
--------	------	---	-----

Select tapping, radial

This cycle is used to tap a thread on the lateral surface of a workpiece.

Meaning of the **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.

- X, Z Starting point
- C Spindle angle (C-axis position)—(default: current spindle angle)
- X1 Start point drill (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 154)
- G60 Protection zone—deactivates the protection zone for the drilling process
 - 0: Active
 - 1: Inactive
- G14 Tool change point (see page 154)
- T Turret pocket number
- ID Tool ID number
- S Spindle speed/cutting speed
- SP Chip breaking depth
- SI Retraction distance
- 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

- 1 Position spindle to **spindle angle C** (in **Machine** 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



- G14 Tool change point (see page 154)
- 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

- 1 Position spindle to **spindle angle C** (in **Machine** 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 MANUALplus uses the thread pitch of the tool. The **return speed SR** ensures that the tool is retracted quickly.

Tool data (drill)

- TO = 8 (tool orientation)
- I = 8.2 (drilling diameter)
- B = 118 (point angle)
- \blacksquare H = 0 (the tool is not a driven tool)

Tool data (tap)

■ TO = 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.

1. hole depth P and the depth reduction value IB define the individual passes, and the **minimum hole depth JB** limits the hole reduction value.

As the **retraction 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 predrilling and through-boring.

Tool data

 \blacksquare TO = 8 (tool orientation)

I = 12 (drilling diameter)

 \blacksquare B = 118 (point angle)

 \blacksquare 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 355.

In **Teach-in** submode 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.

➡ Machine	smart.Turn	🍴 🏷 Tool edit	or	8	
X 300.000 Z 450.000	B 0.000 🖬	, i ⊕ 0.000 _ K	T : ID	L × 0.0000 z 0.0000 001	
Y 0.000 ^{S1} 1 0 0.000 mm/1 0.0 mm/nin	C ₁ C ₂ 1) 1 O ^{1500.0} n/nin 20.0 1	2 0 1500.0 n/ni 0.0 U/ni	Q1 In 1 F 100%	0.000 S1100%	=0
		Milling		P.f	
		Axial slo	et 🕄		12,55
Pattern Pattern linear circula	n .r				Back

	-
Milling cycles	lcon
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

I		1	-
	-	T	-
			F

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 angle)
- 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

- 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

F

8

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 154)
- G14 Tool change point (see page 154)
- 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
- ${\bf 8}$ $\,$ Position to starting point ${\bf Z}$ and deactivate C axis
- 9 Move to the tool change point according to the G14 setting

Figure, axial

J

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 oder Q>4, L<>0)
- Cvcle parameters (first input window)
- Χ, Ζ Starting point
- С Spindle angle (C-axis position)—(default: current spindle angle)
- X1 Diameter of figure center
- C1 Angle of figure center (default: spindle angle C)
- 0 Number of edges (default: 0)
 - Q=0: Circle
 - Q=4: Rectangle, square
 - Q=3: Triangle
 - Q>4: Polygon
- L Edge length
 - Rectangle: Rectangle length
 - Square, polygon: Edge length
 - Polygon: L<0 inscribed circle diameter</p>
 - Circle: No input
 - Rectangle width
 - Rectangle: Rectangle width
 - Square: L=B
- Polygon, circle: No input RE
 - Rounding radius (default: 0)
 - Rectangle, square, polygon: rounding radius
- Circle: circle radius А
 - 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 154)
- Т Turret pocket number
- ID Tool ID number
- S Spindle speed/cutting speed
- F Feed per revolution







В

4.8 Milling cycles

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)
 - O: 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
- RB Retraction plane
- SCI Safety clearance in the working plane
- SCK Safety clearance in infeed direction (see page 154)
- 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.



11T=1



4.8 Milling cycles

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 347).
- **Milling cutter radius 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 approach 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 **approach 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 **approach 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 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 154)
- 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: Deburring
- 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
- RB Retraction plane
- SCI Safety clearance in the working plane
- SCK Safety clearance in infeed direction (see page 154)
- 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 347).
- Milling cutter radius 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 approach 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 **approach 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 **approach 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 oder 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: Rectangle length
 - Square, polygon: Edge length
 - Polygon: L<0: Inscribed circle diameter
 - Circle: No input
- B Width across flats:
 - For Q=1, Q=2: Remaining thickness (remaining material)
 - Rectangle: Rectangle width
 - Square, polygon (Q>=4): Width across flats (use only for even number of surfaces; program "B" as an alternative to "L")
 - Circle: No input







А

Т

ID

S

F

RE Rounding radius (default: 0) ■ Polygon (Q>2): Rounding radius ■ Circle (Q=0): circle radius Angle to X axis (default: 0) Polygon (Q>2): Position of figure Circle: No input G14 Tool change point (see page 154) Turret pocket number Tool ID number Spindle speed/cutting speed Feed per revolution



Cycle parameters (second input window)

- Contour-parallel oversize 1 Κ Infeed-direction oversize X2 Limit diameter Ρ Infeed depth (default: total depth of one infeed) FΖ Approach feed (infeed rate) (default: active feed rate) Е Reduced feed rate for circular elements (default: active feed rate) U Overlap factor (range: 0 < U < 1; default 0,5) 0 Roughing or finishing 0: Roughing ■ 1: Finishing Н Cutting direction
 - 0: Up-cut milling
 - 1: Climb milling
- SCI Safety clearance in the working plane
- SCK Safety clearance in infeed direction (see page 154)
- ΜT 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

F

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 154)
- G14 Tool change point (see page 154)
- 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



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 angle)
- 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: Rectangle length
 - Square, polygon: Edge length
 - Polygon: L<0 inscribed circle diameter</p>
 - Circle: No input
 - Rectangle width
 - Rectangle: Rectangle width
 - Square: L=B
 - Polygon, circle: No input
- RE Rounding radius (default: 0)
 - Rectangle, square, polygon: rounding radius
 - Circle: circle radius
- 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 154)









В

- 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)
 - O: From the inside out (from the inside towards the outside)
 - 1: From the outside in (from the outside towards the inside)
- R 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
- RB Retraction plane
- SCI Safety clearance in the working plane
- SCK Safety clearance in infeed direction (see page 154)







- 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 347).
- Milling cutter radius 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 approach 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 **approach 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 **approach 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

4.8 Milling cycles

ICP contour, radial



Χ, Ζ Starting point

É

- С Spindle angle (C-axis position)
- X1 Milling top edge (diameter)—(default: starting point X) P2 Milling depth
- L Contour-parallel oversize
- Κ Infeed-direction oversize
- Ρ Infeed depth (default: total depth of one infeed)
- FΖ Approach feed (infeed rate) (default: active feed rate)
- Е Reduced feed rate for circular elements (default: active feed rate)
- FΚ ICP contour number
- G14 Tool change point (see page 154)
- Т Turret pocket number
- ID Tool ID number
- S Spindle speed/cutting speed
- F Feed per revolution

Cycle parameters (second input window)

- Ο Roughing or finishing-only for pocket milling
 - 0: Roughing
 - 1: Finishing
 - 2: Deburring
- Н 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 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
- RB Retraction plane
- SCI Safety clearance in the working plane
- SCK Safety clearance in infeed direction (see page 154)
- 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 347).
- Milling cutter radius 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 approach radius R.



ſ

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 **approach 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 **approach 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 thread 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

- Χ, Ζ Starting point
- С Spindle angle (C-axis position)
- X1 Thread diameter
- C1 Starting angle
- Ζ1 Starting point of thread
- Ζ2 End point of thread
- F1 Thread pitch
 - F1 positive: Right-hand thread
 - F1 negative: Left-hand thread
- U Thread depth
- Т 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:"l"
 - Infeed n: I * (1 (n–1) * E)
- Е Cutting depth reduction
- Ρ Run-in length (ramp at the beginning of the slot)
- Κ Run-out length (ramp at the end of the slot)
- G14 Tool change point (see page 154)
- Т 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 154)
- ΜT 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.





- 4.8 Milling cycles
- 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: $\ensuremath{\textbf{Milling}}$

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 fo	r contour milling Cutting direction	Direction of tool rotation	MCRC	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 contour milling



1

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			

i

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)

- TO = 8 (tool orientation)
- I = 8 (milling diameter)
- \blacksquare 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 set and more information, see page 354

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:

4.8 Milling cycles

- X Starting point (diameter): Pre-positioning the tool
- Z Starting point: Pre-positioning the tool
- C Spindle angle: Pre-positioning the workpiece spindle
- 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 154)
- 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 154)
- 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 lateral surface of the workpiece. For character set and more information, see page 354

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): Pre-positioning the tool
- Z Starting point: Pre-positioning the tool
- C Spindle angle: Pre-positioning the workpiece spindle
- TX Text to be engraved
- NF Character number: ASCII code of the character to be engraved
- X2 End point (diameter): 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 154)
- 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 154)
- 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 \boldsymbol{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 MANUALplus 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	Capita	l letters	Num diacri	erals, tics	Specia charac	al sters	
NF	Character	NF	Character	NF	Character	NF	Character	Meaning
97	а	65	А	48	0	32		Space
98	b	66	В	49	1	37	%	Per cent sign
99	С	67	С	50	2	40	(Opening parenthesis
100	d	68	D	51	3	41)	Closing parenthesis
101	е	69	E	52	4	43	+	Plus character
102	f	70	F	53	5	44	1	Comma
103	g	71	G	54	6	45	-	Minus sign
104	h	72	Н	55	7	46	•	Point
105	i	73		56	8	47	/	Forward slash
106	j	74	J	57	9	58	:	Colon
107	k	75	К			60	<	Less than character
108		76	L	196	Ä	61	=	Equal sign
109	m	77	Μ	214	Ö	62	>	Greater than character
110	n	78	Ν	220	Ü	64	@	at
111	0	79	0	223	ß	91	[Opening brackets
112	р	80	Р	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	Т			186	0	Degrees
117	u	85	U			215	*	Multiplication sign
118	V	86	V			33	!	Exclamation point
119	W	87	W			38	&	Ampersand and
120	Х	88	Х			63	?	Question mark
121	У	89	Y			174	®	Trademark
122	Z	90	Z			216	Ø	Diameter sign

4.9 Drilling and milling patterns

Note on using drilling/milling patterns:

- Hole pattern: The MANUALplus 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, 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 371).

Drilling pattern linear, axial



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 angle)
- 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
- li, Ji: Pattern spacing (incremental)

The parameters of the hole are also requested.

Use the following parameter combinations to define the:

Starting point of pattern:

- X1, C1 or
- XK, YK
- Pattern positions:
 - li, Ji and Q
 - I, J and Q







4.9 Drilling and milling patterns

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
 - In Machine mode: machining 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



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 angle)
- 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
- li, 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





4.9 Drilling and milling patterns

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
 - In Machine mode: machining 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



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 angle)
- 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 1 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






4.9 Drilling and milling patterns

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
 - In Machine mode: machining 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





i

Milling pattern circular, axial



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 angle)
- 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 1 slot (default: 0°)
- Wi Angle increment (pattern spacing)—(default: milling operations are regularly spaced 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
 - In Machine mode: machining 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







i

Drilling pattern linear, radial



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 angle)
- Q Number of holes
- Z1 Starting point of pattern (position of 1st hole)
- ZE End point of pattern (default: Z1)
- C1 Angle 1 hole (starting angle)
- Wi Angle increment (pattern spacing)—(default: holes are regularly spaced on the lateral surface)

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 are also requested.







Cycle run

- **1** Positioning (depending on the machine configuration):
 - \blacksquare Without C axis: Position to spindle angle C
 - With C axis: Activate C axis and position to spindle angle C at rapid traverse
 - in **Machine** mode: machining 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
- ${\bf 7}$ $\,$ Position to starting point ${\bf Z}$ and deactivate C axis
- 8 Move to the tool change point according to the G14 setting

Milling pattern linear, 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 angle)
- Q Number of slots
- Z1 Starting point of pattern (position of 1st slot)
- ZE End point of pattern (default: Z1)
- C1 Angle 1 slot (starting angle)
- Wi Angle increment (pattern spacing)—(default: milling operations are regularly spaced on the lateral surface)

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
 - in **Machine** mode: machining 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







4.9 Drilling and milling patterns

Drilling pattern circular, radial



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 angle)
- Q Number of slots
- ZM, CM Center of pattern: position, angle
- K Pattern diameter
- A Angle 1 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
 - in **Machine** mode: machining 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



Milling pattern circular, radial

MILLING PA	ATTERN CIRCULAR, RADIAL		2
	Select milling	øx tr	3 Wi A
	Select slot, radial		
	Select ICP contour, radial		z
Pattern circular	Press the Pattern circular soft key		
			4

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 angle)
- Q Number of slots
- ZM, CM Center of pattern: position, angle
- K Pattern diameter
- A Angle 1 slot (default: 0°)
- Wi Angle increment (pattern spacing)—(default: milling operations are regularly spaced 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.



— ZM——

Q=6



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
 - in **Machine** mode: machining 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
- ${\bf 7}$ $\,$ Position to starting point ${\bf Z}$ 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, radial** 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

- TO = 8 (tool orientation)
- DV = 5 (drilling diameter)
- BW = 118 (point angle)
- AW = 1 (The tool is a driven tool)







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

- TO = 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, radial** 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

- TO = 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

DIN

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 **Machine** 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 cutting**
- Recessing tool: Contour recessing
- Twist drill: **Drilling**
- Indexable insert drill: **Predrilling**
- 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.

4.10 DIN cycles

i



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 the ICP cycles (Teach-in submode, Machine) mode

■ in the smart.Turn mode of operation

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 the cycle dialogs.

ICP contours created in **cycle mode** are saved by the MANUALplus 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 MANUALplus integrates ICP contours created in **smart.Turn** mode 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 can only be edited with ICP.

In smart.Turn mode, contours are part of the NC program. They can be machined with the ICP editor or smart.Turn editor.



In machine parameter 602023, you define whether the control loads the programmed or calculated values into the NC program.



Loading contours

ICP contours that you have created **for cycle programs** can be loaded in **smart.Turn** mode. 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** submode. 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 the **smart.Turn** mode of operation.

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. Small 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, and small deviations in horizontal elements (X direction) are tolerated.

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 397).



Machining attributes

You can assign the following attributes to the contour elements:

Parameters			
U	Oversize (additive to other oversizes)		
F	The ICP generates a G52 Pxx H1. Special feed rate for finishing.		
D	The ICP generates a G95 Fxx. Number of the additive D compensation for the finishing operation (D = 01 to 16).		
FP	The ICP generates a G149 D9xx. Edit element during automatic program generation with TURN PLUS (not available in Teach-in submode)		
	■ 0: No ■ 1: Yes		
IC	Oversize for measuring cut (not available in Teach-in submode)		
KC	Length for measuring cut (not available in Teach-in submode)		
HC	Measuring cut counter: Number of workpieces after which a measurement is performed (not available in Teach-in submode)		



The machining attributes are effective only for the element in which the attributes have been entered in ICP.

Calculation of contour geometry

The MANUALplus 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 submode 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

ICP edit 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 406.
- Contour for the face: See "Face contours in smart.Turn mode" on page 431.
- Contour for the lateral surface: See "Lateral surface contours in smart.Turn mode" on page 439.



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 submode. The input field FK has to be empty when you call the ICP editor submode.
- Load the existing contour. When you exit the ICP editor the name of the contour last edited is loaded into the input field FK.

TCP adj	tor	S court Ti		Tool of	iter		
- Ich eur	.01	V Smart. It	1111	H loor ed	1101		
		шш.,,,,,,,,					
HI Contour :	Henanipulate	Attrabute					
Select LUP C	contours (IN	S:\Project\BH	B_KAPITELS\gt	1()			
Name			 Size 	Changed on	Att		
Ø 1kreise.g	mi		9 kB	38-10-2014 1	10:34		
2schraube	.gm1 i		21 KB 21 kB	30-10-2014 1	10:34		
Cattoring	-						
							510 DOM
							OFF ON
							17 M07
							OFF
							MOO
							Pied
) 🔊 🖂
File name: *.gmi					<u>H</u>		
TNC:\Project	\BHB_KAPITEL5	\gti\1kreise.	gmi				10:35
Details	Next file type	File manager	Sorting	Project	Alphabetic keyboard	Open	Cancel

Creating a new contour

Edit ICP	Define the contour name in the cycle dialog and press the Edit ICP soft key. The submode ICP editor switches to entering the contour.
Edit ICP	Press the Edit ICP soft key. The submode ICP editor opens the window "Selection of ICP contours."
Open	Define the contour name in the "file name" field and press the 0pen soft key. The submode ICP editor switches to entering the contour.
	Press the Contour menu key.
Insert element	Press the Insert element soft key.
The ICP wa	its for you to enter a contour name.

File organization with the ICP editor submode

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 submode opens the "Selection of ICP contours" window.
File manager	Press the File manager soft key. The ICP editor submode switches the soft-key row to the file organization functions.

5.3 ICP editor submode in smart.Turn mode of operation

In the smart.Turn submode you create:

- Contour groups
- 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

Contour groups: The control supports up to four contour groups (workpiece blank, finished part and auxiliary controls) in an NC program. The identifier CONTOUR GROUP initiates the description of a contour group (see "Contour groups" on page 484).

Blank contours and auxiliary blank contours: You describe complex blanks element by element—like finished parts. You select the standard forms bar and tube by menu and describe them with a few parameters (see "Defining the workpiece blank" on page 405). If a definition of the finished part exists, you can also select "Cast part" in the menu.

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.



Cycle programming contours can be loaded and integrated into the smart.Turn program. **smart.Turn** supports loading 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.

In machine parameter 602023, you define whether the control loads the programmed or calculated values into the NC program.

1

Machining a contour in smart.Turn mode

Creating a blank contour



The ICP waits for you to enter a contour name.

→ Machine	smart.Turn	🎦 Tool editor	
Prog Head	ICP 💶 Units» 📕 Go to	Config 📕 Misc 🙀 Extras	Graph.
bar.nc	Contour editing		
&bar.nc "TURN_V:	Workpiece blank		
HEADER	Finished part		
#MEASURE_UNITS	New auxiliary blank		
TURRET	New aux. contour		
T1 ID"001"	📑 Caxis 🕨 🕨 🕨 א 🕨 א א א א א א א א א א א א א	MD4 XE100 ZE50 RS0.8 EW95 SW	80] - END PGM
CLAMPS	¥ axis		DEF
	Insert contour		B M
CLAMPS 2	,		
BLANK			
N 1 G20 X50 Z41	К1		
FINISHED			
N 2 G0 X0 Z0 N 3 G1 X20			
N 4 G1 Z-10			
N 5 G1 X40 Z-20			z
TNC:\Project\BHB_}	APITEL5\ncps\bar.nc		18:35
1			

Loading a contour from the cycle editing





Press the **Contour 1ist** soft key. The **ICP editor** submode 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.



The **ICP editor** submode 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 submode main menu Opens the file selection dialog for ICP Contour contours. list Inverts the definition direction of the Turn contour. contour Retroactively inserts form elements. p-97 Opens the soft-key menu for the zoom functions and displays the zoom frame. Clears an existing element. Delete element Modifies an existing element. Change element Adds an element to the existing Insert contour. element Returns to the dialog that has called Back ICP.



Entering an ICP contour

If the contour is new, the MANUALplus asks first for the coordinates of the **contour starting point**.

Linear contour elements: 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 MANUALplus 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 item.

Press the Contour menu item

Press the **Insert element** soft key.

Menu items of the lines menu				
∰ ヽ ∰ ↗	Line with angle in displayed quadrant			
■ ← # →	Horizontal line in displayed direction			
₩ < ₩ >	Line with angle in displayed quadrant			
## ↑ ## ↓	Vertical line in displayed direction			
**レーベベ	Call the form element menu			
Menu items of arcs menu				
∰C ∰D	Circular arc in the displayed direction			

Soft keys for switching between lines menus and arcs menus

Call the form element menu

Select the line menu.

Select the arc menu.

Specify the starting point.

ICP CONTOUR PROGRAMMING

Insert



Select the element type and enter the known parameters of the contour element.

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.).

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 MANUALplus must know the type of transition that connects the contour elements. The transition to the next contour element is determined by soft key.



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 the 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 the corresponding field of the open dialog box

Soft key switchover for incremental

Increment

Activates the incremental dimension for the current value

Soft key for tangential transition



Activates the tangential condition for the transition in the end point of the contour element

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.

Angular input

Select the desired angle input by soft key.

Linear elements

- **AN** Angle to the Z axis (AN<=90°—within the preselected quadrant)
- **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 polar coordinates				
W	Switches the field to entering the angle W .			

Soft keys for angular input



P

Angle to successor



Angle to predecessor



Contour graphics

As soon as you have entered a contour element, the MANUALplus 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 MANUALplus 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 MANUALplus 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 MANUALplus 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.

C	Activate the section selection.
	Press the Next element soft key until the entire section is marked.
Ľ¢,	Press the Previous element soft key until the entire section is marked.

Ľ?	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.
<u>د</u>	Select range : Activates the range selection.
Selecting	g contour corners (for form elements)
ĽQ	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.
Ľ~1	Mark all corners: Marks all contour corners.
Ľ~_	Corner selection : If the corner selecting function is activated, multiple contour corners can be marked.
Mark	Mark: If the corner selecting function is active, individual contour corners can be selected and marked or unmarked with this soft key.

Selecting contour elements

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



After you exit **ICP editor** submode, the zero point shift cannot be reset. 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 doing a 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

Parameters

Q Number of repeats





5.4 Creatin<mark>g a</mark>n ICP contour

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

Parameters

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 521.) The cycle used determines whether machining is transverse or longitudinal.

If the contour is described in the **–X direction**, a transverse machining cycle or a tool with the orientation 3 must be used.

- ICP cut, longitudinal/transverse (roughing): The MANUALplus machines the workpiece in the contour direction.
- ICP finishing, longitudinal/transverse: The MANUALplus 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 submode main menu

Turn contour Inverts the definition direction of the contour.


5.5 Editing ICP contours

The MANUALplus offers the following possibilities for extending or changing a programmed contour.

Superimposing form elements





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.





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



You can delete several successive contour elements.

Editing contour elements

The MANUALplus 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



Make the changes.



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).



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 the Manipulate menu key. The menu displays functions for trimming, editing and deleting contours.
	Press the menu keys Edit Contour element.
2	Select the contour element to be edited.
-Liter	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 MANUALplus 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.

Parameters

- 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 maintained0: 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-P	Polar dimension of the end point: Angle	
W.	Polar dimension of the end point: Radius	

ſ

5.6 Zoom functions in ICP editor submode

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 MANUALplus 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 mo	difying the displayed section
ţ	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		
t t	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 ${\tt smart.Turn}$ mode of operation, 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)

In smart.Turn, ICP generates 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

In smart.Turn, ICP generates a G20 in the BLANK section.

"Cast part" blank

The function describes an oversize on an existing finished part contour.

Parameters

K Contour-parallel oversize

In smart.Turn, ICP generates a contour in the BLANK section.





405



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 mode

- 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.





Х

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 contour element.

Parameters

Х	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 380

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 380

In smart.Turn, ICP generates a G1.





Line at angle

ĸ

2



Select the line direction.

Enter the line dimensions and define the transition to the next contour element. Always enter the ${\tt angle}~{\tt AN}~(<=90^\circ)$ 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 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 380
- 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 contour 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)
Р	Polar target point (radius)
Pi	Incremental polar target point (distance from starting point to target point)
I, K	Center of arc
li, 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 of arc (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, FF	P: See machining attributes on Page 380
ICP generation	ates a G2 or G3 in smart.Turn







1

Contour form elements

Chamfer/rounding arc





Chamfer/rounding arc as first element: Enter element position AN.

Enter the chamfer width BR or the rounding radius BR.

Parameters

BB	Chamfor	width	/rounding	radiue
DN	Chamler	wiath	/rounding	radius

AN Element position

U, F, D, FP: See machining attributes on Page 380

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** mode 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



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 on Page 380

In smart.Turn, ICP generates a G25.

The MANUALplus calculates unentered parameters automatically from the standard table (see "DIN 76—undercut parameters" on page 617):

- The "thread pitch FP" is determined from the diameter.
- The parameters I, K, W, and R are calculated from the thread pitch FP.



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



- W Undercut angle (default: value from standard table)
- U, F, D, FP: See machining attributes on Page 380

In smart.Turn, ICP generates a G25.

The MANUALplus calculates unentered parameters automatically from the diameter in the standard table (see "DIN 509 E – undercut parameters" on page 619).



Undercuts can be programmed only between two linear elements. One of the two linear elements must be parallel to the X axis.





ICP programming

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 on Page 380

In smart.Turn, ICP generates a G25.

The MANUALplus calculates unentered parameters automatically from the diameter in the standard table (see "DIN 509 F – undercut parameters" on page 619).



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



U, F, D, FP: See machining attributes on Page 380

In ${\tt smart.Turn},$ 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.

ICP programming

i

Undercut type H



Parameters

- R Undercut radius
- W Plunge angle

U, F, D, FP: See machining attributes on Page 380

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



Enter the undercut parameters.

Parameters

- I Undercut depth
- R Undercut radius
- W Angular length
- A Plunge angle

U, F, D, FP: See machining attributes on Page 380

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
- In smart.Turn mode: 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.

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.



Soft keys for polar coordinates



Switches the field to entering the angle $\boldsymbol{C}.$



Switches the field to entering the radius **P**.



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 Retraction plane

In smart.Turn, ICP generates a G100.

Vertical lines on face

5.9 Contour <mark>ele</mark>ments on face ╡╡↓

1 1 Select the line direction.

Enter the line dimensions and define the transition to the next contour element.

Parameters

- YΚ Cartesian target point
- YKi Incremental target point (distance from starting point to target point)
- С Polar target point (angle)
- Ρ Polar target point
- L Length of line
- F: See machining attributes on Page 380

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

- ΧК Cartesian target point
- XKi Incremental target point (distance from starting point to target point)
- С Polar target point (angle)
- Ρ Polar target point
- L Length of line
- F: See machining attributes on Page 380

In smart.Turn, ICP generates a G101.



YK 6 L=YKi YK XK \rightarrow

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) С
- Polar target point (angle)
- Ρ 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 on Page 380

In smart.Turn, ICP generates a G101.





HEIDENHAIN MANUALplus 620

Circular arc on face



Select the arc's direction of rotation.

Enter the arc dimensions and define the transition to the next contour element.

Parameters

XK, YK	Target point (end point of the circular arc)
XKi, YKi	Incremental target point (distance from starting point to target point)
Р	Polar target point (radius)
Pi	Incremental polar target point (distance from starting point to target point)
С	Polar target point (angle)
Ci	Incremental polar target point and angle (relative to the starting point)
I, J	Center of arc
li, Ji:	Incremental center of arc (distance from starting point to center in X, Z)
PM	Polar center of arc
PMi	Incremental polar center of arc (distance from starting point to center)
СМ	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 ma	chining attributes on Page 380
ICP genera	ates a G102 or G103 in smart.Turn







ĺ

Chamfer/rounding arc on face



Chamfer/rounding arc as first element: Enter element position AN.

Parameters

BRChamfer width/rounding radiusANElement position

F: See machining attributes on Page 380

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** mode 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
- In smart.Turn mode: 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.

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.



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.



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 Retraction 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 on Page 380

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 on Page 380

In smart.Turn, ICP generates a G111.



Å

CY

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 on Page 380

In smart.Turn, ICP generates a G111.



Circular arc on lateral surface

#C ∰

5 Select the arc's direction of rotation.

Enter the arc 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)
Р	Target point as polar radius
С	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)
К	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 of arc (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 ma	chining attributes on Page 380

ICP generates a G112 or G113 in smart.Turn







i

Chamfer/rounding arc on lateral surface



Chamfer/rounding arc as first element: Enter **element position AN**.

Parameters

BRChamfer width/rounding radiusANElement position

F: See machining attributes on Page 380

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 ${\tt smart.Turn}$ mode 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 mode

In smart.Turn mode, 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:

- Caxis
 - 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 MANUALplus 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.



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.	

HEIDENHAIN MANUALplus 620



Representation of the ICP elements in the smart.Turn program

Each ICP dialog in **smart.Turn programs** is represented by a section code followed by 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):
 - FACE_C (XC plane)
 - LATERAL_C (ZC plane)
 - FACE_Y (XY plane)
 - LATERAL_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 edit 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. Beispiel: "Rectangle on the face"

FACE_C ZO
N 100 G308 ID"FACE_1" P-5
N 101 G305 XK40 YK10 A0 K30 B15
N 102 G309

Beispiel: "Nested figures"

· · · ·
FACE_C ZO
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 mode

In ${\tt smart.Turn},$ 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 416.

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 429).

ICP generates:

- The FACE_C 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 submode (AWG).

Parameters for defining the starting point

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 Retraction plane

Circle on face

Reference data of face

- ID Contour name
- PT Milling depth
- ZR Reference dimension

Parameters of figure

XKM, YKMCenter of figure (Cartesian coordinates)RRadius

You can find the ${\it reference\ dimension\ ZR}$ with the "select reference plane" function (see page 429).

ICP generates:

- The FACE_C 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.



HC
5.12 Face contours in smart.Turn mode

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 429).

ICP generates:

- The FACE_C 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.





Polygon on face

Reference data of face

Parameters of figure

Contour name

Reference dimension

Number of corners

Length of edge

Rounding arc

Position angle (reference: XK axis)

Milling depth

5.12 Face contours in <mark>sm</mark>art.Turn mode XKM, YKM Center of figure (Cartesian coordinates)

ID

ΡT

ZR

А

Q

Κ

Ki

ΒR



You can find the reference dimension ZR with the "select reference plane" function (see page 429).

Width across flats (inscribed circle diameter)

ICP generates:

- The FACE_C 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.



5.12 Face contours in <mark>sm</mark>art.Turn mode

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 ${\it reference\ dimension\ ZR}$ with the "select reference plane" function (see page 429).

ICP generates:

- The FACE_C 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)
- O2 Direction of rotation
 - CW
 - CCW
- B Width

You can find the **reference dimension ZR** with the "select reference plane" function (see page 429).

ICP generates:

- The FACE_C 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.
- \blacksquare A G302 or G303 with the parameters of the figure.









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

Centering	
0	Diameter
Hole	
В	Diameter
BT	Depth (without algebraic sign)
W	Angle
Countersin	K
R	Diameter
U	Depth
E	Sinking angle
Thread	
GD	Diameter
GT	Depth
К	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 ${\it reference\ dimension\ ZR}$ with the "select reference plane" function (see page 429).

- The FACE_C section code with the reference dimension parameter. In nested contours, ICP generates only one section code.
- \blacksquare A G308 with the parameters contour name and hole depth (–1*BT).
- A G300 with the parameters of the hole.
- A G309.







5.12 Face contours in smart.Turn mode

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 429).

ICP generates:

- The FACE_C 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 or hole depth (-1*BT).
- A G401 with the parameters of the pattern.
- The G function and the parameters of the figure/hole.

A G309.







437

Circular pattern on face

Reference	data of face	YK	
ID	Contour name	Ą	EP
PT	Milling depth		
ZR	Reference dimension		
Parameter	s of pattern		AP
ΧΚ, ΥΚ	Center of pattern (Cartesian coordinates)		
QP	Number of points in pattern	■	DP
DR	Direction of rotation (default: 0)		
	DR=0. without EP: Figures are arranged on a full circle	·	
	\square DB=0, with EP. Figures are arranged on the longer	T	—— XK —
	circular arc		
	DR=0, with EPi: The algebraic sign of EPi defines the direction (EPi<0: clockwise)	YK	
	DR=1, with EP: clockwise	Å	
	DR=1, with EPi: clockwise (algebraic sign of EPi has no effect)		
	DR=2, with EP: counterclockwise		AP
	 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°)	I	
EPi	Angle between two figures		
Н	Element position	H=0	Н
	 0: Normal – the figures are rotated about the circle center (rotation) 		
		<u>ل</u>	

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 429).

- The FACE_C 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 or hole 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 mode

In ${\tt smart.Turn},$ 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 422.

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 429). The reference diameter is used for converting angle dimensions into linear dimensions.

- The LATERAL_C section code with the reference diameter 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 submode (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
- ΒR Chamfer width
- RΒ Retraction plane

1

5.13 Lateral surface contours in <mark>sm</mark>art.Turn mode

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 429).

ICP generates:

- The LATERAL_C section code with the reference diameter 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.





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 ${\tt reference\ diameter\ XR}$ with the "select reference plane" function (see page 429).

ICP generates:

- The LATERAL_C section code with the reference diameter 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.



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 ${\it reference\ diameter\ XR}$ with the "select reference plane" function (see page 429).

- The LATERAL_C section code with the reference diameter parameter. In nested contours, ICP generates only one section code.
- A G308 with the parameters contour name and milling depth.
- \blacksquare 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 429).

ICP generates:

- The LATERAL_C section code with the reference diameter 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.



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 429).

ICP generates:

- The LATERAL_C section code with the reference diameter 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.







Hole on lateral surface

5.13 Lateral surface contours in <mark>sm</mark>art.Turn mode

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 429).

- The LATERAL_C section code with the reference diameter parameter. In nested contours, ICP generates only one section code.
- A G308 with the parameters contour name and hole depth (–1*BT).
- A G310 with the parameters of the hole.
- 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 429).

- The LATERAL_C section code with the reference diameter parameter. In nested contours, ICP generates only one section code.
- A G308 with the parameters contour name and milling depth or hole 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 439)

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)</p>
 - 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 429).

- The LATERAL_C section code with the reference diameter parameter. In nested contours, ICP generates only one section code.
- A G308 with the parameters contour name and milling depth or hole 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 $\ensuremath{\mathsf{smart.Turn}}$, 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.

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 429).

ICP generates:

- The FACE_Y section code 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.

Soft keys for polar coordinates



Switches the field to entering the angle **W**.



Switches the field to entering the radius **P**.





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.



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 on Page 380

In smart.Turn, ICP generates a G171.



γ

- XS

Ϋ́S

Х

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 on Page 380

In **smart.Turn**, ICP generates a G171.



Line at angle in XY plane

Y



➤ Select the line direction.

Enter the line dimensions and define the transition to the next contour element.

Parameters

∠

		_	
Х	Y	Larget	noint
· `,		ruigot	point

	5 1
Xi, Yi	Incremental target point (distance from starting point to target point)
W	Polar target point (angle)
Ρ	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 on Page 380	
In smart.Turn, ICP generates a G171.	





i

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

Х, Ү	Target point (end point of the circular arc)	
Xi, Yi	Incremental target point (distance from starting point to target point)	
Р	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	
li, Ji:	Incremental center of arc (distance from starting point to center in X, Z)	
PM	Polar center of arc	
PMi	Incremental polar center of arc (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 ma	chining attributes on Page 380	
ICP generates a G172 or G173 in smart.Turn		







1

Chamfer/rounding arc in XY plane



Chamfer/rounding arc as first element: Enter element position AN.

Parameters

BR	Chamfer width/rounding radius	
AN	Element position	
F: See machining attributes on Page 380		

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** mode 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 429).

ICP generates:

- The FACE_Y section code 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.



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

Y YM B W XM XM

You can find the **reference dimension ZR** and the **limit diameter IR** with the "select reference plane" function (see page 429).

ICP generates:

The FACE_Y section code 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.

ICP programming

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 429).

- The FACE_Y section code 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 429).

ICP generates:

- The FACE_Y section code 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.



5.14 Contour<mark>s in</mark> the XY plane

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)
- O2 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 429).

ICP generates:

- The FACE_Y section code 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.







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 Spindle angle Limit diameter
 - Reference dimension

Parameters of the hole

XM, YM Center of hole

Centering

Countersink

R

U

F

GT

Κ

F

GΑ

Thread GD

Diameter

Diameter

Diameter

Diameter

Depth

Sinking angle

Run-out length

0: Right-hand thread ■ 1: Left-hand thread

Thread pitch

Depth

Angle

Depth







You can find the reference dimension ZR and the limit diameter IR with the "select reference plane" function (see page 429).

Thread type (right-hand/left-hand)

ICP generates:

- The FACE_Y section code 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 hole depth (-1*BT).
- A G370 with the parameters of the hole.

5.14 Contours in the XY plane

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, JPiDistance between two pattern points (in X or Y direction)APPosition 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 429).

ICP generates:

- The FACE_Y section code 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 or hole depth (-1*BT).
- A G471 with the parameters of the pattern.
- The G function and the parameters of the figure/hole.







Circular pattern in XY plane

Reference data: (see "Reference data in XY plane" on page 449)

Reference of ID PT C IR ZR Parameters X, Y QP	data in XY plane Contour name Milling depth Spindle angle Limit diameter Reference dimension of pattern Center of pattern Number of points in pattern	
DP	 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 (algebraic sign of EPi has no effect) PR=2, with EPi: counterclockwise (algebraic sign of EPi has no effect) Pattern diameter 	
AP EP H	 Starting angle (default: 0°) End angle (no entry: the pattern elements are equally divided into 360°) Angle between two figures 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) 	H=0

You can find the reference dimension ZR and the limit diameter IR with the "select reference plane" function (see page 429).

- The FACE_Y section code 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 or hole 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 surface

- 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 429).

ICP generates:

- The FACE_Y section code 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.





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

- Reference edge
- Q Number of sides (Q>=2)
- K Width across flats
- Ki Length of side
 - 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 429).

ICP generates:

- The FACE_Y section code 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.









Switches the field to entering the width across flats K.

Ζ

В

5.15 Contours in the YZ plane

In $\ensuremath{\mathsf{smart.Turn}}$, 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.

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 429).

ICP generates:

- The LATERAL_Y section code with the parameters 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.

Soft keys for polar coordinates



Switches the field to entering the angle **W**.



Switches the field to entering the radius **P**.





TURN PLUS attributes

In the TURN PLUS attributes you can define settings for the automatic program generation submode (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
- ΒR Chamfer width
- RΒ Retraction plane

1

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.



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

₩↓ 1

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 on Page 380

In smart.Turn, ICP generates a G181.



> Z

Y

467



Horizontal lines in YZ plane



Select the line direction.

Enter the line dimensions and define the transition to the next contour element.

Parameters

5.15 Contours in the YZ plane

- 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 on Page 380

In smart.Turn, ICP generates a G181.



i
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 on Page 380

In smart.Turn, ICP generates a G181.







5.15 Contours in the YZ plane

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)
Р	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) $% \left({{{\rm{D}}_{{\rm{A}}}}_{{\rm{A}}}} \right)$
PM	Polar center of arc
PMi	Incremental polar center of arc (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 ma	chining attributes on Page 380
ICP gener	ates a G182 or G183 in smart.Turn









470



Chamfer/rounding arc in YZ plane



Chamfer/rounding arc as first element: Enter element position AN.

Parameters

BRChamfer width/rounding radiusANElement position

F: See machining attributes on Page 380

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** mode 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 429).

ICP generates:

- The LATERAL_Y section code with the parameters 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 429).

ICP generates:

- The LATERAL_Y section code with the parameters 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 ${\it reference\ diameter\ XR}$ with the "select reference plane" function (see page 429).

ICP generates:

- The LATERAL_Y section code with the parameters 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

Je
ar
d
N
the
2
S
5
3
0
Ť
0
Ō
$\mathbf{\nabla}$
ß
_

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 429).

ICP generates:

- The LATERAL_Y section code with the parameters 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.



5.15 Contour<mark>s in</mark> the YZ plane

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)
- O2 Direction of rotation
 - CW
 - CCW
 - Width

You can find the ${\it reference\ diameter\ XR}$ with the "select reference plane" function (see page 429).

ICP generates:

В

- The LATERAL_Y section code with the parameters 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

5.15 Contours in the YZ plane

Reference data of hole

- ID Contour name С Spindle angle
- XR Reference diameter

Parameters of the hole

/M	7M	Center	of	hole
ι ivi,		Center	01	IIUIE

Centering

Ο

- Hole В Diameter
- ΒT Depth Angle

Countersink

R	Diameter
U	Depth
E	Sinking angle
Thread	
GD	Diameter
GT	Depth
К	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 429).

ICP generates:

- The LATERAL_Y section code with the parameters reference diameter and spindle angle. The section code is omitted for nested contours.
- A G308 with the parameters contour name and hole depth (-1*BT).
- A G380 with the parameters of the hole.

■ A G309.







5.15 Contours in the YZ plane

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, Z1st point of patternQPNumber of points in patternJP, KPEnd point of pattern (Cartesian coordinates)JPi, KPiDistance between two pattern points (in Y or Z direction)APPosition angleRPTotal length of patternRPiDistance 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 429).

ICP generates:

- The LATERAL_Y section code with the parameters reference diameter and spindle angle. The section code is omitted for nested contours.
- A G308 with the parameters contour name and milling depth or hole 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

lane
d
N
>
e
÷
2
S
Ľ'n
ō
Ę
ō
0
2
-/

Reference of	lata in YZ plane
ID	Contour name
PT	Milling depth
С	Spindle angle
XR	Reference diameter
Parameters	of pattern
Υ, Ζ	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
Н	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 $\mbox{reference diameter XR}$ with the "select reference plane" function (see page 429).

ICP generates:

- The LATERAL_Y section code with the parameters reference diameter and spindle angle. The section code is omitted for nested contours.
- A G308 with the parameters contour name and milling depth or hole 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 surface

- 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 $\ensuremath{\text{reference}}$ diameter XR with the "select reference plane" function (see page 429).

ICP generates:

- The LATERAL_Y section code with the parameters 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 Gepth 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

- Reference edge
- Q Number of sides (Q>=2)
- K Width across flats
- Ki Length of side
 - 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 429).

ICP generates:

- The LATERAL_Y section code with the parameters 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.









Switches the field to entering the width across flats K.

Ζ

В

5.16 Loading existing contours

Integrating cycle contours in smart.Turn mode

ICP contours that you have created **for cycle programs** can be loaded in **smart.Turn** mode. 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** submode 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**.

Extension	Group
*.gmi	Turning contours
*.gmr	Contours of workpiece blanks
*.gms	Milling contours on face
*.gmm	Milling contours on lateral surface



- Blank or finished part contour: Supplement or adapt the contour, if necessary.
- **C axis contour:** Complete the reference data.



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 submode.

Contour list	Press the Contour list soft key. The ICP editor submode opens the "Selection of ICP contours" window.			
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. 				



5.17 Contour groups

Contour groups in smart.Turn mode

The control supports up to four contours in one NC program. The identifier CONTOUR GROUP initiates the description of a contour group.

You can create a workpiece, finished piece and auxiliary contours for each contour group. The **ICP editor** submode considers the shift programmed in the specific contour group for the description and display.

Display in NC program

If you have activated graphics in the NC program, the control displays the element on which the cursor is positioned when navigating through the contour description.

The control shows the number of the contour group at top left in the graphic window.

Display in unit programming

If you program an ICP unit in smart.Turn, the control displays ICP contours. The various contours and contour groups can be displayed if a contour has not been selected in the FK parameter.

Navigation keys Switches to the next/previous contour (contour group/workpiece blank/auxiliary contour/ finished part).



Switches to the next contour element.

The control shows the number of the contour group and name of the auxiliary contour if applicable at top left in the graphic window.

➡ Machine		🚯 smart.Tu	ırn	🖹 Tool	editor		
Prog 👖 Hea	d 拱 ICP 🛉	🖽 Units» 捕	Go to 👬 Cor	ıfig 🏭 Mis	c 🚆 Extras 🏭	Graph.	
z_zKonturgru %z_zKonturgrupp	e.nc "TUR	N_V1.0"		J		4	· • •
HEADER #MATERIAL #NORKPIECE #DEPARTMENT #MEASURE_UNITS	Sta Wel HEI MET	hl le DENHAIN RIC				1	=0
TURRET T1 ID"001" T3 ID"001-	capto"				×î	<u>×</u>	
					B18-		
-128	-188	-88	-68	-48	-28 8	28 Z	
TNC:\Project\Be	nutzerhand	buch_CNC_PILC)T_en\ncps\z_	zKonturgru	ope.nc		09:21
۵ ک	Change P contour	TURN PLUS	0		Show tree	ISO Mode	ſ <u>Ŀ</u>

Achine Machine		🚯 smart.Tu	rn	🍴 Tool edi	tor	8	,
Prog III H	ead 🏭 ICP	∰Units× ∰ X	Go to ∰Cor G810 Overv	fig ∰Misc Longitudinal x w. Tool Co	Extras coughing in I ntour Cycle	Graph. CP Global	
		618 8+	Appro	XS 54 ool number	Appro Z	S 3 T 1	=
-188	-58		Auxil	eed utting speed iary c FK ur start block	< no.	F 0.4 S 220 NS 4	
		8-189-	Conto Maxim Overs Overs	ur end block m um infeed ize X ize Z	10.	NE 15 P 4 I 0.5 K 0.2	
TNC:\Project\	Benutzerhand	Houch_CNC_PILC	Conto	ır start block zKonturgruppe	no.	< 1/7 ≥	09:21
NS	NE		R		Window selection	Take over	Cancel





Submode simulation

6.1 The submode Simulation



Press this soft key to call the Simulation submode from the following modes of operation:

- In smart.Turn mode
- In Program Run submode
- In Teach-in submode
- In Machine mode (MDI cycles)

When calling from the smart.Turn mode, the Simulation submode opens the large simulation window and loads the selected program. When you call the Simulation submode 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 submode through the numeric keypad
- **Simulation window:** Displays the workpieces and the tool movements. The **Simulation** submode supports the simultaneous display of multiple views in the simulation window. Use the window selection function from the miscellaneous functions menu to 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 **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 operating modes or submodes **Program Run**, Teach-in and Machine the Simulation submode is started automatically for the current program. In smart.Turn mode, the program only is loaded. Press the soft key to start the Simulation submode.







Using the simulation submode

Charting and standing with a ft low

The **Simulation** submode is controlled by soft keys in all operating states. In addition, you can always use the menu items (numeric keys) to control the simulation, even in the small simulation window, when the menu bar is **not visible**.

otarting 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 soft key indicates that a simulation is running at	

a simulation is running at present. Pressing the soft key stops the simulation.

Starting and stopping with menu items		
	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.

3-D view in the smart. Turn operating mode



This menu item switches to the 3-D view.

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.



Soft keys with active simulation window For calling warnings. If the interpreter Warning issues warnings during simulation no.: 3 (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. In the **Continuous Run** mode, all Contin. cycles of the program are simulated in run the program run mode without stopping. In the Single Block mode, the Single simulation stops after each movement block (basic block). Opens the soft-key menu for the zoom functions and displays the zoom frame (see "Adjusting the visible section" on page 496). Switches the menu **and** the soft-key Misc. row over to the miscellaneous

functions

functions.



- You can operate the Simulation submode 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.
- Use the numeric key [6] to alternate between 2-D and 3-D simulation. The control simulates the workpiece to the end before changing.



- 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 submode Simulation with Cycle ON.

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 490).	
	Activates a startup-block search (see "Simulation with mid-program startup" on page 497).	
	Calls the time calculation (see "Time calculation" on page 499).	
	Switches between the large and the small simulation window (see "Using the simulation submode" on page 487).	
	Switches between the single and multiple window display (see "Multiple window view" on page 491).	
	Saves the contour (see "Saving the contour" on page 500).	
	Dimensioning (see "Dimensioning" on page 501).	





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 traversepath 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.

\smile	

Front face and surface windows operate with a fixed spindle position. Whereas the machine turns the workpiece, the **Simulation** submode 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.

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** submode depicts traverse paths in the windows.

- Automatic: The Simulation submode 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 submode 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.

Main view soft key

Ma	in v:	iew	
XZ	XC	ZC	

- Select the view: Turning view XZ
 - Face view XC
 - Lateral surface view ZC



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. This "falsification" is compensated by the cutting radius compensation.
- Cutting path display: The Simulation submode 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 Simulation submode 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.



% z welle

Single block ×C

Misc. function 14:21

Back

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 in simulation submode

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)



Depiction of tool holder (machine-dependent function): If the machine manufacturer saves a description of the tool holder (e.g. B axis head) and if you assign a holder, the graphic also displays the tool holder.

Material-removal graphic

The material-removal graphic 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 turning view (XZ). You activate this type of simulation by soft key (see table at right).



You can change the speed of simulation in the materialremoval graphic by using the keys in the table at right.

Soft keys for miscellaneous functions



 \mathbf{b}

Switches between wire-frame graphics and cutting-path graphics.

Switches between light-dot and cutting-edge view.



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.	

i (

3-D view



► The **3-D view** menu item switches to a perspective view and shows the programmed finished part.

With the 3-D view, you can display the blank and the finished part with all turning operations, milling contours, drilling and boring operations as well as threads in a solid-model view. Tilted Y planes and machining operations referenced to them, such as pockets or patterns, are also displayed correctly by the MANUALplus.

The MANUALplus depicts milling contours depending on the parameter **HC: Milling/drilling attribute** from **G308**. If you have selected the "contour milling," "pocket milling" or "area milling" values in this parameter, the graphic shows the respective 3-D elements. If other values are specified in the **HC** parameter or if values are missing, the control depicts the described milling contour as a blue line.

Elements that cannot be calculated by the MANUALplus are displayed as an orange line; for example, if an open milling contour is programmed as a pocket.

With the soft keys and the menu functions you can influence the display of the workpiece.

The finished part contour programmed in the **FINISHED** section is shown in the graphic regardless of the machining operation in the NC program.

You can cancel the calculation of the 3-D view by pressing the **ESC** key or the **Cancel** soft key.







The check mode enables you to verify the correct positioning of holes and milling contours, for example.

In the check mode the MANUALplus displays turning contours in gray, and drilling and milling contours in yellow. For a better overview, the control shows all contours as transparent.



Submode simulation



Rotating the 3-D view with the menu functions

With the menu functions you rotate the graphic around the displayed axes (see table at right). The "perspective view" soft key resets the graphic to its initial condition.

Rotating and moving the 3-D view with the mouse

With the right mouse button pressed you can move the displayed workpiece as required.

When you press and hold the left mouse button, you can do the following:

- Vertical movement in the simulation window: Tilts the workpiece forward or backward
- Horizontal movement in the simulation window: Rotates the workpiece horizontally around its own axis
- Horizontal or vertical movement at the edge of the simulation window (gray border): Rotates the workpiece clockwise or counterclockwise
- Movement in any direction: Rotates the workpiece in any direction

Soft keys for 3-D view

	Displays the finished part and the programmed workpiece blank.
	Displays the finished part and the updated workpiece blank.
	Activates/Deactivates the check mode
	Displays a section view.
90%	Selects the side view. Rotates the side view by 90°.
	Selects the perspective view.
Menu for the 3-D view	
C.	Tilt the graphic backward.
N	Rotate the graphic horizontally in the indicated direction.

- Rotate the graphic horizontally in the indicated direction.
- Rotate the graphic counterclockwise.
- Tilt the graphic forward.

Rotate the graphic clockwise.



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 arrow keys and the PgDn and PgUp keys to change the

visible section.

For cycle programs, and when a program is started for the first time in Simulation mode, the MANUALplus 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 OI

pening the zoom function:	
Keys for mod	lifying the displayed section
t t	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

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 f	Keys for modifying the red frame		
ŧ	t	The arrow keys move the red frame in the indicated direction.	
+	+		
PG DN		Reduces the size of the red frame.	
PG UP		Increases the size of the red frame.	



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.

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:

Misc. functions	Switch the menu bar to "Miscellaneous functions."
	Select the "Start block" menu item.
Take over	Enter the number of the startup block, then transfer the Simulation mode start-up block
Back	▶ Go back to the main menu of the Simulation
	Start the simulation. The MANUALplus scans the NO 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 midprogram startup.

Mid-program startup remains on even if you cancel the simulation. If you restart the simulation after cancellation, 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 startup block and activates 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.

i

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 **arrow keys** and the **PgUp/PgDn keys**.

Calling the machining times:



Switch the menu bar to "Miscellaneous functions."



▶ Call the time calculation.

# #	Ⅲ	##	##T					
lachining t	imes							- D -
T - ID	Nach.time No	ncuttg.	Sun [h:n	in:s]				0.0
1-883AP1	8:84	8:81	8:85					
1-883AP1	8:29	8:85	8:34					
1-883AP1	8:28	8:86	8:34					-Th
2-151-688.2	8:18	8:85	8:15					-07
2-151-688.2	8:86	8:84	8:18					
3-261-600.1	8:88	8:81	8:81					
	otal machin:	ing time:						_
								- T
N 84 G8 Z N 84 X	-64 278.800	IZ -6	4.000 C	0.080 Y	0.000 T	3 261-6	99.1	- -



6.7 Saving the contour

Saving the generated contour in simulation submode

You can save a contour generated in the **Simulation** and read it into **smart.Turn**.

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.

With contour generation the control saves all contours in the selected contour group.

The **Simulation** submode considers the following shift of the workpiece zero point and/or a mirror image.

- 0: Only move
- 1: Rotate in the main spindle (mirror)
- 2: Rechuck in the counterspindle (shift and mirror)

Saving the contour:



Select "miscellaneous functions" soft key



6.<mark>7 Sa</mark>ving the contour

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
 - Selection of contour group Q
 - Type of shift H
 - Shift length workpiece K: Shifting the workpiece zero point

Loading the saved contour



Only add the saved contour to newly created or copied programs, because all previously generated contours are overwritten. This action cannot be undone.

In smart.Turn mode, 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.

When inserting in the mode of operation, all contours in all planes are initially automatically deleted. Then all saved contours in all planes are loaded from the **Simulation** submode.

The "Save contour" function in **Simulation** submode converts all contours in all planes of the selected contour group and the NC editor replaces all contours; if the program contains contour groups, the groups where the cursor is positioned are replaced following a confirmation request.





6.8 Dimensioning

Measuring contours in simulation submode

You can measure a contour generated in Simulation submode or have the dimensions used during programming displayed.

Measure contour:



- Select "miscellaneous functions" soft key
- Select the "Misc." menu
- **⊣**┩┥

Select the "Dimensioning" menu

You have the following possibilities:

- Element dimensioning
- Point dimensioning
- Setting a datum

Press the menu item Element dimensioning

The Element dimensioning menu item is automatically active if you selected the measuring function. All data of the marked contour element are shown in the display below the graphic.

- The arrow shows the direction of the contour description.
- To next contour element: "Element forward / back" soft key
- Change contour: "Previous contour" or "Next contour" soft key



With figures, the single elements are measured.

Menu item Point dimensioning

The control displays the dimensions of the contour point relative to the zero point.

- To next contour point: "Point forward / back" soft key
- Change contour: "Previous contour" or "Next contour" soft key



Soft keys for "Dimensioning" function



Menu point Set datum

This function is only available with point dimensioning. It enables you to shift the zero point and measure the distance to the zero point.



Select a new zero point with "Point forward / back"



6.8 Dimensioning

Select the menu item "Set datum": The point symbol changes color



Select the point with "Point forward / back": The control displays the distance relative to the selected zero point

Cancel the reference point



Select "Reference point off": The set zero point is rescinded.

▶ The displayed values once again reference the original zero point.

6.9 3-D simulation

3-D simulation in simulation submode

In the Simulation submode, you can test a program with the 3-D simulation.

Activate 3-D simulation:



Select the "3-D simulation" menu item

Deactivate 3-D simulation:



Select the "2-D simulation" menu item

The following functions are identical to 2-D simulation:

- Using the simulation (see "Using the simulation submode" on page 487)
- 3-D view (see "3-D view" on page 494)
- Finding a start block (see "Startup block with smart.Turn programs" on page 497)
- Time calculation (see "Time calculation" on page 499)
- Saving contours (see "Saving the contour" on page 500)

3-D zoom

The zoom function enables workpiece blanks and finished parts to be shown in various perspectives.



Press the "3-D zoom" soft key



You can rotate the 3-D simulation with menu items and the mouse (see "Rotating the 3-D view with the menu functions" on page 495), (see "Rotating and moving the 3-D view with the mouse" on page 495).

If the cutting edge collides with the workpiece during rapid traverse, the sectional planes are displayed red.



If the cutting edge collides with the workpiece during rapid traverse, the sectional plane is displayed red.



Soft keys for miscellaneous functions



Positions a thread pattern on the surface.



Displays the workpiece edges.

Soft keys for the 3-D zoom



Rotates holes or milling contours to the left.



Rotates holes or milling contours to the right.



Displays a section view.



Selects the side view. Rotates the side view by 90°.



Selects the perspective view.



6.9 3-D simulation

i




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 MANUALplus 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 MANUALplus can save tool data for up to 250 tools (optionally 999), whereby each tool is identified with a number (ID code). The tool list indicates the maximum number of tool data records and the number of data records found. 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 112).

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 technology 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	3	Tool types	
	Standard turning tools (Page 524) Roughing tools Finishing tools		NC center drills (Page 528)
	Button tools (Page 524)	~	Centering tools (Page 529)
F	Recessing tools (Page 525) Recessing tools		Counterbores (Page 530)
	Parting toolsRecess-turning tools		Countersinks (Page 531)
	Thread-cutting tools (Page 526)		Standard milling tools (Page 534)
	Twist drills (Page 527)		Thread mills (Page 535)

→ Machin	e		smart.Tu:	:n	Pi	Tool edi	tor			
x	23.420	B	0.	000 C	1		ID		0	
z	31.710	W	0.	000 C	4		T	0 ×	0.0000) -
Y	0.000	I) 4 (0.0 0.0	n/nin U/nin	ы <u>к</u> 1 К	U N	™	0.0	000	
😫 1 🙋	0.0 mm/nin 0.0 mm/nin	I) 1	1500.0	m∕min 60.0 °	() 2 0	1500.0 m/r 0.0 U/r	tin 1 F 10 R 10	0% S₁ 0% S₁	100%	
Tool list							010 6	040 1	000	
Filter A	II types	Denignal	ion	DC /DV	EU (PU /A7	CLI / CD / LLC	318 of	318 Ma	ax 1999	
001	J. 1	Schruppe	n Aussen	0.80	93.0	80.0	Hartmetall	. 10 1	3	
001-capto	₽, 1	Schruppe	n Aussen	0.80	93.0	80.0	Hartmetall		3	
001AP0	<i>B</i> , 1	Schruppe	n Aussen	0.80	93.8	80.0	Hartmetall		3	
001AP1	B, 1	Schru	lect too	1 type		0.0	Hartmetall		3	
002	<i>₽</i> , 1	Schru		a cype		5.0	Hartmetall		3	198
002-capto	<i>₽</i> , 1	Schruppe	n Aussen	0.80	93.0	80.0	Hartmetall		3	
002AP1	J, 1	Schruppe	n Aussen	0.80	95.0	55.0	Hartmetall		3	
003	B, 1	Schlicht	en Auss	0.80	95.0	35.0	Hartmetall		3	-
003-capto	J, 1	Schruppe	n Aussen	0.80	93.0	80.0	Hartmetall	Θ	3	লি
003-capto_3	L 🖉, 1	Schruppe	n Aussen	0.80	93.0	80.0	Hartmetall	1	3	\square
003-capto 2	2 1	Schrupp	n Aussen	0.80	93.8	80.0	Hartmetall	2	3 ,*	13:14
	F				s ¢			Spe	ecial ool	Back

Tool type	S	Tool types	
E	Indexable-insert drills (Page 527)	Angle cutters (Page 536)	
	■ Taps (Page 533)	Milling pins (Page 537)	
	■ Reamer (Page 532)	Knurling tool (Page 537)	I
	■ Touch probes (Page 539)	Stopper tool (Page 540)	
L L	■ Grippers (Page 541)		

Multipoint tools



This function is also available on machines with a tool magazine. The control uses the magazine list instead of the turret list.

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 point. Then all data records of the multipoint tool are linked together (see "Editing multipoint tools" on page 513).

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 starts from 0.

In the turret list, multipoint tools are displayed with all cutting edges and reference points.

The figure at right shows a tool with two reference points.

Tool life management

The MANUALplus 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 sets **the diagnostic bit 1**. This causes an error message to be issued the next time the tool is called. If no replacement tool is available, the program run is stopped.

To finish the workpiece currently being produced, press NC Start.



7.2 Tool editor mode of operation

Navigating in the tool list

In the tool list, the MANUALplus 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.

You can navigate within the tool list with the **arrow keys** and **PgUp/PgDn** to check the entries. Infrequently used tool parameters are in the list further to the right, and can be displayed by navigating in the columns.

For orientation purposes, the following columns are always displayed:

- ID number
- Tool type
- TO
- Designation







Navigation in the tool list is the same in all modes of operation.

Sorting and filtering the tool list

Displaying the entries of only one tool type



Press the soft key and select the tool type in the following soft-key rows.

The MANUALplus creates a list containing only tools of the selected type.

➡ Machine		🚯 sr	art.Turn			¶ ⊺o	ol edi	tor		B		
х	23.420	B	0.0	00				ID			0	
Z	31.710	W	0.0	00	4			Т	0	z	0.0000	
Y	0.000	i) 4 0	0.0 m 0.0 U	/nin I/nin	K I		U N	- 15	•	0.00	0	
😫 1 🔯	0.0 mm/min 0.0 mm/min	1)10	1500.0 m 360	/nin 1.0 °	1) 2 🤇	15	00.0 n/n 0.0 U/n	in 1 R	100%	S ₁ 1	00%	
Tool list						-					-	
Filter Al	l types							318	of 31	8 Max	999	
ID number	• TO	Designatio	on	RS/DV	EW/BW/	AZ SW	/SB/HG	Cutting	mat.	MU MD	LA 📤	
001	₽ ₊ 1	Schruppen	Aussen	0.80	9 93	.0	80.0	Hartmet	all	3		
001-capto	Define fil	ter detail	s						-	3		
001AP0		Column		Cond:	ition		Val	lue		3		
001AP1	Teel tu			-		00	A11 +	IDAE	-	3		
002	1001 ty	pe		quais		100	ALL U	hes		3		
002-capto			•		•					3		
002AP1						_				3		
003			-		•	1				3		
003-capto	ø., 1	schruppen	AUSSET	0.00	3 33	. 0	80.0	патспес	a11	03		
003-capto_1	₽, 1	Schruppen	Aussen	0.80	9 93	.0	80.0	Hartmet	all	1 3		
003-capto 2	1 لا	Schruppen	Aussen	0.80	3 93	.0	80.0	Hartmet	all	2 3	×	13:15
Delete filter										ОК	:	Cancel

Filtering the tool list



Sorting the tool list



Press the View soft key.

- The tool list switches between sorting by ID number and by tool type (and orientation)".
- The tool list switches between ascending and descending sorting.

Searching for tools by ID number

- Enter the first few letters or digits of the ID number.
- ▶ The MANUALplus jumps to the desired ID number in the open list.



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 MANUALplus 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 512).



The MANUALplus 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.



Press the soft key. The MANUALplus opens the input window with the tool data.

Enter a new **ID number**. Check and edit the other tool data.



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.

Delete entry

Place the cursor on the entry to be deleted.



Press the soft key and answer Yes to the confirmation prompt.

Soft keys f	or tool organization				
New tool	Opens the following type selection for adding a new tool.				
	Back				
Special too	ls:				
ŀ	Back				
	Type selection for special drilling tools:				
	Back				
	Type selection for special milling tools:				
	Back				
	Type selection for handling systems and touch probes:				
	Back				
Editing	Opens the dialog box for the selected tool.				
Сору	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 submode (see Page 542).				
Holder editor	Opens the table of the tool holder.				

Tool control graphics

When the tool dialog box is open, the MANUALplus provides a control graphic with which you can check the entered tools. Press the **Graphic** soft key.

The MANUALplus generates the displayed tool from the entered parameters. The tool control graphic enables you to check the entered data. Changes become effective as soon you exit the input field.





Tool texts

Tool texts are assigned to the tools and displayed in the tool list. The MANUALplus 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, MANUALplus lets you enter tool texts. Press the ${\bf Tool\ texts}$ 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.
Сору	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.

In the tool list, place the cursor on the data record for the first cutting edge.

Editing	Press the soft key.
Multipoint tool	Press the soft key. The Tool editor mode considers this cutting edge to be the main cutting edge (MU=0).

Place the cursor on the data record for the next cutting edge.



X	23.420 B	0,0	000 C	1		ID	0	
7 .	24 740 11		000			т	O X 0.0000) 🔽 🖓
2	31.710 0	0.0	UUU U	4			U Z 0.0000	0.0
Y	0.000 1	4 0 0.0	n/nin U/nin	K I	U N	```	0.000	
🚉 1 🔯	0.0 nm/nin 0.0 nm/nin 1	1 <u>1580.0</u> 38	n/min 50.0 ° 1) 2 😐	1500.0 n/i 0.0 U/i	nin 1 F 100 nin 1 R 100)% S ₁ 100%	
Multiwerkzeu	ıg							
MU ID numb	er T	O Designation	RS/	DV EW/B	W/AZ SW/S	SB/HG MTS		
oor-cap	.0 *,	I SCHLuppen Au	ssen	0.00	55.0	00.0 1		
Tool list								
Filter All	l types					318 of	318 Max 999	
ID number	 TO Des 	signation	RS/DV	EW/BW/AZ	SW/SB/HG	Cutting mat	. MU MD LA 📤	
001	₽, 1 Sch	ruppen Aussen	0.80	93.0	80.0	Hartmetall	3	
001-capto	💐 1 Sch	nruppen Aussen	0.80	93.0	80.0	Hartmetall	3	
001AP0	₩, 1 Sch	nruppen Aussen	0.80	93.0	80.0	Hartmetall	3	
001AP1	🖉 1 Sch	nruppen Aussen	0.80	93.0	80.0	Hartmetall	3	I 💷
062	🖉 🕹 1 Sch	nruppen Aussen	0.80	95.0	55.0	Hartmetall	3 -	
L							<u>.</u>	13:15
Cuttng edge forward	Cuttng edge backvard	Remove cuttng edge	Tool type	fi	More ilters	Viev	Insert secon. edge	Back

Tool editor

B

smart.Turn

➡ Machine

→ Machine		📀 smart.Tur	n	🖹 Tool ed:	itor		
X 23	6.420 B	0.0	000 C 1		ID	0	
Z 31	.710 W	0.0	000 C.4		T	0 X 0.0000 2 0.0000) - 1 2
Y O	.000 1	4 0 0.0	n/nin 🛋	I U K W	HÉ ⊕	0.000	
🚉 1 🙋 🦂	0 nn/nin 1 0 nn/nin 1	1 0 1588.8	^{n/nin} 1) 2	0 1500.0 n/ 0.0 U/	nin 1 F 100	S 100%	
Multiwerkzeug							
MU ID number	T AL	D Designation	RS/DV	EW/BW/AZ SW/	SB/HG MTS		
1 803-capto	1 4	Schruppen Au	ssen 0.00	93.6	80.0 1		
2 803-capto	2	Schruppen Au	ssen 0.80	93.0	80.0 1		
Tool list						_	
Filter All t	ypes				318 of 3	18 Max 999	
ID number	 TO Des 	ignation	RS/DV EW/	BW/AZ SW/SB/H0	GCutting mat.	MU MD LA -	
001	₩, 1 Scł	ruppen Aussen	0.80	93.0 80.	9 Hartmetall	3	
001-capto	💐 1 Sch	ruppen Aussen	0.80	93.0 80.	0 Hartmetall	3	
001AP0	💐 1 Sch	ruppen Aussen	0.80	93.0 80.	9 Hartmetall	3	
001AP1	🖉 1 Sch	ruppen Aussen	0.80	93.0 80.	0 Hartmetall	3	
002	💐 1 Sch	ruppen Aussen	0.80	95.0 55.	9 Hartmetall	3 🖃	
						2	13:16
Cuttng edge C forward	uttng edge backvard	Remove cuttng edge	Tool type	More filters	Viev	Insert secon. edge	Back

Repeat these steps for any further cutting edges of the multipoint tool.

Press the soft key.

Removing a cutting edge from the multipoint tool

Place the curso	or on a cutting edge of the multipoint tool. 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	Place the cursor on the cutting edge no. 0 of the multipoint tool.
Cuttng edge backward	
Remove cuttng edge	The multipoint tool chain is removed.

Tool and technology database

i

Editing tool-life data

The MANUALplus 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** submode 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



Set the soft key to "No. of pieces." The **Tool editor** submode opens the **Tool life MT** 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 associated data record in the **Tool editor**.



Press the soft key. The tool life/limit quantity is set to zero and the diagnostic bits are reset.

Tool life monitoring is activated/deactivated in the Tool life monitoring (see "List of machine parameters", page 551) user parameter.

- 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.



Diagnostic bits

The diagnostic bits store information about the status of a tool. The bits are set either by programming in the NC program or automatically by the tool and load monitoring functions.

The following diagnostic bits are available:

- Bit Meaning
- 1 Tool life expired or workpiece quantity reached
- 2 Breakage according to load monitoring (limit value 2 exceeded)
- 3 Wear according to load monitoring (limit value 1 exceeded)
- 4 Wear according to load monitoring (total load limit)
- 5 Wear determined by tool measurement
- 6 Wear determined by in-process measurement of workpiece
- 7 Wear determined by post-process measurement of workpiece
- 8 Cutting edge new = 1 / used = 0

9-15 Vacant

If tool life monitoring or quantity monitoring is active and a diagnostic bit has been set, the tool concerned will not be re-inserted again during the **Program Run** submode. If a replacement tool has been defined, the control inserts it instead. If no replacement tool has been defined or if the end of the tool sequence of exchange is reached, the NC program is stopped before the next tool call.

You can reset the diagnostic bits in the tool editor as follows:

Editing

Press the EDITING soft key.

▶ Press the NEW CUTTING EDGE soft key.



With the **New cutting edge** soft key, you reset the diagnostic bits and set bit 8 "Cutting edge new." This bit will also be reset as soon as the control inserts the tool.

→ Machine		🚯 smart.Tu	ırn	Ρ, Te	ol ed:	itor		
X [®] 2	23.405 🗠	ĸ	ω		0.00	O ID		
Z :	3 1.703 4	Z	🔀 C 1			Т	0 X 0.0000 Z 0.0000	Ŧ
Y	0.000 🗠	Y	C 4			<u>⊫</u> ₽	0.000	
🗄 1 🙋	0.0 mm/nin 0.0 mm/nin 1) 1 • 300.	0 m/nin 245.0 S 1 0	3000	liso Vmin	et 1 F 100 R 100	${}^{8}_{2}$ S $_{1}$ 100%	
Diagnostic I	oits (DG=0x	0100)		S	s <mark>Har</mark>	tmetall	•	
∏ Standzeit	abgelaufen	oder Stückzah	l erreicht	C	К 1	FK	1	DEE ON
☐ Bruch erm ☐ Verschlei	ittelt durch ß ermittelt	Belastungsüb d. Belastungs	erw. (Grenze überw. (Grenz	2) 🛛	K 1 T 0.0	M07		
☐ Verschlei	ß gemäß Bela	stungsüberv.	(Gesamtbelast	ung) _j	Z 0	RZ	0	OFF
Verschlei	B ermittelt	durch werkzeu durch Inprozm	g vermessung essen Werkstü	ck T	0:	Ready for use	2	M08
└─ Verschlei	ß ermittelt	durch Postpro	zmess. Werkst	ück	UT 0:			
⊠ Schneide								
							2/3	11:34
Holder editor	Tool text	Dvell	New tooth	Diag Bi	iosis ts	Graphic	Save	Back

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 editor

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.

Edit the holder table in Tool editor mode:



editor

Press the "Other tables" soft key

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



→ M:	achine			🚯 smar	t.Turn	F) Tool	editor	ÌE	1	
X⊛	3	00.000	∆X		_	ω	0	.000 ID		001	_ 5
Z	4	50.000	۵Z		×	C 1		Т	1	X 0.0000 Z 0.0000	0 0
Y		0.000	∆Y			C 4		ľ	ۍ	. 000 卢	
^{S1} ₽2	()	0.0 mm/min 0.0 mm/min	1	1 0	300.0 n/nin 116.8 °	S 1 0 3	000 U/m:	e 1	F 100% S	1 100%	
Table	of th	e tool hol	der			_					
NR	4	HID		MTS	XLH	YLH		ZLH	HC	<u>^</u>	
	2 HB1			0	9.1	5	0.0	0.0		61	
	3 016	l-canto-50		1	19.1	3	0.0	30.0		R1	
	4 023	-canto-GE	(1	2.3	1	2.34	2.34		B1	
	5 Sti	rnDreh-11	ì	0	0.1	9	0.0	0.0		B1	
	6 Mar	tDreh-111		0	Θ.	3	0.0	0.0		D1	
	7 Mar	tDreh-113		0	0.1	9	0.0	0.0		C1	
	8 Mar	tDreh-117		Θ	Θ.	9	0.0	0.0		A1 _	
	9 Mar	ntStech-AR		Θ	0.1	3	0.0	0.0		A1	
	10 Mar	ntGev-AL		Θ	0.1	3	0.0	0.0		C2	
	11 Mar	tStirn-338	3	0	0.	3	0.0	0.0		T1 🔪	_ /
ID nur	nber				Text widt	h 17		TNC:\tab	Le\to_hold	l.hld	11:33
Ne li	ev ne	Editing	,	Сору						Delete	Back

- 7.2 Tool e<mark>dito</mark>r mode of operation
- 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

soft key

Set up the manual change system holder in the turret assignment:

 Turret
 Select turret assignment: Press the "Turret list" soft key

 Special
 Select an unassigned turret pocket and press the "Special functions" soft key

 Set-up the holder
 Open the holder table: Press the "Set up holder" soft key

 Transfer
 Select the holder and press the "Transfer of ID no."



of ID no.

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

On the third form page, select MTS 1: MANUAL CHANGE

Open the tool data form: Press the "Edit" soft key

T00L

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 all cutting edges must be assigned the same **MTS** input value.

D-nur	ber clipboard		Г				poo	kets 🗌	6 of	24	
#	ID number		TO	Designation	RS/DV	Change	tool	HID		<u>^</u>	0
1	801	₽,	1	roughing	0.40						
2											
3	020	₽,	1	finishing	0.40						-
4											
5	028	2	1	thread cutting							
6	001-capto	<i>P</i> ,	1	roughing	0.80			016	-capto-50		
	822	10	T	recessing	0.10						
8	045	-	0	aillina	10.00						
10	045	-	0	miling	10.00						
11										*	
able	of the tool holde	er									
NR	 HID 		Μ	TS XLH	YLF	1	ZLH		HC	_	
	1HB1			0 0.6	1	0.0		0.0	B1		
	2 HC1	_		0 0.6	1	0.0		0.0	C1	_	
	3 C10-capto-50			1 19.6	1	0.0		38.0	B1		
	4 C23-capto-GFX			1 2.34	l.	2.34		2.34	B1		一五
	5 StirnDreh-111			0 0.6	1	0.0		0.0	B1		
1										2	
D nur	nber			Text widt	h 17		TNC:	\table\t	o hold.hl	d	44.00

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: -10 mm < DX < 10 mm)
- DZ Wear compensation in Z (Range: -10 mm < DZ < 10 mm)
- DS Special compensation (range: -10 mm < DS < 10 mm)
- MU Multipoint tool
- MD Direction of rotation (default: Not defined)
 - 3: M3
 - 4: M4
- Rem. Remaining time / remaining quantity (with tool life monitoring)
- Status With tool life monitoring
- Diagn. Evaluation of diagnostic bits (with tool life monitoring)
- 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
- PTY Pocket type (machine-dependent function)
- NMX Max. shaft speed







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 MANUALplus needs a unique name for each tool. This "ID number" can have up to 16 alphanumeric characters.
- **Tool orientation (TO):** The MANUALplus 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 managed in a separate list, the reference to the text is entered in QT (see "Tool texts" on page 512).
- Cutting material (SS): This parameter is required if you want to use the cutting data from the technology database (see "Technology database" on page 542).
- 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 MANUALplus 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 MANUALplus 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



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** submode 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 521.

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 521.



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



Recessing tools are used for recessing, parting, recess turning and finishing (only in **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 521.

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: $0^{\circ} \le W \le 180^{\circ}$)
- SW Point angle (range: $0^{\circ} \le SW \le 180^{\circ}$)
- DN Tool width
- SD Shank diameter
- ET Maximum plunging depth
- NL Usable length

For further tool parameters, see Page 521.







1

Twist drills and indexable-insert drills





For drilling operations with constant cutting speed, the **drilling diameter (DV)** is used to calculate the spindle speed.



6

8

8

i

NC center drill

New tool	Select "New tool."	
Special tool	Select special tools.	2
	Select special drilling tools.	→ 4 T0= 8<
	Select NC center drill.	C

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 521.



For drilling operations with constant cutting speed, the hole diameter (DV) is used to calculate the spindle speed.



 \bigcirc



7.3 Tool data

1

Centering tool





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 521.



For drilling operations with constant cutting speed, the **hole diameter (DV)** is used to calculate the spindle speed.







Counterbore

7.3 Tool data

New tool	Select "New tool."
Special tool	Select special tools.
	Select special drilling tools.
6	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 521.



For drilling operations with constant cutting speed, the **hole diameter (DV)** is used to calculate the spindle speed.







i

Countersink





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 521.



For drilling operations with constant cutting speed, the hole diameter (DV) is used to calculate the spindle speed.







Reamer

7.3 Tool data

New tool	Select "New tool."	
Special tool	Select special tools.	2
	Select special drilling tools.	4 T0= 8 6
	Select reamers.	

The help graphics illustrate the dimensions of the tools.

Special parameters for reamers DV Hole diameter

- DH Stud diameter
- BW Drill angle

For further tool parameters, see Page 521.



For drilling operations with constant cutting speed, the **hole diameter (DV)** is used to calculate the spindle speed.



6/

Tool and technology database

i

Тар



The help graphics illustrate the dimensions of the tools.

Special parameters for taps DV Thread diameter

- ΗG Thread pitch
- AL Length of first cut

For further tool parameters, see Page 521.



The thread pitch (HG) is evaluated if the corresponding parameter is not defined in the tapping cycle.



7.3 Tool data





1

Standard milling tools

New tool



Select milling tools.

Select "New tool."

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 521.



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





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 521.

- 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

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 521.

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





The help graphics illustrate the dimensions of the tools.

Special parameters for milling pins

- Cutter diameter DV ΑZ
- Number of teeth
- SL Cutting length FW

- Cutter angle
- DD Cutter diameter compensation

For further tool parameters, see Page 521.

- 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.





Knurling tool

7.3 Tool data



The help graphics illustrate the dimensions of the tools.

Special parameters for knurling tools SL Cutting length

- ΕW Tool angle
- SB Cutting width
- Tool width DN
- SD Shank diameter

For further tool parameters, see Page 521.







1

Touch probes



The help graphics illustrate the dimensions of the tools.

Special parameters for touch probes SL Cutting length

- Cutting length
- ΤP Selection of touch probes

For further tool parameters, see Page 521.



The MANUALplus must be specially prepared by the machine tool builder for the use of a 3-D touch probe.





1

Stopper tool

7.3 Tool data

New tool	Select "New tool."
Special tool	Select special tools.
T.	Select handling systems and touch probes.
0	Select stopper tool.

The help graphics illustrate the dimensions of the tools.

Special parameters for stopper tools DD Special compensation For further tool parameters, see Page 521.








Gripper



The help graphics illustrate the dimensions of the tools.

Special parameters for grippers DD Special compensation

For further tool parameters, see Page 521.







1

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 MANUALplus ascertains these criteria from the following information:

- Machining mode: In cycle programming (Teach-in submode) a machining mode is assigned to each cycle and in the smart.Turn mode a machining mode is assigned 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 MANUALplus 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 machinin	g
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

7.4 Technology database

Technology editor submode

The Technology editor submode is called from the Tool editor and smart.Turn 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 MANUALplus 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 mode:



Press the "Other tables" soft key



Call the Technology editor submode: 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.

01	-	300			
HWork naterials	s ∰Cuttg, n	aterl. 🎹 Cutting da	ta 🎹 Extras		_
ork materials	CODTND				2 1
nai Si	tahl 1				
Rostfreier St	tahl 2				
Alumin	nium 3				
					নিয়া
torial		Toyt width 29	TNC:\table\t	ochdata hto A	\square
iterrar		TEXT WIGHT 20	inc. (table (t	echdata.nte	
					11:33

Deleting a workpiece material:

Delete work piece mat. Press the soft key. After the confirmation request, the MANUALplus 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 cutting mat. Press the soft key. Enter the cutting material designation (maximum 16 characters). The sorting number is assigned sequentially.

Deleting a cutting material:

Delete cutting mat. Press the soft key. After the confirmation request, the MANUALplus 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.

➡ Machine		i 🕹 sma	rt.Turn	🖹 🎦 Technol	ogy editor	8	
∰Work materia	ls 🏭 Cut	tg. nate	rl. 🇰 Cutting da	ita # Extra	s		
Cuttg. materl.							
CUTMAT	SOR	TNR+					0.0
Hartm	etall	1					_
	HSS	2					
Ke	ramik	3					
							-
							(T)
Cutting materia	1		Text width 20	1	NC:\table\teck	ndata.hte 🔒	
							11:33
Append cut- De ting mat. t	lete cut- ing mat.			Edit field			Back



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** submode 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** submode 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** submode displays the cutting data.



The **value 0** in a data record means that no value is transferred to the unit or cycle dialog.

Cutting data for	cutting material:	Hartmeta	all Mai	erial: S	tahl			
TASK	CUTMAT	CSP	FDR	AFDR	DEP	COOL		0
Predrilling	Hartmetall	90	0.25	Θ	θ	Θ		_
Roughing	Hartmetall	200	0.35	0.25	5	Θ		
Finishing	Hartmetall	228	0.15	0.1	θ	Θ		=
Thread cutting	Hartmetall	128	0	Θ	θ	Θ		
Contour recessg.	Hartmetall	169	0.25	0.2	θ	Θ		
Parting	Hartmetall	140	0.25	0.18	8	Θ		
Centering	Hartmetall	θ	Θ	Θ	θ	Θ		
Drilling	Hartmetall	80	0.28	Θ	8	Θ		
Countersinking	Hartmetall	8	Θ	Θ	0	Θ		
Reaming	Hartmetall	0	0	Θ	0	Θ		
Tapping	Hartmetall	68	0	Θ	θ	Θ		
Milling	Hartmetall	64	0.05	0.02	5	Θ		
Finish milling	Hartmetall	74	0.03	0.01	5	Θ		
Deburring	Hartmetall	θ	0	Θ	θ	Θ		
Engraving	Hartmetall	θ	0	Θ	θ	Θ		
Recess turning	Hartmetall	169	0.5	0.3	5	0		Ē
Cutting speed	n/nin	Min. 0.0	000, max	10909.6	TNC: Y	table\	techdata.hte	
Counied: 9 data	records(of maximu	62)						11.3

		••
New catting data New data for Haterial Stahl - Cutting mat. Hartsetall - Data available New Data available Yes		=
		T
	ОК	11:33 Cancel



Editing cutting data:

Edit field

New

data record

Call the table with cutting data.

With the **arrow keys**, select the cutting data field you want to edit.

Press the soft key

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 Technology editor opens the "New cutting data" dialog box.

- Set the desired workpiece material/cutting material combination.
- Decide whether an existing workpiece material/cutting material combination should be used as a template. Otherwise, all entries will be preset with "0."
- Press **0K** 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.

Del. data record

YES

- Press the soft key. The Technology editor asks if you really want to delete the data record.
- Press the soft key. The Technology editor deletes the data record of the given workpiece material/ cutting material combination.

Cutting data for	cutting material	: Hartmeta	all Mat	terial: St	ahl			
TASK	CUTMAT	CSP	FDR	AFDR	DEP (200L		0.0
Predrilling	Hartmetall	98	0.25	Θ	θ	Θ		_
Roughing	Hartmetall	208	0.35	0.25	5	Θ		
Finishing	Hartmetall	228	0.15	0.1	θ	Θ		
Thread cutting	Hartmetall	128	Θ	0	θ	Θ		
Contour recessg.	Hartmetall	169	0.25	0.2	θ	Θ		
Parting	Hartmetall	140	0.25	0.18	θ	Θ		
Centering	Hartmetall	θ	Θ	Θ	θ	Θ		
Drilling	Hartmetall	80	0.28	0	θ	Θ		-
Countersinking	Hartmetall	8	Θ	Θ	θ	Θ		
Reaming	Hartmetall	θ	Θ	0	θ	Θ		
Tapping	Hartmetall	68	Θ	Θ	θ	Θ		
Milling	Hartmetall	64	0.05	0.02	5	Θ		
Finish milling	Hartmetall	74	0.03	0.01	5	Θ		
Deburring	Hartmetall	8	Θ	0	θ	Θ		
Engraving	Hartmetall	8	Θ	Θ	θ	Θ		
Recess turning	Hartmetall	169	0.5	0.3	5	Θ		\square
Cutting speed	n/nin	Min. 0.0	000, max	10909.0.	. TNC:\1	table\tec	hdata.hte	
				_				





Organization mode of operation

8.1 Organization mode of operation

The **Organization** submode 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 MANUALplus to your specific requirements. The **Machine parameter** menu provides functions to display and edit parameters.

Transfer

The **Transfer** submode is used either for exchange of data with other systems or for data security. 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
	Machine parameters, changing
	Submode Transfer:
	Transmit/Receive programsCreate service files
123	All machine parameters, changing
	Submode Transfer:
	Parameter backupBackup/Restore tools
net123	Configuring the network (control name / DHCP)
	Submode 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 MANUALplus 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::

œ₽	Branch exists but is closed
⊡ <mark>⊡</mark> ⊡	Branch is open
H	Empty object, cannot be opened
	Initialized machine parameter
(22222)	Uninitialized (optional) machine parameter
	Can be read but not edited
×	Can neither be read nor edited

Machine parameters (user parameters)

Parameters that are preset for the usual "daily operations" are organized as **Machine parameters**.

To enable you to set machine-specific functions, your machine tool builder can make further parameters available as user parameters.

Refer to your machine manual.

Edit machine parameters







8.2 Parameters

Displaying help texts





Press the SEARCH soft key again.

Exit the parameter editor

END

Press the END soft key

 \bigcirc - Oversize ir - Rotational direction for new units : M3 11:35 COPY FIELD PASTE FIELD FIND END

🗄 Machine par.

Program run is locked LEVEL1 PLC:\...\proc_data.cfg

 \odot

11:35

💾 Machine par.

magazine and too

eously 1 on

🏷 Tool editor

General settings

Access

HELP

PAGE

🏳 Tool editor

CURRENT WORD

END

smart.Turn

e or part

Replace with:

Search forward

🗵 Names

☐ Numbers ☐ MP number ☐ Values

Key name

Comment

FIND

- Safety clea - Safety clea

- Oversize in

RENT WORD

smart.Turn

2

CANCEL



List of machine 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
- KOREAN
- NORWEGIAN
- ROMANIAN
- SLOVAK
- TURKISH
- ... / 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 display settings (604800) /	
/ Axis display (604803)	Type of axis display: Default Actual value Nominal value Following error (servo lag) Distance-to-go
/ File preview during program selection (604804)	
TRUE	File preview is displayed during program selection.
FALSE	File preview is not displayed during program selection.
/ Do not display limit switch warnings (604805)	
TRUE	No limit-switch warning is displayed if an axis is positioned on the software limit switch.
FALSE	The limit switch warning is displayed.
/ Settings for automatic operation (601800) /	
/ Service life management (601801)	
ON	Tool life monitoring is active
OFF	Tool life monitoring is inactive
/ Program run with the most recently selected cycle (601809)	
ON	When Program Run is selected, the most recently selected cycle remains active
OFF	When Program Run is selected, the first cycle is active
/ 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

8.2 Parameters

Organization mode of operation

S
<u> </u>
Ð
ũ,
Ū
σ
<u> </u>
σ
Δ
\mathbf{N}
00

Parameter: System /	Meaning		
/ 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 Machine operating mode (604900) /			
/ Save cycle without simulation (604903)			
TRUE	Cycle can be saved without previous simulation or execution.		
FALSE	Cycle can only be saved after previous simulation or execution.		
/ Run a tool change with NC start (604904)			
TRUE	The tool change with the TSF dialog is conducted with Cycle Start.		
FALSE	The tool change is not conducted with Cycle Start.		
/ Separate dialogs for tool change, speed and feed rate (604906)			
TRUE	Data input for tool change, spindle speed and feed rate in separate dialogs		
FALSE	TSF dialog box with input of all cutting data		
/ Settings for load monitoring (124700) /			
/ Activate load monitoring (124701)			
TRUE	Load monitoring is active		
FALSE	Load monitoring is inactive		
/ Factor for utilization limit 1 [%] (124702)	This value multiplied by the reference value determined by reference machining results in the utilization limit value 1.		
/ Factor for utilization limit 2 [%] (124703)	This value multiplied by the reference value determined by reference machining results in the utilization limit value 2.		
/ Factor for total utilization limit [%] (124704)	This value multiplied by the reference value determined by reference machining results in the total utilization limit value.		

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.
/ Software limit switches active (114803)	
ON	Software limit switches also active during simulation
OFF	Software limit switches not active during simulation
/ 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 20 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."
/ T20	
/ 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."

S	
jte	
M	
rai	
Pa	
Ņ	
00	

Parameter: Simulation / ... Meaning ... / 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:

Para	ameters: processing /	Meaning
/	General settings (602000) /	
	/ Type of tool access (602001)	Default value for tool access:
		 0: First from NC program, then from tool table 1: Only from the NC program 2: First from the NC program, then from the magazine
		 3: First from the NC program, then from the magazine, then from the tool table
	/ 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)
		2: Circuit 2 on
	/ G60 for new units (602011)	Default value for "protection zone":
		0: Active1: Inactive
	/ Safety clearance G47 [mm] (602012)	Default value for "safety clearance G47":
	/ Safety clearance, G147 plane [mm] (602013)	Default value for "safety clearance SCK"
	/ Safety clearance, G147 infeed direction [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"
	/ Direction of rotation for new units (602017)	Default value for "direction of rotation MD"

Parameters: processing / ... Meaning

... / Zero point shift (602022)

OFF

ON

... / Front chuck edge on main spindle (602018)

... / Front chuck edge on opposing spindle (602019)

... / Jaw width on main spindle (602020)

... / Jaw width on opposing spindle (602021)

... / Conversion of ICP contours (602023)

... / Global parameters for finished parts (601900) / ...

... / Max. inward copying angle (EKW) [°] (601903)

... / Centric predrilling (602100) / ...

... / 1st drilling diameter limit (UBD1) [mm] (602101)

... / 2nd drilling diameter limit (UBD2) [mm] (602102)

... / Point angle tolerance (SWT) [°] (602103)

... / Drilling oversize - Diameter [BAX] [mm] (602104)

... / Drilling oversize – Depth [BAZ] [mm] (602105)

... / Traverse for predrilling (ANB) (602106)

... / Traverse for tool change (ABW) (602106)

... / Safety clearance on blank part (SAB) [mm] (602108)

... / Internal safety clearance (SIB) [mm] (602109)

The AWG does not generate a zero point shift.

The AWG generates a zero point shift.

Z position of the front edge of the chuck for calculating the workpiece zero point.

Z position of the front edge of the chuck for calculating the workpiece zero point.

Jaw width in Z direction for calculating the workpiece zero point.

Jaw width in Z direction for calculating the workpiece zero point.

Conversion type:

■ 0: Read out calculated parameters

1: Read out programmed parameters

Tolerance angle for distinguishing turning from recessing.

Limit diameter for the first predrilling step

Limit diameter for the second predrilling step

Permissible point angle tolerance for inclined drilling limitation elements

Machining oversize on drilling diameter in X direction. Radius value

Machining oversize on drilling depth in Z direction.

Approach strategy:

1: XZ - simultaneous
2: XZ - sequential
3: ZX - sequential

Departure strategy:

1: XZ - simultaneous

- 2: XZ sequential
- 3: ZX sequential

Safety clearance on workpiece blank

Retraction distance for deep-hole drilling "B"



Parameters: processing /	Meaning
/ Drilling depth ratio (BTV) (602110)	Ratio for checking the predrilling steps
/ Drilling depth factor (BTF) (602111)	Factor for calculation of the first drilling depth for deephole drilling
/ Depth reduction (BTR) (602112)	Reduction for deep-hole drilling
/ Overhang length – Predrilling (ULB) [mm] (602113)	Default value for "drilling lengths A"
/ Roughing (602200) /	
/ Tool angle –ext./longitudinal (RALEW) [°] (602201)	Tool angle for roughing tool
/ Point angle –ext./longitud. (RALSW) [°] (602202)	Point angle for roughing tool
/ Tool angle –ext./transverse (RAPEW) [°] (602203)	Tool angle for roughing tool
/ Point angle –ext./transv. (RAPSW) [°] (602204)	Point angle for roughing tool
/ Tool angle –int./longitud. (RILEW) [°] (602205)	Tool angle for roughing tool
/ Point angle –int./longitud. (RILSW) [°] (602206)	Point angle for roughing tool
/ Tool angle –int./transverse (RIPEW) [°] (602207)	Tool angle for roughing tool
/ Point angle –int./transverse (RIPSW) [°] (602208)	Point angle for roughing tool
/ Machining –ext./longitudinal (RAL) (602209)	Strategy for roughing:
	 0: Full-surface roughing cycle, including plunge- cutting
	1: Standard roughing cycle without plunge-cutting
/ Machining –int./longitudinal (RIL) (602210)	Strategy for roughing:
	 0: Full-surface roughing cycle, including plunge- cutting
	 1: Standard roughing cycle without plunge-cutting
/ Machining –ext./transverse (RAP) (602211)	Strategy for roughing:
	0: Full-surface roughing cycle, including plunge- cutting
	 1: Standard roughing cycle without plunge-cutting
/ Machining –int./transverse (RIP) (602212)	Strategy for roughing:
	0: Full-surface roughing cycle, including plunge- cutting
	1: Standard roughing cycle without plunge-cutting
/ Adjacent angle tolerance (RNWT) [°] (602213)	Tolerance range for secondary cutting edge
/ Relief cutting angle (RFW) [°] (602214)	Minimum angle differential between the contour and secondary cutting edge

... / Type of oversize (RAA) (602215)

' (

Parameters: processing /	Meaning
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) (602216)	Equidistant oversize or longitudinal oversize
/ Face oversize (RPA) (602217)	Transverse oversize
/ Approach/external roughing (ANRA) (602218)	Approach strategy:
	 1: XZ - simultaneous 2: XZ - sequential 3: ZX - sequential
/ Approach/internal roughing (ANRI) (602219)	Approach strategy:
	 1: XZ - simultaneous 2: XZ - sequential 3: ZX - sequential
/ Departure/external roughing (ABRA) (602220)	Departure strategy:
	 1: XZ - simultaneous 2: XZ - sequential 3: ZX - sequential
/ Departure/internal roughing (ABRI) (602221)	Departure strategy:
	■ 1: XZ - simultaneous
	 2: XZ - sequential 3: ZX - sequential
/ Transv./longitud. ratio, ext. (PLVA) (602222)	Ratio for determination of longitudinal or transverse machining
/ Transv./longitud. ratio, int. (PLVI) (602223)	Ratio for determination of longitudinal or transverse machining
/ Min. roughing transv. length (RMPL) [mm] (602224)	Radius value for determination of the machining operation:
	 RMPL > I1: Without transverse roughing RMPL < I1: With transverse roughing RMPL = 0: Special case
/ Transverse angle variation (PWA) [°] (602225)	Tolerance range in which the first element is declared a transverse element

Length for external rough-machining enabling roughing beyond the target position.

... / Overhang length –outside (ULA) [mm] (602226)

1

Parameters: processing / ...

- ... / Overhang length –inside (ULI) [mm] (602227)
- ... / Retracting length -outside (RAHL) [mm] (602228)
- ... / Retracting length -inside (RIHL) [mm] (602229)
- ... / Cutting depth reduction factor (SRF) (602230)

... / Finishing (602300) / ...

- ... / Tool angle -ext./longitudinal (FALEW) [°] (602301)
- ... / Point angle -ext./longitud. (FALSW) [°] (602302)
- ... / Tool angle -ext./transverse (FAPEW) [°] (602303)
- ... / Point angle -ext./transv. (FAPSW) [°] (602304)
- ... / Tool angle –int./longitud. (FILEW) [°] (602305)
- ... / Point angle –int./longitud. (FILSW) [°] (602306)
- ... / Tool angle --int./transverse (FIPEW) [°] (602307)
- ... / Point angle -int./transverse (FIPSW) [°] (602308)
- ... / Machining -ext./longitudinal (FAL) (602309)
- ... / Machining --int./longitudinal (FIL) (602310)
- ... / Machining -ext./transverse (FAP) (602311)

... / Machining --int./transverse (FIP) (602312)

... / Adjacent angle tolerance (FNWT) [°] (602313)

Meaning

Length for internal rough-machining enabling roughing beyond the target position.

Retracting length for smoothing variants H = 1 and H = 2

Retracting length for smoothing variants H = 1 and H = 2

Factor for reducing the infeed value (cutting depth). For tools machining opposite to the main machining direction

- Tool angle for finishing tool
- Point angle for finishing tool
- Tool angle for finishing tool
- Point angle for finishing tool
- Tool angle for finishing tool
- Point angle for finishing tool
- Tool angle for finishing tool
- Point angle for finishing tool
- Strategy for finishing:
- 0: Full-surface finishing with optimum tool
- 1: Standard finishing; relief turns and undercuts machined with a suitable tool
- Strategy for finishing:
- 0: Full-surface finishing with optimum tool
- 1: Standard finishing; relief turns and undercuts machined with a suitable tool

Strategy for finishing:

- 0: Full-surface finishing with optimum tool
- 1: Standard finishing; relief turns and undercuts machined with a suitable tool

Strategy for finishing:

- 0: Full-surface finishing with optimum tool
- 1: Standard finishing; relief turns and undercuts machined with a suitable tool

Tolerance range for secondary cutting edge



Parameters: processing /	Meaning
/ Relief cutting angle (FFW) [°] (602314)	Minimum angle differential between the contour and secondary cutting edge
/ Approach/external finishing (ANFA) (602315)	Approach strategy:
	1: XZ - simultaneous
	■ 2: XZ - sequential
	3: ZX - sequential
/ Approach/internal finishing (ANFI) (602316)	Approach strategy:
	1: XZ - simultaneous
	2: XZ - sequential
	3: ZX - sequential
/ Departure/external finishing (ABFA) (602317)	Departure strategy:
	1: XZ - simultaneous
	2: XZ - sequential
	■ 3: ZX - sequential
/ Departure/internal finishing (ABFI) (602318)	Departure strategy:
	1: XZ - simultaneous
	2: XZ - sequential
/ Min. transv. finishing depth (FMPL) [mm] (602319)	Value for determination of the machining operation:
	 Without inside contour: Always with transverse cut With inside contour, FMPL >= I1: Without transverse cut
	■ With inside contour, FMPL < I1: With transverse cut
/ Max. finishing cutting depth (FMST) [mm] (602320)	Permissible infeed depth for non-machined undercuts
	FMST > ft: With undercut machining
	■ FMST <= ft: Without undercut machining
/ No. rev. for chamfer/rounding (FMUR) (602321)	Minimum number of revolutions; feed rate is reduced automatically.
/ Recessing (602400) /	
/ Approach/external recessing (ANESA) (602401)	Approach strategy:
	1: XZ - simultaneous
	2: XZ - sequential
	■ 3: ZX - sequential
/ Approach/internal recessing (ANESI) (602402)	Approach strategy:
	■ 1: XZ - simultaneous
	2: XZ - sequential
	3. ZA - Sequentiai

Parameters: processing /	Meaning	rs
/ Departure/external recessing (ABESA) (602403)	Departure strategy: 1: XZ - simultaneous 2: XZ - sequential 3: ZX - sequential 	aramete
/ Departure/internal recessing (ABESI) (602404)	Departure strategy: 1: XZ - simultaneous 2: XZ - sequential 3: ZX - sequential 	8.2 P
/ Appr./ext. contour recessing (ANKSA) (602405)	Approach strategy: 1: XZ - simultaneous 2: XZ - sequential 3: ZX - sequential 	
/ Appr./int. contour recessing (ANKSI) (602406)	Approach strategy: 1: XZ - simultaneous 2: XZ - sequential 3: ZX - sequential 	
/ Dep./ext. contour recessing (ABKSA) (602407)	Departure strategy: 1: XZ - simultaneous 2: XZ - sequential 3: ZX - sequential 	
/ Dep./int. contour recessing (ABKSI) (602408)	Departure strategy: 1: XZ - simultaneous 2: XZ - sequential 3: ZX - sequential 	
/ Recessing width divisor (SBD) (602409)	Value for selecting the tool for contour recessing with linear elements at the recess base	
/ Type of oversize (KSAA) (602410)	Oversize for contour recessing with contour valleys. Standardized recesses are completed in one machining cycle.	
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 (KSLA) (602411)	Equidistant oversize or longitudinal oversize	

Parameters: processing /	Meaning
/ Face oversize (KSPA) (602412)	Transverse oversize
/ Recessing width factor (SBF) (602413)	Factor for determining the maximum tool offset
/ Recessing/finishing (602414)	Sequence of finishing cuts:
	1: Part a horizontal element (previous behavior)2: Move through with lift-off
/ Thread cutting (602500) /	
/ Approach/ext. – thread (ANGA) (602501)	Approach strategy:
	■ 1: XZ - simultaneous
	2: XZ - sequential
/ Approach/int. – thread (ANGI) (602502)	Approach strategy:
	 1: XZ - Simultaneous 2: XZ - sequential
	■ 3: ZX - sequential
/ Departure/ext. – thread (ABGA) (602503)	Departure strategy:
	■ 1: XZ - simultaneous
	2: XZ - sequential
	3: ZX - sequential
/ Departure/int. – thread (ABGI) (602504)	Departure strategy:
	 1: XZ - simultaneous 2: XZ - sequential
	■ 3: ZX - sequential
/ Thread starting length (GAL) [mm] (602505)	Default value for "run-in length B"
/ Thread run-out length (GUL) [mm] (602506)	Default value for "overrun length P"
/ Measuring (602600) /	
/ Measuring loop counter (MC) (602602)	Definition of the measurement/loop intervals.
/ Measuring path length in Z (MLZ) (602603)	Length of departure path in Z
/ Measuring path length in X (MLX) (602604)	Length of departure path in X
/ Measurement oversize (MA) (602605)	Oversize on the element to be measured
/ Measuring cut length (MSL) (602606)	Measuring cut length
/ Drilling (602700) /	

8.2 Parameters

ers	
leti	
am	
Dar	
2 N	
$\mathbf{\infty}$	

Parameters: processing /	Meaning
/ Approach/front face – drilling (ANBS) (602701)	Approach strategy: 1: XZ - simultaneous 2: XZ - sequential 3: ZX - sequential
/ Appr./lateral surface – drilling (ANBM) (602702)	Approach strategy: 1: XZ - simultaneous 2: XZ - sequential 3: ZX - sequential
/ Dep./front face – drilling (ABBS) (602703)	Departure strategy: 1: XZ - simultaneous 2: XZ - sequential 3: ZX - sequential
/ Dep./lateral surface – drilling (ABBM) (602704)	Departure strategy: 1: XZ - simultaneous 2: XZ - sequential 3: ZX - sequential
/ Internal safety clearance (SIBC) [mm] (602705)	Retraction distance for deep-hole drilling "B"
/ Driven drill (SBC) (602706)	Safety clearance for driven tools
/ Non-driven drill (SBCF) (602707)	Safety clearance for tools that are not driven
/ Driven tap (SGC) (602708)	Safety clearance for driven tools
/ Non-driven tap (SGCF) (602709)	Safety clearance for tools that are not driven
/ Drilling depth factor (BTCF) (602710)	Factor for calculation of the first drilling depth for deephole drilling
/ Depth reduction (BTRC) [mm] (602711)	Reduction for deep-hole drilling
/ Diameter tolerance/drill (BDT) [mm] (602712)	For drill selection
/ Milling (602800) /	
/ Approach/front face – milling (ANMS) (602801)	Approach strategy: 1: XZ - simultaneous 2: XZ - sequential 3: ZX - sequential
/ Appr./lateral surface – milling (ANMM) (602802)	Approach strategy: 1: XZ - simultaneous 2: XZ - sequential 3: ZX - sequential

Parameters: processing /	Meaning
/ Dep./front face – milling (ABMS) (602803)	Departure strategy:
	1: XZ - simultaneous
	2: XZ - sequential
	3: ZX - sequential
/ Dep./lateral surface – milling (ABMM) (602804)	Departure strategy:
	1: XZ - simultaneous
	2: XZ - sequential
	3: ZX - sequential
/ Safety clearance in infeed dir. (SMZ) [mm] (602805)	Distance between starting position and top edge of object to be milled
/ Safety clearance, milling dir. (SME) [mm] (602806)	Distance between milling contour and side of mill
/ Oversize in milling direction (MEA) [mm] (602807)	Oversize
/ Oversize in infeed direction (MZA) [mm] (602808)	Oversize
/ ExpertPrograms /	
/ Expert programs (606800) /	Subprograms adapted to the machine configuration
/ Parameter list	Key of the parameter list
/ Parameter lists for expert programs (606900) /	
/ Name of the expert program	Name of the expert program (without path information)
/ Parameter	Value of the parameter

Descriptions of the most important 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

External on blank part [SAR]

Internal on blank part [SIR]

TURN PLUS takes SAR/SIR into account

- 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

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: **SCK** parameter)

Global safety clearance G147 in infeed direction

Default setting for the global safety clearance in infeed direction (start unit: **SCI** parameter)

Global oversize in X direction





Global safety clearance

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: \mathbf{K} parameter)

Front edge of chuck on main spindle

Z position of the front edge of the chuck for calculating the workpiece zero point (AWG submode)

Front chuck edge on counterspindle

Z position of the front edge of the chuck for calculating the workpiece zero point (AWG submode)

Jaw width on main spindle

Jaw width in Z direction for calculating the workpiece zero point (AWG submode)

Jaw width on counterspindle

Jaw width in Z direction for calculating the workpiece zero point (AWG submode)

Further global technology parameters

Global technology parameters

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

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

Default setting for the global safety clearance in infeed direction (start unit: **SCI** parameter)

Global technology parameters

Rotational direction for new units

Default for the spindle direction of rotation **MD** when creating/ opening a new unit ("Tool" tab)

Conversion of the ICP contours

Select type of conversion of ICP contours

- 0: The calculated parameters are output
- 1: The programmed parameters are output

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</p>
- Tool selection: UBD1 <= db1 <= DB1max

2nd drilling diameter limit [UBD2]

- 2nd predrilling step: if UBD2 < DB2max</p>
- 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).



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.





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



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</p>
- 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



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





Roughing – Machining analysis

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</p>
 - 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 relief turns (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 "fullsurface 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</p>
- 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







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</p>

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/departure strategy:

- 1: Move simultaneously in X and Z directions
- 2: First X, then Z direction
- 3: First Z, then X direction







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 roughmachined 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

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 roughmachining 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



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

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

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

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]







8.2 Parameters

8.3 Submode Transfer

The **Transfer** submode 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 MANUALplus 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 MANUALplus 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 EXTERNAL ACCESS 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 MANUALplus supports SMB networks (Server Message Block, WINDOWS) and NFS networks (Network File Service).
- **USB** storage devices can be connected directly to the control. The MANUALplus uses only the first partition of a USB storage device.

Danger of collision!

Other computer systems in the network may overwrite MANUALplus programs. Organize your network in such a way that only authorized persons have access to MANUALplus.



∕!∖

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 MANUALplus opens the ${\it Network}\ connection$ dialog box. This dialog box is for the settings for the connection target.



Press the **Config.** soft key (login required). The **Network configuration** dialog box opens.



➡ Machine	🚯 smart.Tu	rn	🛱 Tool edi	tor			ransfei	
	1	Network config	uration	_	_	_		_
		Control name DHCP		MANUAL OFF _	.plus6	28		
		IP address Subnet mask		192 255	168 255	000 255	000	-1:
		Broadcast Gateway		255 192	255 168	255 080	000 254	
	-							
USB connection acti	ve: E:\TRANSFER							11:36
						s	ave	Back

Ethernet interface (with software 548328-xx)

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 -
- ▶ time0-
- ▶ soft -

Select project folder: The MANUALplus 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.



.3 Submode Transfer

Network configuration soft keys Create Creates a folder of the specified

Create transfer folder	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.
MORE 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 (with software 54843x-xx)

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 Activecolumn)
	 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

Ø HEIDEN	Computername Schultstellen Internet Ping/Routing NFS UID/GID (DHCP-Server	••
	X ethe XXS ONCI-LAN	Ē
	Attivieren Deaktivieren Konfigurieren IP-forwarding IP-forwarding urlauben Pakete, die an einer Schnittstelle ankommen, dürfen	
	OK Angenden Berechtigung Abbrechen	

Ì 🖾 т.

•••

)(^

<u> </u>
Ð
ų.
0
J
<u> </u>
A \
<u>_</u>
σ
0
_
U)
~
<u></u>
<u>m</u>
~

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 whice 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					

Contraction of the local division of the loc	Schnittstelle konfigurieren -×							
Ø HEIDE	Status ☐ Schnittstelle aktiv Name: ethē Steckerverbindung: X26							
	Einstellungen Status-Information							
	Profil							
	Name: DHCP-LAN Speichern Laden Löschen							
	IP-Adresse							
	⊙ IP-Adresse automatisch beziehen (DHCP) O IP-Adresse manuell einstellen							
	Adresse: 0. 0. 0. 0							
	Subnet-flaske: 0, 0, 0, 0							
	Donain Name Server (DNS)							
	Falls DHCP aktiviert ist wird der DNS Server 1. Server:							
	vom DHCP-Server auf diesem Interface bezogen. 2. Server:							
	Domänenname:							
	Default felenen							
	Default GV manuell konfigurieren							
	Falls DHCP aktiviert ist, wird das Default							
	bezogen.							

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

→ Maschi	ine	🚯 smart.Tu	cn .	🎙 Werkzeug	-Editor	💾 Transfer	Ì	
Verbindun	Netzwerkeinste	llungen				_ (×	
HEIDEN	Computernamen Schr	ittstellen Intern	et Ping/Routing	NFS UID/GID DHC	P-Server			
	Provy © Direkte Verbindung zum Internet / MAT Internet-Anfragen Leitet die Steurung an das Default- Gateway welter und mässen dast üher Retvenk Address Translation weltergeweine weten.							
	Advesse'						- •	
	Port:	8						
		1						
	Server für Fernwartung konfiguritet Gar Haschianhersteller var Anslichterung der Haschion. Server sollten Sie ner auf Anweisung des Kundendienstes verändern. Verwende eigenen HTP User Agant-Text HTP User Agant-Text							
	Zertifikat Serv	er	Beschrei	bung			1	
	nc5 rem	oteservice.heidenh	ain.de Heidenha	in Remote Servic	8			
		Hinzufüg	en	<u>_</u>	öschen		T	
Keine V	ŪK	A	nyenden	OEM Berechtigun	g <u>A</u> b	brechen	4:23:30	
ŌK	Anvenden	Abbrechen		Dirskt/ Proxy-Server	<u>H</u> inzufügen	Löschen	OEN Berechtigung	

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.

→ rlasch: Verbindun	Netzwer	keinste	llungen		H werkzeug-	altor		× .
C HEIDEN	UID/GID f	ir NFS-SI	nares setzen	leout ze z-Kennung	und die Benutzer-G	verver)
	User UID	598	die den Zugziff Eingabebereich:	auf NFS-Server 100 bis 65535.	festlegen.	reppo eneorn,		-T ⁰
	User GID	188 [2	1					
								T
Keine V		Ōĸ		Anvenden	OEM Bexechtigung	Āb	brechen	4:24:40
<u>0</u> K	Any	enden	Abbrechen					0EN Bevechtige

<u> </u>
Ð
Ť
S
Ċ
ų
•
Ð
Ŧ
×
0
S
3
m.
S S

SettingMeaningDHCP 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

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: Set 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.



➡ Mas	schine		Ìť	S sr	art.Turn	À	Werkzeug-	Editor	🗎 🗄 Transfer	
erbind	lung wä	hlen		_	_	_	_	_		
ØHEID	ENHAIN einric	CNCP1 hten	LOT648) *
Netzver	klaufwer	k								
Nount	Auto	Тур	Laufverk	ID	Server	Freigabenane	Benutzer	Passwort	Passwort erfragen?	Optioner
		cifs	к:	1	de03pc0094	transfer	JH/A11929	no	2	
							D ₂			
<										
Tren	nen		Auto		Hinzufi	igen	Entfernen	B	opieren	Bearbeite
Status	log									
Comnand Executi	: mount. on succe	cifs //	'de03pc0094	/trans	fer /nnt/k -o	credentials=/t	p/henount_k,	noserverino,	uid-user,gid-user	
				END						
×.										
						Leeren				
OK						Angenden				Abbrech
Netzw	erk	Netz Verbi	werk ndung	Ord	ner Or	dner	10	Freie Ausvahl	Anvahl	Zurüc

USB connection

Select the Organization mode and plug the USB storage device in at the USB port on the MANUALplus.

TRANSFER	Press the Transfer soft key (login required).
Con- nections	Press the Connections soft key.
USB	Press the USB soft key.
The MANU	ALplus opens the USB dialog box. This dialog box is for the



Soft keys
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 f	or USB connection
Create transfer folder	Creates a folder of the specified name on the USB storage device.
K	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 MANUALplus 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 submode.

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
- Miscellaneous functions: Importing the cycle programs and DIN programs of the MANUALplus 4110 and the tool data of the CNC PILOT 4290

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 (submode Teach-in) *.gmz 			
\ncps	DIN programs (mode 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

		- macnine		V Smart. I	u111	H 1001 60	1101		
TRANSFER	Press the Transfer soft key (login required).	TNC:\nc_pros Name Ø 1.nc Ø 2.nc Ø 3.nc Ø 4.nc	a\ncps*.nc Size 6274 46: 444 584	- DIN programs Changed on 3 2014-10-23 0 2 2014-10-23 0 9 2014-10-23 0 5 2014-10-23 0	E:\T Na 8:56 Ø1. 8:52 Ø12 8:22 Ø2. 8:01 Ø3.	RANSFER\ncps\= ne nc 3,.nc nc nc	nc Size Chang 6278 2014- 1646 2014- 462 2014- 462 2014-	ed on	
Con- nections	Press the Connections soft key.	© _generated © _Kopie_fue © bar.nc © Huelse.nc © velle.nc	481 i_by_AAG.nc er_AAG.nc 215 471 198	2014-10-23 0 2014-10-23 0 2014-10-23 0 5 2014-07-08 1 9 2014-10-09 2 3 2014-07-10 1	8:17 Ø 94 8:17 Ø 94 8:17 Ø _K 2:34 Ø 80 2:83 Ø 80 2:30 Ø 80 Ø 80 Ø 80 Ø 80 Ø 80 Ø 80 Ø 80 Ø 8	_variabl.nc 0318_01.nc opie_fuer_AAG. hr_001.nc ku_Bohrungen_1 ku_Flansch.nc ku_Huelse.nc ku_Mantel.nc ku_orange.nc	1438 2014- 1190 2014- 1495 2014- 1495 2014- 958 2014- 8129 2014- 2171 2014- 557 2014-1	07-67 03:18 10-22 15:10 10-22 14:51 06-23 15:20 07-04 10:31 07-04 10:40 10-22 14:49 07-04 10:03 07-04 10:56	
USB	Press the USB soft key.				& Dol & Fe: & Fe: & He: & Hu: & Jol & Jol	ku_Y-Mantel_C. lge_02.nc lge_03.nc lix_Cyl.nc lix_Rohr.nc else.nc hn00.nc hn2.nc bn4.nc	2014-0 3947 2014-0 1626 2014-0 435 2014-0 441 2014-0 4719 2014-0 1163 2014-0 1163 2014-0	07-04 10:07 04-10 17:22 06-04 10:38 07-02 13:17 05-08 14:59 - 10-09 22:03 07-02 13:26 07-02 13:31 06-18 16:06	
	Dragg the Naturnk astr key	USB connec	tion active:	E-\TRANSEED	@ Jol	hn5.nc	431 2014-	07-02 13:53 07-02 13:59 _	
Network	Fless the network solt key.	Transmit	Delete	Program selection	Program view	Mark everything	Mark	Internal project	Back
Selection	Select a project folder and press the Selection soft	Soft	keys f	or prog	gram g	group s	electi	on	
		DI progr	N ams	*.nc: DIN and smart.Turn main programs. The Transfer submo				main submoo	de
Connect	Connect (network) soft key.			scans and a transf	sks wl er the	nether y m as w	s for si /ou wa ell.	ubprog ant to	rams
Back	Return to data selection.	DI subpro	N ogram	*. ncs subpr transf	: DIN a ogram erred.	and sma is. Help	art.Tur graph	n ics are	also
Programs	Switch to program transfer.	Cyc. progr	le cams	*.gm subm subpr asks v them	z: Cycl ode so ogram wheth as we	e progra cans the is and li er you v ell.	ams. T e progi CP cor want to	The Tra in rams for ntours, no transf	nsfer or and fer
Program	Open a selection of program types.	IC	P	ICP contours for cycle programs					
		conto	ours	*.gmi (turning contour)					
DIN	Activate DIN programs (or other program types) for			■ *.g	mr (w	orkpiec	e-blanl	k conto	ur)
	transfer.			*.gms (C axis, face)					
,				*.gmm (C axis, lateral surface)					;)
	With the transfer of automatic jobs, the control	Ne» file	t type	Switc You c here.	hes be an als	etween o select	possib autor	ole file to natic jo	ypes. ıbs
	automatically transfers the selected jobs with all main programs and the subprograms contained in these.	Fre exter	e :nal	Allow USB s projec	s sele storag ct folde	cting pr e device er.	ogram e witho	n files o out usir	n a ng a
				Mask	ing the	e file na	mes v	vithin tł	ne

File mask

selected program group.

🗄 Transfer

Tool editor



Selecting the program

In the window on the left, the MANUALplus 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 **arrow 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 MANUALplus 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 $\ensuremath{\text{Program}}$ view soft key.

Press the Transmit or Receive soft key to start transferring the files.

During transfer, the MANUALplus 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 MANUALplus 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 MANUALplus 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 142).



The **Internal project** soft key enables you to manage projects and transfer complete project folders (see also "Project management" on page 142).





Soft keys	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 MANUALplus (login required).

Parameter selection

You can also create a parameter backup without connecting to an external storage device.

Open the parameter transfer.	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 MANUALplus 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.

➡ Machine		smart.Tu	ırn		🍴 Tool edi	tor	💾 Transfe	r
TNC:\bck\par Name Om_Confi Sryc_Confi Jrr_Confi Jrr_Confi Jrr_Confi Jrr_Confi Jrr_2014101	a(*.zip = Fai Size g_Fileszip g_Fileszip g_Fileszip 5.zip 170 5.zip 10002	Changed on Changed on 2814-18-15 6; 2814-18-15 6; 2814-18-15 6; 2814-18-15 6; 2814-18-15 6;	7:12 7:12 7:12 7:12 7:12	E:\TRA Name ⊘_Oem ⊘_Sys	NSFER\para\•. Config_files Config_files	zip Size Change zip 2014-1 zip 2014-1	d on 3-15 07:12 3-15 07:12	
USB connect	tion active:	E:\TRANSFER						11:37
Transmit	Delete	Back up parameters	Rest Paramo	ore eters	Mark everything	Mark	File mask	Back

8.3 Submode Transfer

Soft keys	s 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.
File mask	Opens the file mask

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 MANUALplus (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

Backup tools

> Tool list

Turret list

Magazine list

Mark

When creating a tool backup, you can choose whether to save the data of specific tools or all tools. Select the desired tools in the tool list or turret list:

Open the magazine list (machine-dependent)

Press the **Backup tools** soft key

➡ Machine		smart.T	urn	🍴 🏷 Tool ed	itor	💾 Transfer	
TNC:\bck\tool	*.zip - To	ols	E:	\TRANSFER\tool*	.zip		
Name	Size	Changed on		Name	Size Changed	on	1
Ø TO_020.zip Ø TO_20141023	15014 .zip 14987	2014-10-15 0 2014-10-23 0	97:11 Ø	TO_020.zip TO_20141023.zip	15014 2014-18 14987 2014-18	-15 07:11 -23 07:21	0 0
							T
USB connecti	ion active:	E:\TRANSFER					11:37
Transmit	Delete	Backup tools	Restor Tools	e Mark everything	Mark	File mask	Back

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).

Confirm selection

Mark tools

Open the tool list

Open the turret list



The MANUALplus displays a selection window. Here you can define which tool data you want to save.

Selection for the content of backup files:

- Tools
- Tool texts
- Technology data
- 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 backup files are displayed. With the **Tool list** soft key, you can select individual tools from a backup file.

Choose the tool data you want to restore from the backup file.

Service files

8.3 Submode Transfer

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 MANUALplus 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



Saving service files

You can also create service files without connecting to an external storage device.



Enter a file name under which you want to save the service file.



Press 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



- 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 MANUALplus 620. 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 MANUALplus. 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

➡ Machine	🚯 si	nart.Ti	Jrn	🏷 Tool edi	tor	💾 Transfer		
Import from USB	device E:		_	_	_			
	Look in	: 🖃 E :	\TRANSFER\			*		
CHEIDENHAIN CM	CPILOT648	🗎 da	ata				0.0	
• = C:		⊇ gʻ	tz					
TRANSFER			CD2				-R	
∞ a9G:		😑 pa	ara				-02	
*⊋H:		P:	roject					
		a te	ool					
		a te	ool_4290					
1	File name							
	File name: *							
	oave as type	. 1100.	15			1		
USB connection	active: E:\TRAN	ISFER					11:37	
Details			Sorting	Mark everything	Alphabetic keyboard	Start import	Back	

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).

ENT

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 everything	Mark all NC programs.
Start import	Start the import filter for converting the program(s) to the MANUALplus format.



The names of imported cycle programs, ICP contour descriptions, DIN programs and DIN subprograms are given the prefix "CONV_..." In addition, the MANUALplus adapts the extension and imports the NC programs to the correct folders.

Converting cycle programs

MANUALplus 4110 and MANUALplus 620 have different solutions for tool management, technology data, etc. Also, the cycles of the MANUALplus 620 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 into the fields **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 MANUALplus 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 **MANUALplus 4110 DIN programs**:

- 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 MANUALplus 620.
- 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 **CNC PILOT 4290 DIN programs**:

- **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 MANUALplus and then testing them before using them for production.



Importing tool data of the CNC PILOT 4290

The format of the tool list of the CNC PILOT 4290 differs from the format of the MANUALplus 620. You can use the program converter to adapt tool data to the new control.

Importing tool data 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	Press the Tools soft key.
• ENT	Use the cursor keys to select the folder, then press the Enter key to switch to the right window.
Use the arrow	keys to select the tool data.

Mark everything	Mark all tool data.
Start import	Start the import filter for the conversion.

The MANUALplus 620 generates a table named **CONV_*.HTT** for each imported file. You can load this table with the Restore function by setting the file mask to the ***.htt** file type.

➡ Machine	🛃 Machine 📄 🚯 smart.Turn 📄 🎦 Tool editor 📄 🖺 Transfer								
Import from U	SB device E			_	_	_			
		Look in:	⊨ E:'	\TRANSFER\too	1_4290\		•		
C HEIDENHAIN	CNCPILOT644 FER a s s ject le l 1 1 4290)	BS	IPVZDOR_4298.v PVZDRE.vkz PVZFRA.vkz	vkz				
	File name: *								
	Save as type: Tools								
USB connecti	ion active:	E:\TRANS	FER					11:37	
Details				Sorting	Mark everything	Alphabetic keyboard	Start import	Back	



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 601).

Connect a USB stick and switch to the Organization mode of operation.



→ Machine smart.Turn Tool editor Basic data Load NC software My CNC PLC: TNC: 2 REDO_C_TURN_MLST4_500Devel... 2 REDO_C_TURN_MLST4_500Devel... 2 REDO_C_TURN_MLST4_500Devel... 2 REDO_C_TURN_MLST4_500Devel... 🔹 🗀 confia config.bak
config setun zin ⇒updates ⊇zip File name: File type: ZIP files (*.zip) 5 PAGE PAGE SELECT UNDO PATH END UPDATE ÌB → Machine smart.Turn Tool editor Basic data My CNC PLC: TNC:) setup.zi 10 🗉 🗀 zip IISBA ⇒ ⊇project1 ⇔data Do you really want to switch off? YES NO

File name

YES

File type: ZIP files (*.zip)



The MANUALplus 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 MANUALplus automatically restarts.





Tables and overviews

9.1 Thread pitch

Thread parameters

To determine the thread parameters, the MANUALplus 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 611.).
- 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 \leq 6: ac = 0.5
- Thread pitch <= 13: ac = 1

Thread type Q		F	Р	R	Α	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°

				~	VV
	-	_	_	_	_
External	*	0.61343*F	F	30°	30°
Internal	*	0.54127*F	F	30°	30°
External	*	0.61343*F	F	30°	30°
Internal	*	0.54127*F	F	30°	30°
External	*	0.61343*F	F	30°	30°
Internal	*	0.54127*F	F	30°	30°
External	*	0.8*F	F	30°	30°
Internal	*	0.8*F	F	30°	30°
External	*	0.8*F	F	30°	30°
Internal	*	0.8*F	F	30°	30°
External	*	0.8*F	F	30°	30°
Internal	*	0.8*F	F	30°	30°
External	*	0.8*F	F	30°	30°
Internal	*	0.8*F	F	30°	30°
	External Internal External Internal External Internal External Internal External Internal External Internal	External * Internal *	External * 0.61343*F Internal * 0.54127*F External * 0.61343*F Internal * 0.8*F Internal * 0.8*F	External * 0.61343*F F Internal * 0.54127*F F External * 0.61343*F F Internal * 0.61343*F F Internal * 0.54127*F F External * 0.61343*F F Internal * 0.61343*F F External * 0.61343*F F Internal * 0.61343*F F External * 0.61343*F F External * 0.61343*F F External * 0.61343*F F Internal * 0.8*F F External * 0.8*F F Internal * 0.8*F F External * 0.8*F F Internal * 0.8*F F Internal * 0.8*F F Internal * 0.8*F F Internal * 0.8*F F	External * 0.61343*F F 30° Internal * 0.54127*F F 30° External * 0.61343*F F 30° External * 0.61343*F F 30° Internal * 0.54127*F F 30° External * 0.61343*F F 30° External * 0.61343*F F 30° Internal * 0.61343*F F 30° External * 0.61343*F F 30° Internal * 0.61343*F F 30° External * 0.84F F 30° External * 0.8*F F 30° Internal * 0.8*F F 30° External * 0.8*F F 30° Internal * 0.8*F F 30° Internal * 0.8*F F 30° Internal * 0.8*F F 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				

1

Q = 8 Cylindrical round thread

Thread pitch
2.54
3.175
4.233
6.35
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	2.82222222
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	5.64444444
0.216''	5.4864	1.058333333	2 1/4''	57.15	5.64444444
1/4''	6.35	1.27	2 1/2''	63.5	6.35
5/16''	7.9375	1.41111111	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	1.41111111
0.112''	2.8448	0.529166666	5/8''	15.875	1.41111111
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	1.41111111
1/4''	6.35	0.79375	1 1/8''	28.575	1.41111111
5/16''	7.9375	0.79375	1 3/16''	30.1625	1.41111111
3/8''	9.525	0.79375	1 1/4''	31.75	1.41111111
7/16''	11.1125	0.907142857	1 5/16''	33.3375	1.41111111
1/2''	12.7	0.907142857	1 3/8''	34.925	1.41111111
9/16''	14.2875	1.058333333	1 7/16''	36.5125	1.41111111
5/8''	15.875	1.058333333	1 1/2''	38.1	1.41111111
11/16''	17.4625	1.058333333	1 9/16''	39.6875	1.41111111
3/4''	19.05	1.27	1 5/8''	41.275	1.41111111
13/16''	20.6375	1.27	1 11/16''	42.8625	1.41111111
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	1.41111111	5''	141.3	3.175
3/8''	17.145	1.41111111	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	1.41111111	1''	33.401	2.208695652
3/8''	17.145	1.41111111			

9.2 Undercut parameters

DIN 76-undercut parameters

The MANUALplus 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	1	К	R	W	Thread pitch	1	К	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	L I	К	R	W	Thread pitch	1	К	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 MANUALplus 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	1	К	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	К	R	W	Р	Α
<=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

Components	MC 7410T main computer with
componente	CC 61xx controller unit
	■ 12.1-inch TFT color flat-panel display
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: 0.2 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

Tables and overviews

1

Fast and simple troubleshooting through integrated diagnostic aids

■ Operation: 5 °C to 40 °C

■ Storage: –20 °C to +60 °C



Diagnostics

Ambient temperature

User functions	
Configuration	Basic version: X and Z axis, spindle
	Y axis (optional)
	Driven tool (optional)
	C axis (optional)
	B axis (optional)
	Digital current and speed control
	Rear-face machining with the opposing spindle (optional)
Machine mode of 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 submode	Sequential linking of Teach-in cycles, where each machining cycle is run immediately after input, or is graphically simulated and subsequently saved.
Program run submode	All are possible in single-block and full-sequence modes:
	DINplus programs
	smart.Turn programs (optional)
	Tech-in programs (optional)
Setup functions	Setting the workpiece zero point
	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 – Cycle programming (optional)	Turning cycles for simple and complex contours, and contours described with ICP
	Contour-parallel turning cycles
	Recessing cycles for simple or complex contours, as well as 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)	Contour definition with linear and circular contour elements
(optional)	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
	Litting 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, magazine assignment and other setup information in the smart.Turn program
	Parallel programming
	Parallel simulation

1

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, magazine 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 pencil-trace 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/magazine

assignment

Tables and overviews

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 MANUALplus 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)



User functions	
User functions Conversational languages	 ENGLISH GERMAN CZECH FRENCH ITALIAN SPANISH PORTUGUESE SWEDISH DANISH FINNISH DUTCH POLISH HUNGARIAN RUSSIAN CHINESE CHINESE_TRAD SLOVENIAN ROMANIAN SLOVAK
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

Control software for PCs for programming, archiving, and training for the MANUALplus:

 \blacksquare Full version with license for single station or multiple stations

Demo version (free of charge)

DataPilot CP 640, MP 620

Option number	Option	ID	Description
0 to 7	Additional Axis	354540-01	Additional control loops
		353904-01	
		353905-01	
		367867-01	
		367868-01	
		370291-01	
		353292-01	
		353293-01	
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	Thread
			Thread recutting
			Handwheel superimposition during thread cutting
17	Touch 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
			Automatically measuring workpieces
18	HEIDENHAIN DNC	526451-01,	Communication with external PC applications over COM component
42	DXF import	632231-01	DXF import
			Loading of DXF contours
54	B-axis machining	825742-01	Machining with the B axis
			Rotating the machining position of the tool

Option number	Option	ID	Description
55	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
77	4 additional axes	634613-01	4 additional control loops
78	8 additional axes	634614-01	8 additional control loops
94	Parallel axes	661881-01	Support of parallel axes (U, V, W)
101 to	OEM option	579651-01	Options of the machine tool builder
130		to	
		579651-30	
131	Spindle synchronism	806270-01	Synchronization (of two or more spindles)
132	Opposing spindle	806275-01	Opposing spindle (spindle synchronism, rear-face machining)
135	Synchronizing functions	1085731-01	Expanded synchronization of axes and spindles
143	Load Adaptive Control (LAC)	800545-01	LAC: Dynamic adaptation of the control parameters
151	Load monitoring	1111843-01	Monitoring of the tool load

9.4 Compatibility in DIN programs

The format of DIN programs of the CNC PILOT 4290 predecessor control differs from the format of the MANUALplus 620. However, you can use the program converter to adapt programs of the predecessor control to the new control.

When opening an NC program, the MANUALplus 620 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** submode (**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

Variable programming:

- 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 MANUALplus 620 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 codes:

- The following G functions are currently not supported by the CNC PILOT 640: G62, G63, G98, G162, G204, G710, G906, G907, G915, G918, G975.
- 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 G48 because the principle of function has changed.
- 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 MANUALplus 620

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 MANUALplus 620
- 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]	V
	SCHEIBENMAGAZIN [PLATE MAGZN.]	V
	REVOLVER [TURRET]	V
	MAGAZINE	V
	SPANNMITTEL [CLAMPS]	Х
Contour definition	CONTOUR / CONTOUR GROUP	Х
	ROHTEIL [BLANK]	V
	FERTIGTEIL [FINISHED]	V
	HILFSKONTUR [AUXIL_CONTOUR]	V
C-axis contours	STIRN [FACE_C]	V
	RUECKSEITE [REAR_C]	V
	MANTEL [LATERAL_C]	V
Workpiece machining	BEARBEITUNG [MACHINING]	V
	ZUORDNUNG [ASSIGNMENT]	•
	ENDE [END]	V
Subprograms	UNTERPROGRAMM [SUBPROGRAM]	V
	RETURN	V
Others	CONST	V
Y-axis contours	STIRN_Y [FACE_Y]	V
	RUECKSEITE_Y [REAR_Y]	V
	MANTEL_Y [LATERAL_Y]	V

1

G commands for turning contours		
Workpiece-blank definition	G20-Geo Chuck part, cylinder/tube	V
	G21-Geo Cast part	V
Basic contour elements	G0-Geo Starting point of contour	V
	G1-Geo Line segment	V
	G2-Geo Circular arc with incremental center dimensioning	V
	G3-Geo Circular arc with incremental center dimensioning	V
	G12-Geo Circular arc with absolute center dimensioning	V
	G13-Geo Circular arc with absolute center dimensioning	V
Contour form elements	G22-Geo Recess (standard)	V
	G23-Geo Recess/relief turn	V
	G24-Geo Thread with undercut	V
	G25-Geo Undercut contour	V
	G34-Geo Thread (standard)	V
	G37-Geo Thread (general)	V
	G49-Geo Bore hole at turning center	V
Help commands for contour definition	G7-Geo Precision stop on	V
	G8-Geo Precision stop off	V
	G9-Geo Precision stop blockwise	V
	G10-Geo Peak-to-valley height	Х
	G38-Geo Feed rate reduction	Х
	G39-Geo Attributes for superimposed elements	-
	G52-Geo Blockwise oversize	Х
	G95-Geo Feed per revolution	Х
	G149-Geo Additive compensation	Х

programs
Ζ
Ξ
•=
oatibility
č
Con
9.4

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	V
	G101-Geo Line segment on front face	V
	G102-Geo Circular arc on front face	~
	G103-Geo Circular arc on front face	V
	G300-Geo Bore hole on front face	V
	G301-Geo Linear slot on front face	V
	G302-Geo Circular slot on front face	V
	G303-Geo Circular slot on front face	V
	G304-Geo Full circle on front face	V
	G305-Geo Rectangle on front face	V
	G307-Geo Eccentric polygon on front face	V
	G401-Geo Linear pattern on front face	V
	G402-Geo Circular pattern on front face	V
Lateral surface contours	G110-Geo Starting point of lateral surface contour	V
	G111-Geo Line segment on lateral surface	V
	G112-Geo Circular arc on lateral surface	V
	G113-Geo Circular arc on lateral surface	V
	G310-Geo Bore hole on lateral surface	V
	G311-Geo Linear slot on lateral surface	V
	G312-Geo Circular slot on lateral surface	V
	G313-Geo Circular slot on lateral surface	V
	G314-Geo Full circle on lateral surface	V
	G315-Geo Rectangle on lateral surface	V
	G317-Geo Eccentric polygon on lateral surface	V
	G411-Geo Linear pattern on lateral surface	V
	G412-Geo Circular pattern on lateral surface	V

G commands for Y-axis cont	ours	
XY plane	G170-Geo Starting point of contour	V
	G171-Geo Line segment	V
	G172-Geo Circular arc	V
	G173-Geo Circular arc	V
	G370-Geo Hole	V
	G371-Geo Linear slot	V
	G372-Geo Circular slot	V
	G373-Geo Circular slot	V
	G374-Geo Full circle	V
	G375-Geo Rectangle	V
	G376-Geo Single surface	V
	G377-Geo Eccentric polygon	V
	G471-Geo Linear pattern	V
	G472-Geo Circular pattern	Va
	G477-Geo Centric polygon	V

G commands for Y-axis contours		
YZ plane	G180-Geo Starting point of contour	V
	G181-Geo Line segment	V
	G182-Geo Circular arc	V
	G183-Geo Circular arc	V
	G380-Geo Hole	V
	G381-Geo Linear slot	V
	G382-Geo Circular slot	V
	G383-Geo Circular slot	V
	G384-Geo Full circle	V
	G385-Geo Rectangle	V
	G387-Geo Eccentric polygon	V
	G481-Geo Linear pattern	V
	G482-Geo Circular pattern	V
	G386-Geo Single surface	V
	G487-Geo Centric polygon	V

G commands for machining		
Tool positioning without machining	G0 Positioning at rapid traverse	V
	G14 Move to the tool change position	V
	G701 Rapid traverse to machine coordinates	V
Simple linear and circular movements	G1 Linear movement	V
	G2 Circular movement with incremental center dimensioning	V
	G3 Circular movement with incremental center dimensioning	V
	G12 Circular movement with absolute center dimensioning	V
	G13 Circular movement with absolute center dimensioning	~

G commands for machining		
Feed rate and spindle speed	Gx26 Speed limitation	V
	G48 Reduce rapid traverse	Х
	G64 Interrupted feed	V
	G192 Feed per minute for rotary axis	-
	Gx93 Feed per tooth	V
	G94 Feed per minute	V
	Gx95 Feed per revolution	V
	Gx96 Constant surface speed	V
	Gx97 Speed	V
Tool-tip radius compensation	G40 Switch off TRC/MCRC	V
	G41 TRC/MCRC, left	V
	G42 TRC/MCRC, right	V
Zero point shifts	G51 Relative zero point shift	V
	G53 Parameter-dependent zero point shift	V
	G54 Parameter-dependent zero point shift	V
	G55 Parameter-dependent zero point shift	V
	G56 Additive zero point shift	V
	G59 Absolute zero point shift	V
	G121 Contour mirroring/shifting	V
	G152 Zero point shift, C axis	V
	G920 Deactivate zero point shift	V
	G921 Deactivate zero point shift, tool dimensions	V
	G980 Activate zero point shift	V
	G981 Activate zero point shift, tool dimensions	V
Oversizes	G50 Switch off oversize	V
	G52 Switch off oversize	V
	G57 Axis-parallel oversize	V
	G58 Contour-parallel oversize	V

ns
rar
0g
рг
Ζ
Δ
.
it∨
i
ţ
Da
Ē
Col
4
σ

G commands for machining		
Safety clearances	G47 Set safety clearances	~
	G147 Safety clearance (milling cycles)	V
Tools, types of compensation	T Tool call	V
	G148 Switching the tool edge compensation	~
	G149 Additive compensation	V
	G150 Compensation of right-hand tool tip	V
	G151 Compensation of left-hand tool tip	V
	G710 Adding tool dimensions	•
Cueles for turning		
Cycles for turning		
Simple turning cycles	G80 End of cycle	V
	G81 Simple longitudinal roughing	V
	G82 Simple face roughing	V
	G83 Simple contour repeat cycle	V
	G85 Undercut	V
	G86 Simple recessing cycle	~
	G87 Transition radii	~
	G88 Chamfers	~
Drilling cycles	G36 Tapping	~
	G71 Simple drilling cycle	~
	G72 Counterboring, countersinking, etc.	V
	G73 Tapping cycle	V
	G74 Deep hole drilling cycle	V



Cycles for turning		
Contour-based turning cycles	G810 Longitudinal roughing cycle	V
	G820 Face roughing cycle	v
	G830 Contour-parallel roughing cycle	V
	G835 Contour-parallel with neutral tool	V
	G860 Universal recessing cycle	V
	G866 Simple recessing cycle	V
	G869 Recess turning cycle	V
	G890 Finishing cycle	V
Thread cycles	G31 Thread cycle	V
	G32 Single thread cycle	V
	G33 Single thread cut (Thread single path)	V
	G933 Thread switch	_
	G799 Thread milling, axial	V
	G800 Thread milling in XY plane	V
	G806 Thread milling in YZ plane	V

Synchronization commands		
Assigning contour to operation	G98 Assignment of spindle to workpiece	-
	G99 Workpiece group	Х
Slide synchronization	G62 One-sided synchronization	*
	G63 Synchronous start of slides	*
	G162 Synchronization marking	*
Contour follow-up	G702 Storing/loading contour follow-up	V
	G703 Contour follow-up ON/OFF	V
	G706 K default branch	-

programs
7
Δ
_
.=
>
÷
5
Ť
Ö
Ĭ
F
0
Ú
-
7
6

Spindle synchronization, workpiece	G30 Converting and mirroring	V
transier	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	V
	G120 Reference diameter for lateral-surface machining	V
	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	V
	G112 Circular arc on lateral surface	V
	G113 Circular arc on lateral surface	V

Synchronization commands

1

Variable programming, program branches		
Programming with variables	# variables Evaluation during program conversion	V
	V variables Evaluation during program run	V
Program branches, program repeats	IFTHEN Program branching	V
	WHILE Program repeat	V
	SWITCH Program branching	V
Special functions	\$ Slide code	V
	/ Skip level	v
Data input and data output	INPUT Input (# variables)	v
	WINDOW Open output window (# variables)	V
	PRINT Output (# variables)	v
	INPUTA Input (V variables)	V
	WINDOWA Open output window (V variables)	V
	PRINTA Output (V variables)	V
Subprograms	L Subprogram call	V
Measuring functions, load monitoring		
In-process measuring	G910 Activate in-process measuring	V
	G912 Actual-value determination for in-process measuring	V
	G913 Deactivate in-process measuring	V
	G914 Deactivate touch probe monitoring	V
Post-process measuring	G915 Post-process measuring	•

G995 Specifying the monitoring zone

G996 Type of load monitoring

Load monitoring

i

v v

Other G codes Other G codes **G4** Dwell time V **G7** Precision stop on V **G8** Precision stop off V **G9** Precision stop (blockwise) V G15 Move rotary axes _ G60 Deactivate protection zone V V **G65** Display chucking equipment ٠ **G66** Component position ٠ G204 Waiting for time **G717** Updating nominal values _ G718 Move lag error _ **G901** Actual values in variables V G902 Zero point shift in variables V **G903** Lag error in variables V **G907** Zero point shift in variables ٠ **G908** Zero point shift in variables V G909 Zero point shift in variables V **G918** Zero point shift in variables _ **G919** Zero point shift in variables V V **G920** Zero point shift in variables **G921** Zero point shift in variables V G930 Zero point shift in variables V ٠ **G975** Zero point shift in variables **G980** Zero point shift in variables ~ **G981** Zero point shift in variables V **G940** Zero point shift in variables _ **G941** Zero point shift in variables _

Working planesG16 Tilting working plane✓G17 XY plane (front or rear face)✓G18 XZ plane (turning)✓G19 YZ plane (plan view / lateral surface)✓Tool positioning without machiningG0 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✓G842 Area milling, roughing✓G843 Centric polygon milling, roughing✓G844 Pocket milling, roughing✓G845 Pocket milling, roughing✓G846 Pocket milling, finishing✓G800 Thread milling in XY plane✓
G17 XY plane (front or rear face)✓G18 XZ plane (turning)✓G19 YZ plane (plan view / lateral surface)✓Tool positioning without machiningG0 Positioning at rapid traverse✓G14 Move to the tool change position✓G600 Preselect tool✓G701 Rapid traverse to machine coordinates✓G712 Define tool position✓G712 Define tool position✓G841 Area milling, roughing✓G843 Centric polygon milling, roughing✓G844 Centric polygon milling, finishing✓G845 Pocket milling, roughing✓G846 Pocket milling, finishing✓G846 Pocket milling, finishing✓
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 ✓ G841 Area milling, roughing ✓ G842 Area milling, finishing ✓ G844 Centric polygon milling, roughing ✓ G845 Pocket milling, finishing ✓ G846 Pocket milling, finishing ✓ G846 Pocket milling, finishing ✓ G840 Thread milling in XY plane ✓
G19 YZ plane (plan view / lateral surface)✓Tool positioning without machiningG0 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✓G841 Area milling, roughing✓G842 Area milling, finishing✓G844 Centric polygon milling, roughing✓G845 Pocket milling, roughing✓G846 Pocket milling, finishing✓G800 Thread milling in XY plane✓
Tool positioning without machiningG0 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 cyclesG841 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✓
G14 Move to the tool change position✓G600 Preselect tool✓G701 Rapid traverse to machine coordinates✓G714 Insert magazine tool◆G712 Define tool position◆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✓G840 Thread milling, finishing✓
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, roughing ✓ G845 Pocket milling, roughing ✓ G846 Pocket milling, finishing ✓ G846 Pocket milling, finishing ✓ G800 Thread milling in XY plane ✓
G701 Rapid traverse to machine coordinates✓G714 Insert magazine toolG712 Define tool positionMilling cyclesG841 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✓
G714 Insert magazine toolG712 Define tool positionMilling cyclesG841 Area milling, roughingG842 Area milling, finishingG843 Centric polygon milling, roughingG844 Centric polygon milling, roughingG845 Pocket milling, roughingG846 Pocket milling, finishingG846 Pocket milling, finishingG840 Thread milling in XY plane
G712 Define tool positionMilling cyclesG841 Area milling, roughingG842 Area milling, finishingG843 Centric polygon milling, roughingG844 Centric polygon milling, finishingG845 Pocket milling, roughingG846 Pocket milling, finishingG800 Thread milling in XY plane
Milling cyclesG841 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✓G840 Thread milling in XY plane✓
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✓
G843 Centric polygon milling, roughing✓G844 Centric polygon milling, finishing✓G845 Pocket milling, roughing✓G846 Pocket milling, finishing✓G800 Thread milling in XY plane✓
G844 Centric polygon milling, finishing✓G845 Pocket milling, roughing✓G846 Pocket milling, finishing✓G800 Thread milling in XY plane✓
G845 Pocket milling, roughing✓G846 Pocket milling, finishing✓G800 Thread milling in XY plane✓
G846 Pocket milling, finishing✓G800 Thread milling in XY plane✓
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 movementsG1 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





Overview of cycles

10.1 Workpiece blank cycles, single cut cycles

Workpiece k	blank cycles	Page
	Overview	155
	Standard blank	156
	ICP blank	157
Circula cost o		Demo
Single cut c		Page
_ _	Overview	158
	Rapid traverse positioning	159
	Move to the tool change position	160
	Linear machining, longitudinal Single longitudinal cut	161
	Linear machining, transverse Single transverse cut	162
	Linear machining at angle Single oblique cut	163
	Circular machining Single circular cut	165
	Circular machining Single circular cut	165
	Chamfer For machining a chamfer	167
	Rounding arc For machining a rounding	169
⇒ ^M	M function Entering an M function	171



i

10.2 Turning cycles

Turning cycles		Page
	Overview	172
	Cut longitudinal Roughing and finishing cycle for simple contours	175
	Cut transverse Roughing and finishing cycle for simple contours	176
	Cut with longitudinal plunge Roughing and finishing cycle for simple contours	187
	Cut with transverse plunge Roughing and finishing cycle for simple contours	189
	ICP contour-parallel, longitudinal Roughing and finishing cycle for any type of contour	202
Y	ICP contour-parallel, transverse Roughing and finishing cycle for any type of contour	204
	ICP cut longitudinal Roughing and finishing cycle for any type of contour	208
	ICP cut transverse Roughing and finishing cycle for any type of contour	210



10.3 Recessing and recess-turning cycles

Recessing cycles		Page
	Overview	218
	Recessing, radial Recessing and finishing cycles for simple contours	220
	Recessing, axial Recessing and finishing cycles for simple contours	222
	ICP recessing, radial Recessing and finishing cycles for any contour	236
	ICP recessing, axial Recessing and finishing cycles for any contour	238
	Undercut H	268
Lo	Undercut K	269
	Undercut U	270
	Parting Cycle for parting the workpiece	272

Recess-turning cycles		Page
	Overview	244
	Recess turning, radial Recess-turning and finishing cycles for simple contours	245
	Recess turning, axial Recess-turning and finishing cycles for simple contours	246
4	ICP recess turning, radial Recess-turning and finishing cycles for any type of contour	260
	ICP recess turning, axial Recess-turning and finishing cycles for any type of contour	262

10.4 Thread cycles

Thread cycles		Page
	Overview	276
[Thread cycle Longitudinal single or multi-start thread	280
imm,	Tapered thread Tapered single or multi-start thread	284
API .	API thread Single or multi-start API thread (API: American Petroleum Institute)	287
	Thread recutting Recut longitudinal single or multi-start thread	289
	Tapered thread, recutting Recut tapered single or multi-start thread	293
API	API thread, recutting Recut single or multi-start API thread	295
	Undercut DIN 76 Thread undercut and thread chamfer	297
DIN 509 E	Undercut DIN 509 E Undercut and cylinder chamfer	299
DIN 509 F	Undercut DIN 509 F Undercut and cylinder chamfer	301



10.5 Drilling cycles

Drilling cycles		Page
	Overview	305
	Axial drilling cycle For drilling single holes and patterns	306
	Radial drilling cycle For drilling single holes and patterns	308
	Axial deep-hole drilling cycle For drilling single holes and patterns	310
U	Radial deep-hole drilling cycle For drilling single holes and patterns	312
	Axial tapping cycle For drilling single holes and patterns	314
	Radial tapping cycle For drilling single holes and patterns	316
	Thread milling For milling threads in existing holes	318
10.6 Milling cycles

Milling cycles		Page
	Overview	322
	Rapid traverse positioning Activate C axis; position tool and spindle	323
9	Slot, axial For milling single slots or slot patterns	324
	Figure, axial For milling a single figure	326
	Axial ICP contour For milling single ICP contours or contour patterns	330
	Face milling For milling surfaces or polygons	333
	Slot, radial For milling single slots or slot patterns	336
	Figure, radial For milling a single figure	338
	Radial ICP contour For milling single ICP contours or contour patterns	342
e	Helical-slot milling, radial For milling a helical slot	345
3	Thread milling For milling threads in existing holes	318



10.6 Milling cycles

i

Α

Absolute coordinates ... 46 Additive compensation ... 128 Additive compensation for cycle programming ... 154 Alphanumeric keyboard ... 57 API thread ... 287 API thread, recutting ... 295 Automatic job ... 126 Axis designations ... 45 Axis values. setting ... 101, 102, 103, 104

В

Backup ... 583 Backup name ... 594 Bar/tube blank ... 156 Basic-block display Display during program execution ... 125 Block number Cycle programming ... 120

С

Caxis, fundamentals ... 38 Calculator ... 58 Chamfer ... 167 Character set ... 354 Circular drilling pattern, axial ... 360 Circular drilling pattern, radial ... 367 Circular machining ... 165 Circular milling pattern, axial ... 362 CNC PILOT 620 Ethernet interface ... 585 CNC PILOT 640 Ethernet interface ... 586 Comments Comment blocks in cycle programs ... 149 Comments in cycles ... 149 Compatibility in DIN programs ... 629 Compensation, additive ... 128 Compensations ... 127 Context-sensitive help ... 66 Continuous run Program execution ... 125 Contour follow-up in Teach-in mode ... 148 Contour group ... 484 Conversion into DIN format ... 143 Coordinate system ... 46 Coordinates, absolute ... 46

С

Coordinates, incremental ... 47 Coordinates, polar ... 47 Copy Circular ... 395 Linear ... 394 Mirror ... 395 Cut, ICP contour-parallel, longitudinal ... 202 Cut, ICP contour-parallel, longitudinal finishing ... 206 Cut. ICP contour-parallel. transverse ... 204 Cut, ICP contour-parallel, transverse finishing ... 207 Cut, ICP longitudinal ... 208 Cut, ICP longitudinal finishing ... 212 Cut, ICP transverse ... 210 Cut, ICP transverse finishing ... 213 Cut, longitudinal ... 175 Cut, longitudinal finishing ... 181 Cut, longitudinal finishing plunge ... 195 Cut, longitudinal finishing plungeexpanded ... 198 Cut, longitudinal finishingexpanded ... 183 Cut, longitudinal plunge ... 187 Cut, longitudinal plunging expanded ... 191 Cut, longitudinal-expanded ... 177 Cut, transverse ... 176 Cut, transverse finishing ... 182 Cut, transverse finishing plunge ... 196 Cut, transverse finishing plungeexpanded ... 200 Cut, transverse finishingexpanded ... 185 Cut, transverse plunge ... 189 Cut. transverse plungingexpanded ... 193 Cut, transverse-expanded ... 179 Cutting direction (cycle programming) ... 347, 348 Cutting limits SX, SZ ... 154 Cycle keys ... 148 Cycle menu ... 150

Cycle programming Cycle keys ... 148 Cycle programs, converting ... 603 Cycle starting point ... 146 Cycle status ... 88 Cycles, addresses used ... 154

D

Data backup ... 42 Data entry-fundamentals ... 56 Data transfer ... 583 DataPilot ... 73 Deep-hole drilling, axial ... 310 Deep-hole drilling, radial ... 312 Defining the tool change position ... 105 DIN cycle ... 374 DIN cycle (cycle programming) ... 374 DIN macros ... 147 DIN programs, converting ... 604 Direction of rotation (tool parameters) ... 522 Display operating times ... 110 Drilling and milling patterns, cycle programming ... 355 Drilling cycles, cycle programming ... 305 Drilling pattern linear, axial ... 356 Drilling, axial ... 306 Drilling, radial ... 308 Driven tool ... 523 Driven tools ... 96 Dry run mode ... 130 DXF contours ... 482

Ε

Editing the turret list ... 92 End point of ICP contour ... 387 Engraving, character set ... 354 Engraving, face ... 350 Engraving, lateral surface ... 352 Equidistant line (MCRC) ... 50 Equidistant line (TRC) ... 50 Error log ... 64 Error messages ... 62 Ethernet ... 584 Ethernet interface ... 584 Configuration ... 587 Connection options ... 586 Introduction ... 586 Example of drilling cycles ... 320 Example of milling cycle ... 349 Example of pattern machining ... 371 Example of thread and undercut cycles ... 303 Examples of recessing cycles ... 274 Examples of turning cycles ... 214 External access ... 583

Index

F

Feed angle ... 278 Feed rate ... 88 Feed rate reduction for drilling Cycle programming Deep-hole drilling ... 311, 313 Drilling cycle ... 307, 309 File organization ... 139 Fit calculation ... 389 Fits ... 389 Form elements (ICP) Fundamentals ... 379 Form elements, ICP ... 379 Full-surface machining Fundamentals ... 39

G

Geometry calculations (ICP) ... 380

Н

Handwheel operation ... 118 Handwheel resolution ... 144 Help files, downloading ... 71 Help graphics ... 147 Help system ... 66

I

ICP absolute or incremental dimensioning ... 389 ICP angle input ... 390 ICP basic elements, turning contour ... 406 ICP centric polygon in XY plane ... 464 ICP centric polygons in YZ plane ... 480 ICP chamfer in XY plane ... 454 ICP chamfer in YZ plane ... 471 ICP chamfer of turning contour ... 410 ICP chamfer on face ... 421 ICP chamfer on lateral surface ... 427 ICP circle in XY plane ... 455 ICP circle in YZ plane ... 472 ICP circle on face ... 432 ICP circular arc in XY plane ... 453 ICP circular arc in YZ plane ... 470, 471 ICP circular arc on face ... 420 ICP circular arc on lateral surface ... 426 ICP circular arc, turning contour ... 409 ICP circular on lateral surface ... 441 ICP circular pattern in XY plane ... 462 ICP circular pattern in YZ plane ... 478 ICP circular pattern on face ... 438 ICP circular pattern on lateral surface ... 448

I

ICP circular slot in XY plane ... 459 ICP circular slot in YZ plane ... 475 ICP circular slot on face ... 435 ICP circular slot on lateral surface ... 445 ICP contour direction ... 396 ICP contour element, deleting ... 398 ICP contour elements Face ... 416, 431 ICP contour elements on the lateral surface ... 422 ICP contour elements, adding ... 397 ICP contour elements, editing ... 399 ICP contour elements, face ... 416 ICP contour elements, turning contour ... 406 ICP contour form elements ... 410 ICP contour graphics ... 391 ICP contour programming ... 388 ICP contours with C-axis machining ... 428 ICP contours with Y-axis machining ... 428 ICP contours, editing ... 387, 397 ICP contours, fundamentals ... 378 ICP editor in cycle mode ... 381 ICP editor in smart.Turn ... 383 ICP editor submode in cycle mode ... 381 ICP editor submode in smart.Turn mode of operation ... 383 ICP face contours in smart.Turn ... 431 ICP form elements. superimposing ... 397 ICP geometry calculations ... 380 ICP hole in XY plane ... 460 ICP hole in YZ plane ... 476 ICP hole on face ... 436 ICP hole on lateral surface ... 446 ICP horizontal lines in XY plane ... 451 ICP horizontal lines in YZ plane ... 468 ICP horizontal lines of a turning contour ... 407 ICP horizontal lines on face ... 418 ICP horizontal lines on lateral surface ... 424 ICP last contour element, editing or deleting ... 398 ICP lateral surface contours in smart.Turn ... 439 ICP line at angle in XY plane ... 452 ICP line at angle in YZ plane ... 469

I

ICP line at angle on face ... 419 ICP line at angle on lateral surface ... 425 ICP line at angle on turning contour ... 408 ICP linear pattern in XY plane ... 461 ICP linear pattern in YZ plane ... 477 ICP linear pattern on face ... 437 ICP linear pattern on lateral surface ... 447 ICP linear slot in XY plane ... 458 ICP linear slot in YZ plane ... 474 ICP linear slot on face ... 435 ICP linear slot on lateral surface ... 444 ICP machining attributes ... 380 ICP nested contours and holes ... 429 ICP polar coordinates ... 390 ICP polygon in XY plane ... 457 ICP polygon in YZ plane ... 473 ICP polygon on face ... 434 ICP polygon on lateral surface ... 443 ICP programming Absolute or incremental dimensioning ... 389 Contour direction ... 396 Contour elements on face ... 416, 431 ICP recess turning, axial ... 262 ICP recess turning, axial finishing ... 266 ICP recess turning, radial ... 260 ICP recess turning, radial finishing ... 264 ICP recessing axial, finishing ... 242 ICP recessing cycles, axial ... 238 ICP recessing radial ... 236 ICP recessing radial, finishing ... 240 ICP rectangle in XY plane ... 456 ICP rectangle in YZ plane ... 472 ICP rectangle on face ... 433 ICP rectangle on lateral surface ... 442 ICP reference data ... 429 ICP reference data in XY plane ... 449 ICP reference data in YZ plane ... 465 ICP rounding arc in XY plane ... 454 ICP rounding arc of turning contour ... 410 ICP rounding arc on face ... 421 ICP rounding arc on lateral surface ... 427 ICP selection functions ... 393 ICP selection of solutions ... 392

I

ICP single surface in XY plane ... 463 ICP single surface in YZ plane ... 479 ICP starting point of contour in XY plane ... 450 ICP starting point of contour in YZ plane ... 467 ICP starting point of face contour ... 416 ICP starting point of lateral surface contour ... 422 ICP starting point of turning contour ... 406 ICP transitions between contour elements ... 389 ICP undercut DIN 509 E ... 412 ICP undercut DIN 509 F ... 413 ICP undercut DIN 76 ... 411 ICP undercut type H ... 415 ICP undercut type K ... 415 ICP undercut type U ... 414 ICP vertical lines in XY plane ... 450 ICP vertical lines in YZ plane ... 467 ICP vertical lines of a turning contour ... 407 ICP vertical lines on face ... 418 ICP vertical lines on lateral surface ... 424 ICP workpiece blank contour ... 157 ICP workpiece blank, "bar" ... 405 ICP workpiece blank, "cast part" ... 405 ICP workpiece blank, "tube" ... 405 ICP zoom ... 404 Inch, units of measure ... 48 Incremental coordinates ... 47 Input fields ... 56 Input window ... 53 Inside thread calculation ... 389 Invert ... 395

J

Job selection ... 126

Κ

Keystroke log ... 65

L

Last cut in thread cycles ... 279 Linear dimension ... 422 Linear drilling pattern, radial ... 364 Linear machining at angle ... 163 Linear machining, longitudinal ... 161 Linear machining, transverse ... 162

L

Linear milling pattern, axial ... 358 List operations ... 57 Load monitoring ... 131 Log, Error log ... 64 Log, keystroke log ... 65

Μ

M functions ... 171 M functions in cycles ... 149 Machine data display ... 84 Machine data, entering ... 80 Machine mode of operation ... 76 Machine setup ... 100 Machine with magazine ... 90 Machine with multifix ... 89 Machine with turret ... 90 Machine zero point ... 47 Machining attributes, ICP ... 380 Manual Operation ... 118 Marking (program transfer) ... 596 Menu selection ... 55 Metric, units of measure ... 48 Mid-program startup ... 124 Milling cutter radius compensation (MCRC) ... 50 Milling cycles, cycle programming ... 322 Milling direction for contour milling ... 347 Milling direction for pocket milling ... 348 Milling pattern Cycle programming Notes ... 355 Milling pattern circular, radial ... 369 Milling pattern linear, radial ... 366 Milling, axial figure ... 326 Milling, axial slot ... 324 Milling, face milling ... 333 Milling, ICP contour, axial ... 330 Milling, ICP contour, radial ... 342 Milling, radial figure ... 338 Milling, radial helical slot ... 345 Milling, radial slot ... 336 Mirroring Copying a contour section by mirroring ... 395 Monitoring EnDat encoders ... 77 Move to the tool change position ... 160 Multipoint tools, editing ... 513

Ν

NC programs, importing from predecessor controls ... 602, 606 Network connections ... 584 Network settings ... 587

0

Offset definition ... 102 Operating modes ... 40, 54 Operating modes, tool editor ... 506 Operation, the basics ... 54 Optical gauge ... 115 Organization ... 548 Organization mode of operation ... 548

Ρ

Parameters ... 549 Machining parameters ... 565 Parting ... 272 Parting tools ... 506 Pattern, circular drilling pattern, axial ... 360 Pattern, circular drilling pattern, radial ... 367 Pattern, circular milling pattern, axial ... 362 Pattern, circular milling pattern, radial ... 369 Pattern, linear drilling pattern, axial ... 356 Pattern, linear drilling pattern, radial ... 364 Pattern, linear milling pattern, axial ... 358 Pattern, linear milling pattern, radial ... 366 Pocket table setup ... 89 Polar coordinates ... 47 Position encoders ... 45 Positioning Spindle positioning in cycle mode ... 80 Program execution ... 125 Program information ... 139 Program run ... 122 Program run submode ... 122 Program selection ... 139 Program types ... 61 Programming station ... 73 Proportioning of cuts ... 278 Protection zone Display of protection zone status ... 104

Index

R

Rapid positioning milling ... 323 Rapid traverse positioning ... 159 Recess turning with ICP, axial ... 262 Recess turning with ICP, radial ... 260 Recess turning with ICP, radial finishing ... 264 Recess turning, axial ... 246 Recess turning, axial finishing ... 254 Recess turning, axial finishingexpanded ... 258 Recess turning, axial—expanded ... 250 Recess turning, ICP axial, finishing ... 266 Recess turning, radial ... 245 Recess turning, radial finishing ... 252 Recess turning, radial finishingexpanded ... 256 Recess turning, radialexpanded ... 248 Recess turning—fundamentals of cycle programming ... 244 Recessing axial, finishing ... 230 Recessing axial, finishingexpanded ... 234 Recessing cycles ... 218 Recessing cycles, contour forms ... 219 Recessing cycles, direction of cutting and infeed ... 218 Recessing cycles, undercut position ... 219 Recessing radial, finishing ... 228 Recessing radial, finishingexpanded ... 232 Recessing tools ... 506 Recessing, axial ... 222 Recessing, axial-expanded ... 226 Recessing, radial ... 220 Recessing, radial-expanded ... 224 Recess-turning tools ... 506 Recut (longitudinal) thread ... 289 Recut (longitudinal) thread expanded ... 291 Reference machining ... 133 Reference mark ... 45 Reference run ... 78, 103 Rounding arc ... 169

S

Safety clearance G47 ... 154 Safety clearances SCI and SCK ... 154 Screen ... 53 Service files, saving ... 65

S

Set the protection zone ... 104 Setting C-axis values ... 106 Setting up machine dimensions ... 108 Set-up clearance ... 172 Shift zero point ... 394 Simulation ... 138, 486 Contour generation during simulation ... 500, 501 Simulation window ... 490 Simulation with mid-program startup ... 497 Simulation, 3-D view ... 494 Simulation, material-removal graphic ... 493 Simulation, miscellaneous functions ... 489 Simulation, operating the ... 487 Simulation, setting up the views ... 490 Simulation, tool display ... 493 Simulation, traverse path ... 492 Simulation, zoom ... 496 Single block mode Program execution ... 125 Single cut cycles 158 Skip level ... 125 Slide position ... 37 smart.Turn dialogs ... 56 Soft ... 55 Soft keys ... 55 Sorting functions ... 139 Special compensation (recessing tools) ... 524, 525 Speed limitation Definition in cycle mode ... 80 Spindle ... 88 Spindle utilization ... 84 Starting point of ICP contour ... 387 Stopping angle (cycle mode) ... 80 Submode Transfer ... 583 Surface roughness Machining parameters ... 567 Switching functions for cycles ... 149 Switch-off ... 79 Switch-on ... 77 System time, setting ... 111

Т

Tapered thread ... 284 Tapered thread, recutting ... 293 Tapping, axial ... 314 Tapping, radial ... 316 Teach-in ... 120

Teach-In mode ... 120 Teach-in submode ... 120 Technical characteristics ... 620 Technology database ... 542 Technology editor ... 543 Technology editor submode ... 543 Thread Cycle programming API thread ... 287 Tapered thread ... 284 Thread cycle (longitudinal) ... 280 Thread cycle (longitudinal)expanded ... 282 Thread cycles ... 276 Thread depth ... 278 Thread milling, axial ... 318 Thread parameters ... 610 Thread pitch ... 611 Thread position, cycle programming ... 276 Thread run-in ... 278 Thread run-out ... 278 Time calculation (simulation) ... 499 TNCquide ... 66 TNCremo ... 583 Tool call ... 95 Tool carrier magazine ... 90 Tool carriers for multifix ... 89 Tool carriers, turret ... 90 Tool change position G14 ... 154 Tool compensation ... 116, 127, 153 Tool control graphics ... 511 Tool dimensions, fundamentals ... 49 Tool editor ... 508 Tool editor mode of operation ... 508 Tool life monitoring ... 97 Tool list ... 508 Tool list, comparing ... 123 Tool measurement ... 112 Tool measurement by touching the workpiece ... 113 Tool measurement with a touch probe ... 114 Tool position in turning cycles ... 173 Tool touch probe, calibrating ... 109 Tool types ... 506 Tool-life data, editing ... 515 Tools Driven tools ... 96 Tool compensation, entering ... 116, 153 Tool list ... 508, 509 Tool management ... 506

Т

Tools in different quadrants ... 96 Tools in different quadrants ... 96 Tools measurement with an optical gauge ... 115 Tool-tip radius compensation (TRC) ... 50 Touch probe ... 114 Touch-off ... 113 Transfer ... 583 Transformations Mirroring ... 403 Rotating ... 402 Shifting ... 402 Turning cycles ... 172 Turning cycles, example ... 214 Turret list, filling with the tool list ... 91

U

Undercut Parameters, undercut DIN 509 E, DIN 509 F ... 619 Parameters, undercut DIN 76 ... 617 Undercut cycles ... 276 Undercut DIN 509 E ... 299 Undercut DIN 509 F ... 301 Undercut DIN 76 ... 297 Undercut position, cycle programming ... 276 Undercutting type H ... 268 Undercutting type K ... 269 Undercutting type U ... 270 Units of measure ... 48 Unresolved contour elements (ICP) ... 380 USB interface ... 584

W

Wear compensation ... 506 Working with cycles ... 146 Workpiece blank contour, ICP ... 157 Workpiece blank cycles ... 155 Workpiece blank definition ICP ... 405 Workpiece zero point ... 48 Workpiece zero point, defining ... 101

Υ

Y axis, fundamentals ... 38

HEIDENHAIN

 DR. JOHANNES HEIDENHAIN GmbH

 Dr.-Johannes-Heidenhain-Straße 5

 83301 Traunreut, Germany

 [®] +49 8669 31-0

 ^E +49 8669 32-5061

 E-mail: info@heidenhain.de

 Technical support

 ^E +49 8669 31-3104

 E-mail: service.ms-support@heidenhain.de

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

