HEIDENHAIN

CNC PILOT 640
MANUALplus 620

User's Manual
smart.Turn and
ISO Programming

NC Software
548431-17
68894x-17

English (en)
10/2022

Controls and displays

Keys

If you are using a control with touch operation, you can replace some keystrokes with hand gestures.

Keys on visual display unit

Key	Function
	Switch the help graphics between outside and inside machining (only during cycle programming)
	No function
	Select the function in the display unit with the soft key selection key
	Shift between soft-key rows

Operating mode keys

Key	Function
	Select machine operating modes: ■ Machine ■ Teach-in ■ Program run ■ Reference

	Select programming operating modes:
	■ smart.Turn
	  ■ DIN PLUS – Unit mode
	  ■ ISO Mode
	■ Simulation
	■ AWG

	Select tool data and technology data: ■ Tool editor ■ Technology editor

	Select Organization operating mode:
	■ Machine parameters
	■ Transfer
	  ■ Project management
	  ■ Network connection
	■ Diagnostics

Numeric keypad

Key	Function
0 9	Number keys 0 to 9: ■ Enter numbers ■ Operate the menu
.	Insert a decimal point
−/+	Switch over between positive and negative values
ESC	Escape ■ Cancel the dialog ■ Navigate to the top of the menu
INS	Insert ■ Confirm a dialog ■ Create a new NC block in the editor
DEL	Delete Delete the selected range
	Backspace Erase the character to the left of the cursor
CE	Clear Entry Clear error messages in the Machine operating modes
>>	Enable dialog input fields for further input
ENT	Enter Confirm the entry

Navigation keys

Key	Function
	Move the cursor up or down
	Move the cursor to the left or to the right
PG UP PG DN	Page Up and Page Down Scroll back or forward one screen or dialog page
HOME END	Go to beginning of program/list or end of program/list

smart.Turn keys

Key	Function
	Go to the next form
	Go to next or previous group

Special keys

Key	Function
ERR	Error Open the error window
CALC	Calculator Start the integrated calculator
i	Information ■ Show additional information in the parameter editor ■ Call TURNguide
GOTO	Go to ■ Select alternative input ■ Activate the alphabetic keyboard
PRT SC	Print Screen Create a screenshot
	DIADUR ■ Use the function in connection with the Remote Desktop Manager ■ Open the HEROS menu

Machine operating panel

Key	Function
	Start or stop machining
	Stop the feed rate
	Stop the spindle
	Switch on the spindle
	Spindle **jog** The spindle rotates as long as you press the key.
	Move the axes (e.g., in +X or +Y direction)
	Change the spindle (machine-dependent)
	Change the slide (machine-dependent)

Operating panel of the control

TE 745T with handwheel

TE 725T FS

TE 361T

Contents

1	Fundamentals	31
2	First steps	39
3	NC Programming	61
4	smart.Turn Units (Option 9)	97
5	smart.Turn Units for the Y Axis (Option 9 and Option 70)	235
6	DIN/ISO Programming	273
7	Touch Probe Cycles	577
8	ISO Programming for the Y Axis (Option 70)	627
9	TURN PLUS (Option 63)	681
10	B Axis (Option 54)	721
11	Overview of Units (Option 9)	729
12	Overview of G Codes	741

1 Fundamentals.. 31

1.1 About this manual.. 32

1.2 Software and features.. 34

Software options.. 35

New functions of NC software 68894x-17.. 37

2 First steps.......... 39

2.1 Overview.......... 40

2.2 Switching on the machine.......... 41

2.3 Programming the first workpiece.......... 42

Selecting an operating mode.......... 42
Important controls and displays.......... 42
Creating a new NC program.......... 42
Setting up the turret list.......... 44
Programming a contour in ICP submode (option 8 or 9).......... 45
Programming the machining process in smart.Turn (option 9).......... 47
Closing an NC program.......... 49
Programming a contour in ISO Mode.......... 50
Programming the machining process in ISO Mode.......... 52
TURN PLUS programming (option 63).......... 58

2.4 NC program verification in the simulation.......... 59

3 NC Programming.......... 61

3.1 smart.Turn and DIN/ISO programming.......... 62

Contour follow-up.......... 62
Structured NC program.......... 63
Linear and rotary axes.......... 65
Units of measure.......... 65
Elements of an NC program.......... 66
Creating a new NC program.......... 67

3.2 Fundamentals: smart.Turn editor.......... 68

Menu structure.......... 68
Parallel editing.......... 69
Screen layout.......... 69
Selecting the editor functions.......... 70
Editing with active tree view.......... 71
Shared menu items.......... 72

3.3 Program section code.......... 80

PROGRAM HEAD section.......... 82
CHUCKING EQUIPMENT section.......... 84
TURRET / MAGAZINE section.......... 85
MANUAL TOOL section.......... 85
Contour group section.......... 85
BLANK section.......... 86
FINISHED PART section.......... 86
AUX. BLANK section.......... 86
AUXILIARY CONTOUR section.......... 86
FRONT, REAR SIDE section.......... 86
LATERAL section.......... 86
FRONT Y / REAR SIDE Y section.......... 86
SURFACE Y section.......... 87
MACHINING section.......... 88
END code.......... 88
SUBROUTINE section.......... 88
RETURN code.......... 88
CONST code.......... 89
VAR code.......... 89
SLIDE ALLOCATION code.......... 90

3.4 Tool programming.......... 91

Setting up a turret list.......... 91
Editing tool entries.......... 93
Multipoint tools.......... 93
Replacement tools.......... 94

3.5 Automatic job.......... 95

4 smart.Turn Units (Option 9)..... 97

4.1 Units – smart.Turn units..... 98

Units menu item..... 98
smart.Turn unit..... 98

4.2 Units – Roughing..... 106

Unit G810 Longitudinal roughing in ICP..... 106
G820 Roughing in ICP unit..... 108
Unit G830 Contr. parallel in ICP..... 110
Unit G835 Bidirectional in ICP..... 111
Unit G810 Roughing, direct longitudnl..... 112
Unit G820 Roughing, direct transverse..... 114
Unit G895 Simultaneous roughing (option 54)..... 115

4.3 Units – Recessg..... 119

Unit G860 Contour recess in ICP..... 119
Unit G869 Recess turning in ICP..... 120
Unit G860 Contour recess, direct..... 122
Unit G869 Recess turning, direct..... 123
Unit G859 Parting..... 124
Unit G85X Undercutting (H,K,U)..... 125
Unit G870 ICP Recessing – Recess cycle..... 126

4.4 Units – Drilling / Centric..... 127

Unit G74 Centric drilling..... 127
Unit G73 Centric tapping..... 128
Unit G72 Boring, cntrsnkg..... 130

4.5 Units – Drillg / Face C, Lateral C, and ICP C..... 131

Unit G74 Single hole, front face C..... 131
Unit G74 Drilling lin. pattern, front face C..... 133
Unit G74 Drilling circ. pattern, front face C..... 135
Unit G73 Tapping, front face C..... 137
Unit G73 Tapping linear pattern, front face C..... 138
Unit G73 Tapping circular pattern, front face C..... 139
Unit G74 Single hole, lat. surface C..... 141
Unit G74 Drilling lin. patt., lat. surface C..... 143
Unit G74 Drilling circ. patt., lat. surface C..... 145
Unit G73 Tap hole, lateral surface C..... 147
Unit G73 Tapping linear pattern, cylindrical C..... 148
Unit G73 Tapping circ. pattern, lateral surf. C..... 149
Unit G74 Drilling in ICP C (option 55)..... 151
Unit G73 Tapping in ICP C (option 55)..... 152
Unit G72 Boring, cntrsinkg ICP C (option 55)..... 153
Units G75 Bore milling ICP C (option 55)..... 154

4.6	**Units – Drillg / Predrilling, milling in C (option 55)**	**158**
	Unit G840 Predrill, contour mill, figure on face in C	158
	Unit G845 Predrill, pocket mill, figure on face in C	160
	Unit G840 Predrill, contour mill, ICP on face in C	162
	Unit G845 Predrill, pocket mill, ICP on face in C	163
	Unit G840 Predrill, contour mill, fig. on lateral in C	164
	Unit G845 Predrill, pocket mill, fig. on lateral in C	166
	Unit G840 Predrill, contour mill, ICP on lateral in C	168
	Unit G845 Predrill, pocket mill, ICP on lateral in C	169
4.7	**Units – Finishing**	**171**
	Unit G890 Contouring in ICP	171
	Unit G890 Contouring, direct longitdnl	173
	Unit G890 Contouring, direct transverse	175
	Unit G890 Relief, type E,F,DIN76 – Recess	177
	Unit G809 Measuring cut	179
	Unit G891 Simultaneous finishing (option 54)	180
4.8	**Units – Thread**	**184**
	Overview of thread units	184
	Handwheel superimposition (option 11)	184
	Parameter V: Type of infeed	185
	Unit G32 Thread, direct	186
	Unit G31 Thread, ICP	187
	Unit G352 API thread	189
	Unit G32 Tapered thread	190
4.9	**Units – Millg / C axis, face, C axis, ICP face (option 55)**	**192**
	Unit G791 Linear slot, front face	192
	Unit G791 Slot in linear pattern, front face	193
	Unit G791 Slot in circ. pattern, front face	194
	Unit G797 Face milling C	195
	Unit G799 Thread milling, front face C	196
	Unit G840 Contour mllng, figures front face C	197
	Unit G84X Pocket mllng, figures front face C	199
	Unit G801 Engraving in C axis on face	201
	Unit G840 ICP contour milling, front face C	202
	Unit G845 ICP pocket milling, front face C	204
	Unit G840 ICP deburring, front face C	206
	Unit G797 face milling ICP	207
	Unit G847 ICP Troch. contour mllng, front face C	208
	Unit G848 ICP Troch. pocket mllng, front face C	210
4.10	**Units – Milling / C axis, lateral, C axis, ICP latrl. (option 55)**	**212**
	Unit G792 Linear slot, lat. surface	212
	Unit G792 Slot in lin. pattern, lat. surface	213

Unit G792 Slot in circ. pattern, lat. surface ... 214
Unit G798 Helical slot milling ... 215
Unit G840 Contour mllng, figures lat. surf. C ... 216
Unit G84X Pocket mllng, figures lat. surface C ... 218
Unit G802 Engraving in C on latrl. surface ... 220
Unit G840 ICP contour milling, lat. surface C ... 221
Unit G845 ICP pocket milling, lat. surface C ... 222
Unit G840 ICP deburring, lat. surface C ... 223
Unit G847 ICP Troch. contour mllng, lat. surf. C ... 224
Unit G848 ICP Troch. pocket mllng, lat. surf. C ... 226

4.11 Units – Spec (special operations) ... 228

Unit Program beginning (START) ... 228
Unit C axis ON (option 9) ... 229
Unit C axis OFF (option 9) ... 230
Unit Subprogram call ... 230
Unit Process logic / Repetition – program section repeat ... 231
Unit Program end (END) ... 232
Unit Tilt plane ... 233

5 smart.Turn Units for the Y Axis (Option 9 and Option 70)........ 235

5.1 Units – Drilling / ICP Y........ 236

Unit G74 Drilling in ICP Y........ 236
Unit G73 Tapping in ICP Y........ 237
Unit G72 Boring, cntrsinkg ICP Y........ 238
Units G75 Bore milling Y........ 239

5.2 Units – Drilling / Predrilling, milling in Y........ 243

Unit G840 Predrill, contour mill, ICP on face in Y........ 243
Unit G845 Predrill, pocket mill, ICP on face in Y........ 244
Unit G840 Predrill, contour mill, ICP on lateral in Y........ 246
Unit G845 Predrill, pocket mill, ICP on lateral in Y........ 247

5.3 Units – Millg / Y axis, face, Y axis, latrl........ 249

Unit G840 ICP contour milling on face in Y........ 249
Unit G845 ICP pocket milling on face in Y........ 250
Unit G840 ICP deburring on front face in Y........ 252
Unit G841 Single surface in Y axis on face........ 253
Unit G843 Centric polygon, Y axis on face........ 254
Unit G803 Engraving in Y axis on face........ 255
Unit G800 Thread milling on face in Y........ 256
Unit G847 ICP Troch. contour mllng, front face Y........ 257
Unit G848 ICP Troch. pocket mllng, front face Y........ 259
Unit G840 ICP contour mill, latrl. surf. Y........ 261
Unit G845 ICP pocket mill, lateral surf. Y........ 262
Unit G840 ICP deburring on lateral surf. Y........ 263
Unit G841 Single surface in Y axis, latrl........ 264
Unit G843 Centric polygon Y axis, latrl........ 265
Unit G804 Engraving in Y on latrl. surface........ 266
Unit G806 Thread milling, lateral in Y........ 267
Unit G847 ICP Troch. contour mllng, lat. surf. Y........ 268
Unit G848 ICP Troch. pocket mllng, lat. surf. Y........ 270

6 DIN/ISO Programming.. 273

6.1 Programming in ISO Mode.. 274

Geometry and machining commands.. 274
Contour programming.. 276
NC blocks of the DIN/ISO program.. 278
Creating, editing and deleting NC blocks.. 279
Address parameters.. 280
Machining cycles.. 282
Subprograms and expert programs.. 283
NC program interpretation.. 283
ISO programs of predecessor controls.. 284
Geometry menu item.. 286
Machining menu item.. 286

6.2 Definition of workpiece blank.. 287

Chuck part bar or tube G20-Geo.. 287
Casting G21-Geo.. 287

6.3 Basic elements of turning contour.. 288

Starting point of turning contour G0–Geo.. 288
Machining attributes for form elements.. 288
Line segment in a turning contour G1–Geo.. 289
Circular arc of turning contour G2-/G3-Geo.. 290
Circular arc of turning contour G12-/G13-Geo.. 291

6.4 Form elements of turning contour.. 293

Recess (standard) G22–Geo.. 293
Recess (general) G23–Geo.. 295
Thread with undercut G24–Geo.. 296
Undercut G25–Geo.. 297
Thread (standard) G34–Geo.. 300
Thread (general) G37–Geo.. 301
Bore hole (centric) G49–Geo.. 303

6.5 Attributes for contour description.. 304

Surface roughness G10–Geo.. 304
Feed reduction G38-Geo.. 305
Attributes for superimposed elements G39-Geo.. 305
Separation point G44.. 306
Oversize G52-Geo.. 306
Feed per revolution G95-Geo.. 307
Additive correction G149-Geo.. 308

6.6 C-axis contours—fundamentals.. 309

Position of milling contours.. 309
Circular pattern with circular slots.. 312

6.7	**Front/Rear face contours**	**315**
	Starting point of front/rear face contour G100-Geo	315
	Line segment in front/rear face contour G101-Geo	315
	Circular arc in front/rear face contour G102-/G103-Geo	316
	Bore hole on front/rear face G300-Geo	317
	Linear slot on front/rear face G301-Geo	317
	Circular slot on front/rear face G302-/G303-Geo	318
	Full circle on front/rear face G304-Geo	318
	Rectangle on front/rear face G305-Geo	319
	Text for front face CG306-Geo	319
	Polygon on front/rear face G307-Geo	320
	Linear pattern on front/rear face G401-Geo	320
	Circular pattern on front/rear face G402-Geo	321
	DataMatrix face C G405-Geo	322
6.8	**Lateral surface contours**	**323**
	Starting point of lateral surface contour G110-Geo	323
	Line segment in a lateral surface contour G111-Geo	323
	Circular arc in lateral surface contour G112-/G113-Geo	324
	Hole on lateral surface G310-Geo	325
	Linear slot on lateral surface G311-Geo	325
	Circular slot on lateral surface G312-/G313-Geo	326
	Full circle on lateral surface G314-Geo	326
	Rectangle, surface G315-Geo	327
	Text for lateral face C G316-Geo	327
	Polygon on lateral surface G317-Geo	328
	Linear pattern on lateral surface G411-Geo	328
	Circular pattern on lateral surface G412-Geo	329
	DataMatrix for lateral surface G415-Geo	330
6.9	**Tool positioning**	**331**
	Rapid traverse G0	331
	Rapid traverse in machine coordinates G701	331
	Tool change point G14	332
	Defining the Tool change point G140	332
6.10	**Linear and circular movements**	**333**
	Linear movement G1	333
	Circular arc ccw G2/G3	334
	Circular arc ccw G12/G13	335
6.11	**Feed rate, shaft speed**	**336**
	Speed limitation G26	336
	Reduce rapid traverse G48	336
	Interrupted feed G64	337
	Feed per tooth Gx93	338

Constant feed G94 (feed per minute) 338
Feed per revolution Gx95 339
Constant cutting speed Gx96 339
Spindle speed Gx97 340

6.12 Tool-tip and cutter radius compensation 341

Fundamentals 341
Switch off TRC/MCRC G40 341
Switch on TRC/MCRC G41/G42 342

6.13 Datum shifts 343

Datum shift G51 344
Datum shifts – G53/G54/G55 345
Additive datum shift G56 345
Absolute datum shift G59 346

6.14 Oversizes 347

Switching off oversize G50 347
Axis-parallel oversize G57 347
Contour-parallel oversize (equidistant) G58 348

6.15 Safety clearance 349

Safety clearance G47 349
Safety clearance G147 349

6.16 Tools, compensations 350

Inserting tools – T 350
Correction of cut G148 (changing the cutter compensation) 351
Additive correction G149 352
Compensation of tool tip G150/G151 353

6.17 Contour-based turning cycles 354

Working with contour-based cycles 354
Longitud. roughing G810 356
Face roughing G820 359
Contour-parallel roughing G830 362
Contour cycle, bidirectional G835 (contour-parallel with neutral tool) 364
Recessing G860 366
Repeating recessing cycle G740 368
Repeating recessing cycle G741 368
Recess turning cycle G869 370
Recessing cycle G870 373
Contour finishing G890 374
Simultaneous roughing G895 (option 54) 378
Simultaneous finishing G891 (option 54) 384
Measuring path G809 389

6.18	**Contour definitions in the machining section**	**390**
	Cycle end / simple contour G80	390
	Linear slot on front/rear face G301	391
	Circular slot on front/rear face G302/G303	391
	Full circle on front/rear face G304	392
	Rectangle on front/rear face G305	392
	Polygon on front/rear face G307	393
	Linear slot on lateral surface G311	393
	Circular slot on lateral surface G312/G313	394
	Full circle on lateral surface G314	394
	Rectangle, surface G315	395
	Eccentric polygon on lateral surface G317	395
6.19	**Thread cycles**	**396**
	Overview of thread cycles	396
	Handwheel superimposition	396
	Parameter V: Type of infeed	397
	Universal tapping cycle G31	398
	Single thread cycle G32	403
	Thread single path G33	405
	Metric ISO thread G35	407
	Tapered API thread G352	408
	Contour thread G38	410
6.20	**Parting cycle**	**413**
	Cut-off cycle G859	413
6.21	**Undercut cycles**	**414**
	Undercut cycle G85	414
	Undercut DIN 509 E with cylinder machining G851	415
	Undercut DIN 509 F with cylinder machining G852	416
	Undercut DIN 76 with cylinder machining G853	417
	Undercut type U G856	419
	Undercut type H G857	420
	Undercut type K G858	420
6.22	**Drilling cycles**	**421**
	Overview of drilling and boring cycles and contour reference	421
	Tapping G36 – Single path	422
	Simple drilling G71	423
	Boring/cnt-sink G72	425
	Tapping G73	426
	Deep boring G74	428
	Bore milling G75	431
	Linear pattern on front face G743	433
	Circular pattern on front face G745	434

	Linear pattern on lateral surface G744	436
	Circular pattern on lateral surface G746	437
	Thread milling axial G799	439
6.23	**C-axis commands**	**440**
	Reference diameter G120	440
	Datum shift in C axis G152	440
	Standardize C axis G153	441
	Short path in C G154	441
6.24	**Front and rear face machining**	**442**
	Rapid traverse on front/rear face G100	442
	Linear segment on front/rear face G101	443
	Circular arc on front/rear face G102/G103	445
6.25	**Lateral surface machining**	**447**
	Rapid traverse on lateral surface G110	447
	Surface linear G111	447
	Circular arc on lateral surface G112/G113	449
6.26	**Milling cycles**	**451**
	Overview of milling cycles	451
	Linear slot, front face G791	453
	Linear slot, lat. surface G792	454
	Contour and figure milling cycle on front face G793	455
	Contour and figure milling cycle on lateral surface G794	457
	Area milling on front face G797	460
	Helical slot milling G798	462
	Mill. contour G840	463
	Pocket milling - roughing G845	472
	Pocket milling - finishing G846	478
	Trochoidal contour milling G847	480
	Trochoidal pocket milling G847	482
6.27	**Engraving cycles**	**484**
	Character sets	484
	Engraving on front face G801	487
	Engraving on lateral surface G802	488
6.28	**Contour follow-up**	**489**
	Saving/loading Contour follow-up G702	489
	Contour follow-up on/off G703	489
6.29	**Other G codes**	**490**
	Clamping G65	490
	Blank contour G67 (for graphics)	490
	Period of dwell G4	490

Precision stop ON G7 490
Precision stop OFF G8 491
Precision stop blockwise G9 491
Switch off protection zone G60 491
Actual values in variable G901 491
Datum in variable G902 491
Lag error in variable G903 492
Fill variable memory G904 492
Feed rate override 100 % G908 492
Interpreter stop G909 492
Spindle override 100% G919 493
Deactivating datum shifts G920 493
Deactivating datum shift, tool lengths G921 493
Fluctuating spindle speed G924 493
Convert lengths G927 494
TCPM G928 494
Look-ahead parameter G932 495
Calculating variables automatically G940 496
Information to DNC G941 498
Misalignment compensation G976 498
Lift off after NC stop – LIFTOFF G977 499
Activating datum shift G980 500
Activating datum shifts, tool lengths G981 500
Monitoring zone G995 500
Load monitoring G996 501
Activating direct program-run continuation G999 501
Force reduction G925 502
Sleeve monitoring G930 503
HDT mode G931 504
Eccentric turning G725 506
Eccentric transition G726 507
Non-circular X G727 509
Compens. for helical teeth G728 510

6.30 Measuring the machine condition (option 155) 511

Measuring the machine condition – fingerprint G238 511
Component monitoring G939 513

6.31 Programming variables 514

Fundamentals 514
Variable types 515
Reading tool data 520
Reading diagnostic bits 523
Reading the current NC information 524
Reading general NC information 525
Reading configuration data – PARA 528

	Determining the index of a parameter element – PARA	529
	Expanded variable syntax CONST – VAR	529
6.32	**Data input and data output**	**533**
	Output window for variables – WINDOW	533
	File output for variables – WINDOW	533
	Input of variables – INPUT	534
	Output of # variables – PRINT	534
6.33	**Conditional block run**	**535**
	Program branch IF..THEN..ELSE..ENDIF	535
	Interrogating variables and constants	537
	Program repeat WHILE..ENDWHILE	538
	Program branch SWITCH..CASE	540
	Skip level	541
6.34	**Subprograms**	**542**
	Subprogram call L "xx" V1	542
	Dialogs in subprogram calls	543
	Help graphics for subprogram calls	544
6.35	**M commands**	**545**
	M commands for program-run control	545
	Machine commands	546
6.36	**Assignment, synchronization, workpiece transfer**	**547**
	Converting and mirroring G30	547
	Transformations of contours G99	548
	Setting a synchronizing mark G162	548
	One-sided synchronization G62	549
	Synchronous start of slides G63	550
	Synchronous function M97	550
	Spindle synchronization G720	551
	Offset C angle G905	552
	Traverse to a fixed stop G916	553
	Controlled parting using lag error monitoring G917	554
6.37	**G codes from previous controls**	**555**
	Fundamentals	555
	Undercut G25 – contour definitions in the machining section	555
	Simple longitudinal roughing G81 – simple turning cycles	557
	Simple face roughing G82 – simple turning cycles	558
	Simple contour repeat cycle G83 – simple turning cycles	559
	Recessing G86 – simple turning cycle	561
	Radius cycle G87 – simple turning cycles	562
	Chamfer cycle G88 – simple turning cycles	562

Simple longitudinal single-start thread G350 – 4110 563
Simple longitudinal multi-start thread G351 – 4110 564

6.38 DIN PLUS program example 565

Example of a subprogram with contour repetitions 565

6.39 Connection between geometry and machining commands 567

Turning 567
C-axis machining – front/rear face 568
C-axis machining – lateral surface 568

6.40 Full-surface machining 569

Fundamentals of full-surface machining 569
Programming of full-surface machining 570
Full-surface machining with counter spindle 571
Full-surface machining with single spindle 573

6.41 Program templates 575

Fundamentals 575
Opening a program template 575

7 Touch Probe Cycles... 577

7.1 General information on touch probe cycles (option 17).. 578

Fundamentals.. 578
Principle of function of touch probe cycles.. 578
Touch probe cycles for automatic mode.. 579

7.2 Touch probe cycles for single-point measurement... 581

Single-point meas. for tool comp. G770.. 581
Single-point measurmnt., datum G771.. 583
Datum, single point w/ C axis G772... 585
Datum, C-axis object center G773.. 587

7.3 Touch probe cycles for two-point measurement.. 589

Two-point meas. G18 transverse G775.. 589
Two-pt. meas. G18 longitudinal G776... 591
Two-point measurement G17 G777.. 593
Two-point measurement G19 G778.. 595

7.4 Calibrating touch probes.. 597

Calibrate touch probe standard G747... 597
Calibrate touch probe via two points G748.. 599

7.5 Measuring with touch probe cycles.. 601

Paraxial probing G764.. 601
Probing in C axis G765... 603
Probing w/ 2 axes in ZX plane G766.. 604
Probing w/ 2 axes in ZX plane G767.. 605
Probing w/ 2 axes in ZY plane G768.. 606
Probing w/ 2 axes in XY plane G769.. 607

7.6 Search cycles.. 608

Find hole in C face G780... 608
Find hole C lateral surface G781... 610
Find stud in C face G782... 612
Find stud C lateral surface G783.. 614

7.7 Circular measurement... 616

Circular measurmnt G785.. 616
Calc. of pitch circle G786... 618

7.8 Angular measurement... 620

Angular measurmnt G787... 620
Misalignment compensation after angle measurement G788.. 622

7.9 In-process measrmnt... 623

Workpiece measurement.. 623

Switch on measurement G910 623
Activating measuring path monitoring G911 624
Actual-value determination G912 624
End measuring G913 624
Deactivating measuring path monitoring G914 624
Example: Measuring and compensating workpieces 625

8	**ISO Programming for the Y Axis (Option 70)**	**627**
8.1	**Y-axis contours – fundamentals**	**628**
	Position of milling contours	628
	Cutting limit	628
8.2	**Contours in the XY plane**	**629**
	Starting point of contour in XY plane G170-Geo	629
	Line segment in XY plane G171-Geo	629
	Circular arc in XY plane G172-/G173-Geo	630
	Hole in XY plane G370-Geo	631
	Linear slot in XY plane G371-Geo	632
	Circular slot in XY plane G372/G373-Geo	632
	Full circle in XY plane G374-Geo	633
	Rectangle in XY plane G375-Geo	633
	Single surface in XY plane G376-Geo	634
	Polygon in XY plane G377-Geo	634
	Text for front face Y G378-Geo	635
	Linear pattern in XY plane G471-Geo	635
	Circular pattern in XY plane G472-Geo	636
	DataMatrix pattern in XY plane G475-Geo	637
	Centric polygon in XY plane G477-Geo	637
8.3	**Contours in the YZ plane**	**638**
	Starting point of contour in YZ plane G180-Geo	638
	Line segment in YZ plane G181-Geo	638
	Circular arc in YZ plane G182/G183-Geo	639
	Hole in YZ plane G380-Geo	640
	Linear slot in YZ plane G381-Geo	640
	Circular slot in YZ plane G382/G383-Geo	641
	Full circle in YZ plane G384-Geo	641
	Rectangle in YZ plane G385-Geo	642
	Polygon in YZ plane G387-Geo	642
	Text for lateral surface Y G388-Geo	643
	Linear pattern in YZ plane G481-Geo	643
	Circular pattern in YZ plane G482-Geo	644
	DataMatrix pattern in YZ plane G485-Geo	645
	Single surface in YZ plane G386-Geo	645
	Centric polygon in YZ plane G487-Geo	646
8.4	**Working planes**	**647**
	Y-axis machining	647
	Tilt working plane G16	648
	Tilt working plane G16	649
8.5	**Tool positioning in Y axis**	**650**
	Rapid traverse G0	650

	Approach the Tool change point G14	650
	Rapid traverse in machine coordinates G701	651
8.6	**Linear and circular movements in the Y axis**	**652**
	Milling: Linear movement G1	652
	Milling: Circular arc cw G2, G3 – incremental center coordinates	653
	Milling: Circular arc cw G12, G13 – center coordinates	654
8.7	**Milling cycles for the Y axis**	**655**
	Area milling - roughing G841	655
	Area milling - finishing G842	656
	Centric polygon - roughing G843	657
	Centric polygon - finishing G844	658
	Pocket milling - roughing G845 (Y axis)	659
	Pocket milling - finishing G846 (Y axis)	664
	Engraving in XY plane G803	666
	Engraving in YZ plane G804	667
	Thread milling in XY plane G800	668
	Thread milling in YZ plane G806	669
	Skiving G807	669
	Hobbing G808	674
8.8	**Example program**	**675**
	Machining with the Y axis	675

9	**TURN PLUS (Option 63)**	**681**
9.1	**The function TURN PLUS**	**682**
	TURN PLUS concept	682
9.2	**Automatic working plan generation (AWG) submode**	**683**
	Generating a working plan	684
	Machining sequence – fundamentals	685
	Editing and managing the Machining sequence	686
	Overview of machining sequences	688
9.3	**AWG control graphic**	**703**
	Setting the AWG control graphic	703
9.4	**Machining information**	**704**
	Tool selection and turret assignment list	704
	Hollowing (internal profiling)	705
	Contour recessing, Recessing	706
	Drillg	706
	Cutting data and coolant	707
	Inside contours	707
	Shaft machining	709
9.5	**Example**	**711**
9.6	**Full-surface machining with TURN PLUS**	**716**
	Rechucking the workpiece	716
	Defining the chucking equipment for full-surface machining	717
	Automatic working plan generation for full-surface machining	719
	Rechucking the workpiece in the main spindle	719
	Transferring the workpiece from the main spindle to the counter spindle	720
	Parting and picking-off the workpiece with the counter spindle	720

10 B Axis (Option 54)...... 721

10.1 Fundamentals...... 722

High Dynamic Turning...... 724

10.2 B axis compensation...... 725

10.3 Simulation...... 726

11 Overview of Units (Option 9).. 729

11.1 Units – "Turning" group.. 730

11.2 Units – "Drilling" group.. 732

11.3 Units – "Predrilling in C axis" group.. 734

11.4 Units – "Milling in C axis" group... 735

11.5 Units – "Drilling, predrilling in Y axis" group.. 737

11.6 Units – "Milling in Y axis" group... 738

11.7 Units – "Special units" group.. 739

12 Overview of G Codes.. 741

12.1 Section codes.. 742

12.2 Overview of G commands in the CONTOUR section... 743

12.3 Overview of G commands in the MACHINING section.. 746

1

Fundamentals

1.1 About this manual

Safety precautions

Comply with all safety precautions indicated in this document and in your machine manufacturer's documentation!

Precautionary statements warn of hazards in handling software and devices and provide information on their prevention. They are classified by hazard severity and divided into the following groups:

DANGER

Danger indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **will result in death or severe injury.**

WARNING

Warning indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **could result in death or serious injury**.

CAUTION

Caution indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **could result in minor or moderate injury.**

NOTICE

Notice indicates danger to material or data. If you do not follow the avoidance instructions, the hazard **could result in property damage**.

Sequence of information in precautionary statements

All precautionary statements contain the following four sections:

- Signal word indicating the hazard severity
- Type and source of hazard
- Consequences of ignoring the hazard for example: "There is danger of collision during subsequent machining operations"
- Escape – hazard prevention measures

Informational notes

Observe the informational notes provided in these instructions to ensure reliable and efficient operation of the software.
In these instructions, you will find the following informational notes:

i	The information symbol indicates a **tip**. A tip provides important additional or supplementary information.

⚙	This symbol prompts you to follow the safety precautions of your machine manufacturer. This symbol also indicates machine-dependent functions. Possible hazards for the operator and the machine are described in the machine manual.

📖	The book symbol indicates a **cross reference**. A cross reference leads to external documentation for example the documentation of your machine manufacturer or other supplier.

Have you found any errors or would you like to suggest changes?

We continuously strive to improve our documentation for you.
Please help us by sending your suggestions to the following e-mail address:

tnc-userdoc@heidenhain.de

1.2 Software and features

This manual describes functions that are available in the control with NC software 688946-17 and 688947-17.

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

The machine manufacturer adapts the usable features of the control to his machine by setting the machine parameters. Some of the functions described in this manual may not be among the features possible on your machine tool.

Control functions that may not be available on your machine include:

- Positioning of spindle (**M19**) and driven tool
- Machining with the C or Y axis
- Machining with the B axis
- Machining with multiple slides

In order to find out about the specific support for your machine, please contact the machine manufacturer.

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 users of the control.

HEIDENHAIN also offers the DataPilot MP 620 or DataPilot CP 640 software package for PCs, which is designed for use with the respective control. The DataPilot is suitable for both shop-floor programming as well as off-location program creation and testing. It is also ideal for training purposes. The DataPilot can be run on PCs with WINDOWS operating systems. HEIDENHAIN provides the DataPilot as a Windows programming station and running in an Oracle VM Virtual Box. Oracle VM VirtualBox is a software application (virtual machine), in which the control software runs as a separate system, i.e. in a virtual environment.

Intended place of operation

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

Legal information

The control software contains open-source software that is subject to special terms of use. These special terms of use have priority.

To call further information on the control:

- Switch to the **Organization** operating mode (floppy disk symbol).
- Switch to the second level of the soft-key row
- Press the **LICENSE INFO** soft key

Software options

The CNC PILOT 640 features various software options that can be enabled by your machine tool builder. Each option is to be enabled separately and contains the following respective functions:

Additional Axis (option 0 to option 7)	
Additional axis	Additional control loops 1 to 8
Teach-in (option 8)	
Teach-in functionality	■ Contour description with ICP ■ Cycle programming ■ Technology database with 9 workpiece-material/tool-material combinations
smart.Turn (option 9)	
	■ Contour description with ICP ■ Programming with smart.Turn ■ Technology database with 9 workpiece-material/tool-material combinations
Tools and Technology (option 10)	
Tool and technology database	■ Tool database expanded to 999 entries ■ Technology database expanded to 62 workpiece-material/tool-material combinations ■ Tool life monitoring with exchange tools
Thread Recutting (option 11)	
Threads	■ Recutting of threads in **Teach-in** submode ■ Handwheel superimposition during thread cutting
HEIDENHAIN DNC (option 18)	
	Communication with external PC applications over COM component
DXF Import (option 42)	
	Loading of DXF contours
B-Axis Machining (option 54)	
B-axis machining	■ Tilting the working plane ■ Tilting the tool carrier ■ High Dynamic Turning
C-Axis Machining (option 54)	
C-axis machining	Boring, drilling, and milling operations on the face and lateral surface
TURN PLUS (option 63)	
	Automatic generation of smart.Turn programs
Y-Axis Machining (option 70)	
Y-axis machining	Boring, drilling, and milling operations on the face and lateral surface

Parallel Axes (option 94)	
Parallel axes	Support of parallel axes (U, V, W)
Spindle Synchronism (option 131)	
Spindle synchronism	Synchronism of several lathe spindles
Counter Spindle (option 132)	
Counter spindle	■ Synchronism of main spindle and counter spindle ■ Rear-face machining
Remote Desktop Manager (option 133)	
Remote operation of external computer units	■ Windows on a separate computer unit ■ Incorporated in the control's interface
Synchronizing Functions (option 135)	
Synchronizing functions	Advanced synchronization of axes and spindles
Load Monitoring (option 151)	
Load monitoring	Monitoring of the axes and spindles
Multichannel (option 153)	
Multi-channel capability	Up to three channels for asynchronous multi-slide machining

New functions of NC software 68894x-17

- The **G160** function allows you to conveniently tilt the working plane. Define a tilting position, up to three spatial angles, and optionally an additional shift after tilting.

  Further information: "Tilt working plane G16", Page 649
- The **G807** function allows you to machine helical or straight-cut cylindrical gears. After selecting the function, you can specify whether machining will take place before or behind the center of rotation or at the inside/outside of the workpiece. You can optionally define an inclination of the tool.

  Further information: "Skiving G807", Page 669

Modified functions in NC software 68894x-17

- With the **DF** or **DFF** parameter, you can define a retraction feed rate for drilling cycles and units.

  Further information: "Units – Drilling / Centric", Page 127

  Further information: "Drilling cycles", Page 421
- If you select the tool manually in the submode **AWG** (option 63), use the **Tool graphics** soft key to display the tool control graphics of the current tool. The control will also take the tool holder into account.

  Further information: "Tool selection and turret assignment list", Page 704
- You can select contour elements using touch gestures or the mouse (e.g., in the **ICP editor** submode or when using the **Dimensioning** function).
- If two holes have been specified with **G49-Geo** in the **FINISHED PART** section, the control will display a warning instead of an error message.

  Further information: "Bore hole (centric) G49–Geo", Page 303
- If multiple slides have been specified and a constant cutting speed is active, the control will display the currently active slide (e.g., **$1** (option 153)) in the machine data display.
- If you stop program execution during a tapping cycle, press the Z key to retract the tool from the thread.

  If the machine manufacturer has activated the optional machine parameter **CfgBackTrack** (no. 122000), you can no longer resume program run with the **NC Start** key after moving the tool manually. You need to restart the program using the **Start blck search** soft key.
- The **Diffuse** HEROS tool was removed.
- In the **Certificate and keys** window you can select a file with additional public SSH keys in the **Externally administered SSH key file** area. This allows you to use SSH keys without needing to transmit them to the control.
- You can export and import existing network configurations in the **Network settings** window.
- The machine manufacturer uses the machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402) to define whether the control disables non-secure LSV2 or RPC connections even if user administration is not active. These machine parameters are included in the data object **CfgDncAllowUnsecur** (135400).

  When the control detects a non-secure connection, it displays an informational notice.

2

First steps

2.1 Overview

This chapter is intended to help you quickly learn to handle the most important procedures on the control. For more information on the respective topic, see the section referred to in the text.

The following topics are covered in this chapter:

- Switching on the machine
- Programming the workpiece
- Graphically testing the workpiece

The following topics are covered in the User's Manual:

- Switching on the machine
- Setting up tools
- Setting up the workpiece
- Machining the workpiece

2.2 Switching on the machine

DANGER

Caution: hazard to the user!

Machines and machine components always pose mechanical hazards. Electric, magnetic, or electromagnetic fields are particularly hazardous for persons with cardiac pacemakers or implants. The hazard starts when the machine is powered up!

- Read and follow the machine manual
- Read and follow the safety precautions and safety symbols
- Use the safety devices

Refer to your machine manual.
Switching on the machine and traversing the reference points can vary depending on the machine tool.

To switch on the machine, proceed as follows:

- Switch on the power supply for the control and the machine
- > The control starts the operating system. This process may take several minutes.
- > The control displays the **Power interrupted** dialog

CE

- Press the **CE** key
- > The control compiles the PLC program.
- > The control displays the error message **Switch on external dc voltage**.

I

- Switch on the machine control voltage
- > The control checks the functioning of the emergency stop circuit.
- > The control is in the **Machine** operating mode.

Further information on this topic

- Switching on the machine
  Further information: User's Manual

2.3 Programming the first workpiece

Selecting an operating mode

To create an NC program, select the **smart.Turn** operating mode.

- Press the **smart.Turn** key
- > The control switches to the **smart.Turn** operating mode.

Further information on this topic

- smart.Turn operating mode
  Further information: "smart.Turn and DIN/ISO programming", Page 62

Important controls and displays

Key	Functions for programming
ENT	▪ Confirm your input ▪ Activate the next dialog prompt
INS	▪ Confirm a dialog ▪ Create a new NC block in the editor
0 9	Number keys 0 to 9: ▪ Enter numbers ▪ Operate the menu
	Go to the next form
ESC	▪ Cancel the dialog ▪ Navigate to the top of the menu
	Select the function in the display unit with the soft key selection key

Further information on this topic

- Navigating in smart.Turn
  Further information: "Selecting the editor functions", Page 70
- Controls and displays
  Further information: User's Manual

Creating a new NC program

Create a new NC program as follows:

- Select the **Prog** menu item
- Select the **New** menu item
- Select the **New DIN PLUS program Ctrl+N** menu item
- > The control opens the **Save as** dialog box.
- Enter a program name

Save

- Press the **Save** soft key
- > The control opens the **Program head (short)** dialog box.

- Define the **Program head (short)**. Example:
  - **Material** = **Aluminum**
  - **Unit** = **Metric**
- Press the **OK** soft key
- > The control opens the NC program.

The control automatically generates the last block of the NC program. You do not need to enter an end-of-program command.

There are various methods of programming available to you:

- **ISO** programming: You program the contour and machining operations with line segments, circular arcs, and simple turning cycles in **ISO Mode**.
- **DIN PLUS** programming: You define the workpiece blank and finished part contours in **ICP editor** submode (option 8 or 9). The workpiece is machined using contour-based turning cycles in **ISO Mode**.
- **smart.Turn** programming (option 9): You program the workpiece blank and finished part contours in **ICP editor** submode. The workpiece is machined by means of **smart.Turn units**.
- **TURN PLUS** (option 63): You program the workpiece blank and finished part contours in **ICP editor** submode. The control automatically generates the working plan and the machining process.

Further information on this topic

- Programming in **smart.Turn** operating mode
  Further information: "smart.Turn and DIN/ISO programming", Page 62
- **ICP editor** submode
  Further information: User's Manual
- Programming in **ISO Mode**
  Further information: "Programming in ISO Mode", Page 274
- Programming with **smart.Turn**
  Further information: "Units – smart.Turn units", Page 98
- Programming with **TURN PLUS**
  Further information: "TURN PLUS (Option 63)", Page 681
- Navigating with menu items
  Further information: "Shared menu items", Page 72

Setting up the turret list

You need to set up the turret list before you can call tools in the NC program. The turret list shows the tools currently in the turret.

Filling the turret list

- ▶ Select the **Head** menu item
- ▶ Select the **Set up the turret list** menu item
- > The control opens the turret list.

The turret list has already been defined in **Machine** operating mode:

- ▶ Press the **Spec. functions** soft key
- ▶ Press the **Transfer machine** soft key
- > The control adopts the turret assignment from the **Machine** operating mode.

Turret list not defined yet:

- ▶ Press the **Tool list** soft key
- > The control displays the tool list.
- ▶ Use the **Pocket down** and **Pocket up** soft keys to select the desired line in the turret list
- ▶ Select the desired tool in the tool table
- ▶ Press the **Load tool** soft key
- > The selected tool is loaded into the turret list.
- ▶ Transfer all required tools to the turret list
- ▶ Press the **Back** soft key
- ▶ Press the **Save** soft key
- > The control saves the turret assignment list to the **TURRET** section in the NC program.

Further information on this topic

- Turret list
  Further information: "Tool programming", Page 91
- Tools
  Further information: User's Manual
- Structure of an NC program
  Further information: "Program section code", Page 80

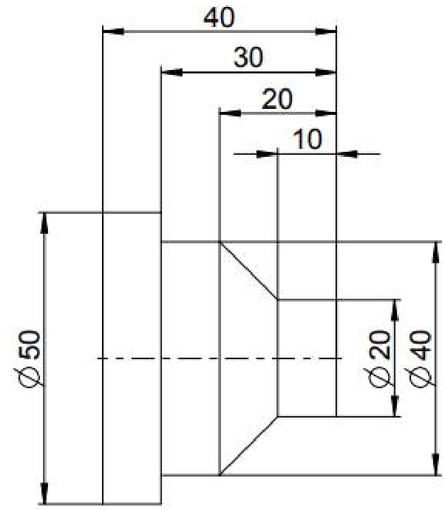
Programming a contour in ICP submode (option 8 or 9)

Your task is to create a turning program for the aluminum part shown to the right. You have already created the NC program.

To define the workpiece blank and finished part contour in **ICP editor** submode, proceed as follows:

Defining the workpiece blank

- ▶ Select the **ICP** menu item
- ▶ Select the **Workpiece blank** menu item
- > The control opens the **ICP editor** submode.
- ▶ Select the **Bar** menu item
- > The control opens the **Bar** dialog box.
- ▶ Enter the workpiece blank dimensions:
  - **X**: **Diameter** = 60 mm
  - **Z**: **Length** of workpiece blank = 60 mm
  - **K**: **Oversz** – Distance between workpiece datum and right edge = 1 mm
- ▶ Press the **Save** soft key
- > The **ICP editor** submode displays the workpiece blank.
- ▶ Press the **Back** soft key



Defining the finished part contour

- ▶ Select the **ICP** menu item
- ▶ Select the **Finished part** menu item
- > The control opens the **ICP editor** submode.
- ▶ Select the **Contour** menu item
- ▶ Select the **Line** menu item
- ▶ Enter the coordinates:
  - **XS**: **Start point** of contour = 0 mm
  - **ZS**: **Starting point** of contour = 0 mm
  - **X**: **Target point** = 20 mm
- ▶ Press the **Save** soft key
- ▶ Select the **Line** menu item
- ▶ **Z**: **Target point** = –10 mm
- ▶ Press the **Save** soft key
- ▶ Select the **Line** menu item
- ▶ Enter the coordinates:
  - **X**: **Target point** = 40 mm
  - **Z**: **Target point** = –20 mm
- ▶ Press the **Save** soft key

- Select the **Line** menu item
- **Z**: **Target point** = –30 mm
- Press the **Save** soft key

- Select the **Line** menu item
- **X**: **Target point** = 50 mm
- Press the **Save** soft key

- Select the **Line** menu item
- **Z**: **Target point** = –40 mm
- Press the **Save** soft key

- Select the **Line** menu item
- **X**: **Target point** = 0 mm
- Press the **Save** soft key

- Press the **Back** soft key

- Press the **Back** soft key
- > The control saves the defined contours to the NC program.

Further information on this topic

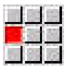
- **ICP editor** submode
  Further information: User's Manual
- Program sections for contours
  Further information: "Structured NC program", Page 63

Programming the machining process in smart.Turn (option 9)

After defining the workpiece blank and the finished part contour in **ICP editor** submode, you program the machining of the workpiece using **smart.Turn units**.

Program the machining process as follows:

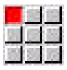
Defining the Start unit



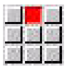
- ▶ Select the **Units»** menu item
- > The control opens the **Program beginning** dialog window.
- ▶ In **S0**, enter the maximum speed for spindle 1 (e.g., 4000 rpm)
- ▶ Define the maximum speed for all of the available spindles

- ▶ Press the **STORE** soft key
- > The control closes the dialog window and saves the unit.

Transverse roughing in ICP



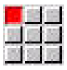
- ▶ Select the **Roughing** menu item



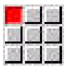
- ▶ Select the **G820 Transverse in ICP** menu item
- > The control opens the **G820 Roughing in ICP** dialog window.
- ▶ Define the parameters:
  - **XS**: **Approach position X** (e.g., 60 mm)
  - **ZS**: **Approach position Z** (e.g., 2 mm)
  - **T**: **Tool number** – turret pocket number
  - **F**: **Feed per revolution** (e.g., 0.4 mm/rev)
  - **S**: **Cutting speed** in m/min (e.g., 220 m/min)
  - **NS**: **Starting block no. of contour** – beginning of contour section = 3
  - **NE**: **Contour end block no.** – end of contour section = 3
  - **P**: **Maximum infeed** (e.g., 5 mm)

- ▶ Press the **STORE** soft key
- > The control closes the dialog window and saves the unit.

Longitudinal roughing in ICP



- Select the **Roughing** menu item

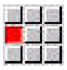


- Select the **G810 Longitudinal in ICP** menu item
- > The control opens the **G810 Longitudinal roughing in ICP** dialog window.
- Define the parameters:
  - **XS**: **Approach position X** (e.g., 60 mm)
  - **ZS**: **Approach position Z** (e.g., 2 mm)
  - **T**: **Tool number** – turret pocket number
  - **F**: **Feed per revolution** (e.g., 0.4 mm/rev)
  - **S**: **Cutting speed** in m/min (e.g., 220 m/min)
  - **NS**: **Starting block no. of contour** – beginning of contour section = 4
  - **NE**: **Contour end block no.** – end of contour section = 8
  - **P**: **Maximum infeed** (e.g., 5 mm)

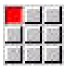
- Press the **STORE** soft key
- > The control closes the dialog window and saves the unit.

Finishing in ICP

Transverse finishing:



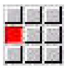
- Select the **Finishing** menu item



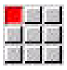
- Select the **G890 Contouring in ICP** menu item
- > The control opens the **G890 ICP for contouring** dialog window.
- Define the parameters:
  - **XS**: **Approach position X** (e.g., 20 mm)
  - **ZS**: **Approach position Z** (e.g., 2 mm)
  - **T**: **Tool number** – turret pocket number
  - **F**: **Feed per revolution** (e.g., 0.25 mm/rev)
  - **S**: **Cutting speed** in m/min (e.g., 240 m/min)
  - **NS**: **Starting block no. of contour** – beginning of contour section = 3
  - **NE**: **Contour end block no.** – end of contour section = 3

- Press the **STORE** soft key
- > The control closes the dialog window and saves the unit.

Longitudinal finishing:



- Select the **Finishing** menu item



- Select the **G890 Contouring in ICP** menu item
- > The control opens the **G890 ICP for contouring** dialog window.
- Define the parameters:
  - **XS**: **Approach position X** (e.g., 20 mm)
  - **ZS**: **Approach position Z** (e.g., 2 mm)
  - **T**: **Tool number** – turret pocket number
  - **F**: **Feed per revolution** (e.g., 0.25 mm/rev)
  - **S**: **Cutting speed** in m/min (e.g., 240 m/min)
  - **NS**: **Starting block no. of contour** – beginning of contour section = 4
  - **NE**: **Contour end block no.** – end of contour section = 8

- Press the **STORE** soft key
- > The control closes the dialog window and saves the unit.

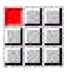
Further information on this topic

- Program beginning unit
  Further information: "Unit Program beginning (START) ", Page 228
- smart.Turn units
  Further information: "Units – smart.Turn units", Page 98
- Contour-based NC programming
  Further information: "smart.Turn unit", Page 98

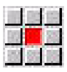
Closing an NC program

Up to six NC programs can be open simultaneously in **smart.Turn** operating mode. The control automatically saves new NC programs you created.

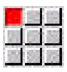
Close an NC program as follows:



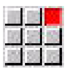
- Select the **Prog** menu item



- Select the **Save** menu item
- > The control saves the NC program.



- Select the **Prog** menu item



- Select the **Close** menu item
- > **smart.Turn** operating mode closes the NC program.

Further information on this topic

- Prog menu item
  Further information: "Shared menu items", Page 72

Programming a contour in ISO Mode

Your task is to create a turning program for the aluminum part shown to the right. You use **ISO Mode** for programming. You have already created the NC program and set up the turret list.

Proceed as follows to display the contour during programming:

- ▶ Press the **Graph.** soft key
- > The control opens a graphic window.
- ▶ Press the **Graph.** soft key to update the graphic.
- > The graphic shows the newly programmed contours.

Switch off the graphic as follows:

- ▶ Select the **Graph.** menu item
- ▶ Select the **Graphic OFF** menu item
- > The control closes the graphic window.

Create the workpiece blank and the finished part contour as follows:

Defining the workpiece blank

ISO Mode

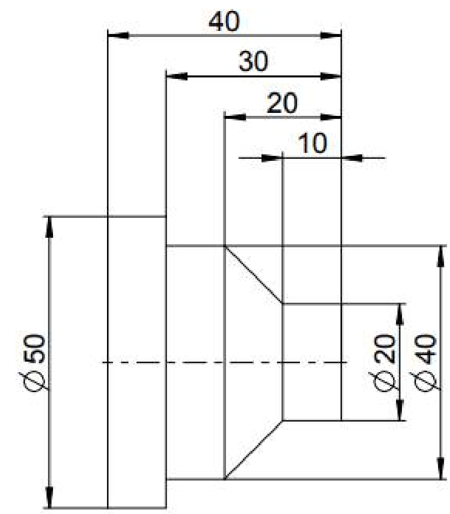
- ▶ Press the **ISO Mode** soft key
- ▶ Select the **Geo»** menu item
- > The control opens a G-code menu for defining the contour.
- ▶ Position the cursor in the **BLANK** program section
- ▶ Select the **G** menu item
- ▶ Enter **20**

OK

- ▶ Press the **OK** soft key
- > The control opens the command **G20**, **Chuck part cyl./tube**.
- ▶ Enter the workpiece blank dimensions:
  - **X**: **Diameter** = 60 mm
  - **Z**: **Length** of workpiece blank = 60 mm
  - **K**: **Righthnd edge** – Transverse oversize = 1 mm

Save

- ▶ Press the **SAVE** soft key
- > The control saves the workpiece blank.



Defining the finished part

- Position the cursor in the **FINISHED PART** program section
- Select the **G** menu item
- Enter **0**
- Press the **OK** soft key
- > The control opens the command **G0**, **Start point**.
- Enter the coordinates:
  - **X**: **Start point** X = 0 mm
  - **Z**: **Start point** Z = 0 mm
- Press the **SAVE** soft key
- > The control saves the starting point.
- Select the **Line** menu item
- > The control opens the command **Distance G1**.
- **X**: **Final point** (diameter value) = 20 mm
- Press the **Save** soft key

- Select the **Line** menu item
- **Z**: **Final point** = –10 mm
- Press the **Save** soft key

- Select the **Line** menu item
- Enter the coordinates:
  - **X**: **Final point** = 40 mm
  - **Z**: **Final point** = –20 mm
- Press the **Save** soft key

- Select the **Line** menu item
- **Z**: **Final point** = –30 mm
- Press the **Save** soft key

- Select the **Line** menu item
- **X**: **Final point** = 50 mm
- Press the **Save** soft key

- Select the **Line** menu item
- **Z**: **Final point** = –40 mm
- Press the **Save** soft key

- Select the **Line** menu item
- **X**: **Final point** = 0 mm
- Press the **Save** soft key
- > The finished-part contour is defined.

Further information on this topic

- Menu item bar in **ISO Mode**
  Further information: "Geometry menu item", Page 286
- NC program sections
  Further information: "Structured NC program", Page 63
- ISO programs
  Further information: "Geometry and machining commands", Page 274
- Description of workpiece blank
  Further information: "Chuck part bar or tube G20-Geo", Page 287
- Description of finished part
  Further information: "Basic elements of turning contour", Page 288

Programming the machining process in ISO Mode

After defining the workpiece blank and the finished part contour, you program the machining process using machining cycles.

Program the machining process as follows:

Defining the speed limit

- ▶ Position the cursor in the **MACHINING** program section
- > The menu item bar displays the G codes for machining.
- ▶ Select the **G** menu item
- ▶ Enter **26**

OK

- ▶ Press the **OK** soft key
- > The control opens the command **Speed limitation G26**.

Navigating using the **G-menu**:

- ▶ Select the **G-menu** menu item
- ▶ Select the **Feed rate, rpm** menu item
- ▶ Select the **Speed** menu item
- ▶ Select the **Limitation m.spin G26** menu item
- > The control opens the command **Speed limitation G26**.
- ▶ Enter the maximum speed (e.g., 4000 rpm)

Save

- ▶ Press the **STORE** soft key
- > The control saves the speed limit.

Defining machining cycles

HEIDENHAIN recommends programming a machining cycle as follows:

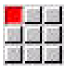
- Insert the tool
- Define the cutting data
- Position the tool in front of the machining area
- Define the safety clearance
- Call the cycle
- Retract the tool
- Move to the tool change position

Inserting the tool

Moving to the tool change point:

- Press the **INS** key
- > The control opens a dialog window for confirming the block number.

- Press the **INS** key
- > The control creates a new NC block.



- Select the **G** menu item
- Enter **14**

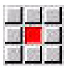
- Press the **OK** soft key
- > The control opens the command **Tool change point G14**

- Press the **STORE** soft key
- > The control saves the traverse. The tool change position is approached simultaneously.

Calling the tool:

- Press the **INS** key
- > The control opens a dialog window for confirming the block number.

- Press the **INS** key
- > The control creates a new NC block.



- Select the **T** menu item
- > The control opens the command **Tool**.
- Enter the tool number

- Press the **STORE** soft key
- > The control saves the NC block.

Defining the cutting data

Defining the feed rate:

- Press the **INS** key to create a new NC block
- Select the **F** menu item
- > The control opens the command **Feed per rotation**.
- Enter the feed rate value (e.g., 0.4 mm/rev)
- Press the **STORE** soft key

Defining the cutting speed:

- Select the **S** menu item
- > The control opens the command **Cutting speed**.
- Enter the cutting speed value (e.g., 220 m/min)
- Press the **STORE** soft key

Switching on the spindle in counterclockwise rotation:

- Select the **M** menu item
- Enter **4**
- Press the **OK** soft key
- > The control saves the command **M4**, **Spindle on, CCW**.

Positioning the tool in front of the machining area

- Press the **INS** key to create a new NC block
- Open the command **Rapid traverse G0**
- Press the **OK** soft key
- Enter the coordinates to be approached:
  - **X** = 62 mm
  - **Z** = 2 mm
- Press the **STORE** soft key

Defining the safety clearance

- Press the **INS** key to create a new NC block
- Open the command **Safety clearance G47**
- Press the **OK** soft key
- Enter the safety clearance (e.g., 2 mm)
- Press the **STORE** soft key

Calling the roughing cycle

Transverse roughing:

INS

- Press the **INS** key to create a new NC block
- Open the command **Face roughing G820**

OK

- Press the **OK** soft key
- Define the parameters:
  - **NS**: **Starting block no. of contour** – beginning of contour section = 3
  - **NE**: **Contour end block no.** – end of contour section = 3
  - **P**: **Maximum infeed** (e.g., 5 mm)
  - **K**: **O-size Z** = 0.2 mm

Save

- Press the **STORE** soft key

Switching on the coolant:

- Select the **M** menu item
- Enter **8**

OK

- Press the **OK** soft key
- > The control saves the command **M8**, **Coolant circuit 1 on**.

Longitudinal roughing:

INS

- Press the **INS** key to create a new NC block
- Open the command **Longitud. roughing G810**

OK

- Press the **OK** soft key
- Define the parameters:
  - **NS**: **Starting block no. of contour** – beginning of contour section = 4
  - **NE**: **Contour end block no.** – end of contour section = 8
  - **P**: **Maximum infeed** (e.g., 5 mm)
  - **I**: **O-size X** = 0.5 mm
  - **K**: **O-size Z** = 0.2 mm

Save

- Press the **STORE** soft key

Switching off the coolant:

- Open the menu item **M9**, **All circuits off**

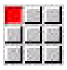
OK

- Press the **OK** soft key

Inserting a new tool

Moving to the tool change position:

- Press the **INS** key to create a new NC block



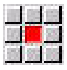
- Open the command **Tool change point G14**

- Press the **OK** soft key

- Press the **STORE** soft key

Calling the tool:

- Press the **INS** key to create a new NC block



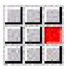
- Select the **T** menu item
- > The control opens the command **Tool**.
- Enter the tool number

- Press the **STORE** soft key

Defining the cutting data

Defining the feed rate:

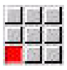
- Press the **INS** key to create a new NC block



- Select the **F** menu item
- Define **Feed per rotation** (e.g., 0.25 mm/rev)

- Press the **STORE** soft key

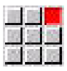
Defining the cutting speed:



- Select the **S** menu item
- Define the **Cutting speed** (e.g., 240 m/min)

- Press the **STORE** soft key

Switching on the spindle in counterclockwise rotation:



- Open the menu item **M4**, **Spindle on, CCW**

- Press the **OK** soft key

Pre-positioning the tool

INS

- Press the **INS** key to create a new NC block
- Open the command **Rapid traverse G0**

OK

- Press the **OK** soft key
- Enter the coordinates to be approached:
  - **X** = 20 mm
  - **Z** = 2 mm

Save

- Press the **STORE** soft key

Calling the finishing cycle

Transverse finishing:

INS

- Press the **INS** key to create a new NC block
- Open the command **Finish contour G890**

OK

- Press the **OK** soft key
- Define the parameters:
  - **NS**: **Starting block no. of contour** – beginning of contour section = 3
  - **NE**: **Contour end block no.** – end of contour section = 3

Save

- Press the **STORE** soft key

Switching on the coolant:

- Open the menu item **M8**, **Coolant circuit 1 on**

OK

- Press the **OK** soft key

Longitudinal finishing:

INS

- Press the **INS** key to create a new NC block
- Open the command **Finish contour G890**

OK

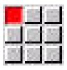
- Press the **OK** soft key
- Define the parameters:
  - **NS**: **Starting block no. of contour** – beginning of contour section = 4
  - **NE**: **Contour end block no.** – end of contour section = 8

Save

- Press the **STORE** soft key

Retracting the tool

- Press the **INS** key to create a new NC block



- Open the command **Tool change point G14**

- Press the **OK** soft key

- Press the **STORE** soft key

Further information on this topic

- Programming in **ISO Mode**
  Further information: "Programming in ISO Mode", Page 274
- Menu item bar in **ISO Mode**
  Further information: "Machining menu item", Page 286
- Defining the datum
  Further information: "Datum shifts", Page 343
- Machine setup
  Further information: User's Manual
- Feed rate and spindle speed
  Further information: "Feed rate, shaft speed", Page 336
- Tool change position G14
  Further information: "Tool change point G14", Page 332
- Miscellaneous functions M
  Further information: "Machine commands", Page 546
- Pre-positioning the tool
  Further information: "Rapid traverse G0", Page 331
- Safety clearance
  Further information: "Safety clearance G47", Page 349
- Machining cycles
  Further information: "Contour-based turning cycles", Page 354

TURN PLUS programming (option 63)

To create an NC program with **TURN PLUS**, you define the workpiece blank and the finished part in **ICP editor** submode. Then you have the control create the working plan and the NC program based on a defined **Machining sequence**.

For detailed explanations and a step-by-step example of this topic, refer to:

- Step-by-step example see "Example", Page 711
- Detailed information on **TURN PLUS** see "The function TURN PLUS", Page 682
- Detailed information on the **Machining sequence** see "Automatic working plan generation (AWG) submode", Page 683

2.4 NC program verification in the simulation

To verify your NC program, you can test the machining process in **Simulation** submode.

Open the NC program in **Simulation** submode as follows:

- ▶ Switch to **smart.Turn** operating mode
- ▶ Select the **Prog** menu item
- ▶ Select the **Open ...** menu item
- > The control opens the **Open** dialog box.
- ▶ Select the desired NC program
- ▶ Press the **Open** soft key
- > The control opens the NC program.
- ▶ Press the **Simulation** soft key
- > The control opens the **Simulation** submode.
- ▶ Press the **Start simulation** soft key

Close the **Simulation** submode as follows:

- ▶ Press the **Back** soft key
- > The control opens the **smart.Turn** operating mode.

Further information on this topic

- Simulation submode
  Further information: User's Manual

3

NC Programming

3.1 smart.Turn and DIN/ISO programming

The control supports the following types of programming:

- **Conventional ISO programming**: Program the basic contour with line segments, circular arcs, and simple turning cycles. Use **ISO Mode** in **smart.Turn** operating mode
- **DIN PLUS programming**: The geometrical description of the workpiece and the machining process are separated. First, program the geometry of the workpiece blank and the finished part. Then, machine the workpiece using contour-related turning cycles. Use **ISO Mode** in **smart.Turn** operating mode
- **smart.Turn programming**: The geometrical description of the workpiece and the machining process are separated. Program the geometry of the blank and the finished part and program the machining blocks as **Units»**. Use these **Units»** in the **smart.Turn** mode of operation

Depending on the type and complexity of your machining task, you can use either simple DIN/ISO programming, DIN PLUS (ISO) programming or smart.Turn programming. All three named programming modes can be combined in one NC program.

In DIN PLUS and smart.Turn programming, contours can be described with **ICP** interactive graphics. **ICP** saves the contour descriptions as **G** codes in the NC program.

Parallel operation: While you are editing and testing programs, your machine can run **another** NC program.

> In **smart.Turn** operating mode you can create a program list (Automatic jobs) that is run automatically in Program run mode.

Contour follow-up

With DIN PLUS and smart.Turn programs, the control uses Contour follow-up. The control takes the Workpiece blank as a basis and accounts for each cut and each cycle in Contour follow-up. Thus you can inspect the current contour of the workpiece during each machining stage. With the **contour follow-up** function, the control optimizes the approach and departure paths and avoids air cut passes.

Contour follow-up is only available for turning operations if a Workpiece blank has been programmed. It is also performed with Auxiliary contour.

Structured NC program

smart.Turn and DIN PLUS programs are structured in fixed sections.

The following program sections are created automatically with a new NC program:

> Your machine tool builder can define the contents of newly created NC programs in a default template.
>
> If there is no default template defined, the control will automatically take into account the number of turrets, etc. This may cause more than one **TURRET** section being created in the new NC program.
>
> For machines with a counter spindle, the control will automatically insert the **CHUCKING EQUIPMENT 2** section.

- **PROGRAM HEAD**: Contains information on the material of the workpiece, the unit of measure, as well as further organizational data and setup information in the form of a comment
- **CHUCKING EQUIPMENT**: Description of the workpiece clamping situation—for machines with a counter spindle, a second line is generated
- **BLANK**: The BLANK is stored. Programming a workpiece blank activates Contour follow-up
- **FINISHED PART**: The FINISHED PART is stored. It is advisable to describe the complete workpiece as a FINISHED PART. The units or machining cycles use **NS** and **NE** to indicate the workpiece section to be machined.
- **MACHINING**: Use units and cycles to program the individual machining steps. In a smart.Turn program, the START unit is located at the beginning of the machining process, and the END unit at the end
- **END**: Indicates the end of the NC program

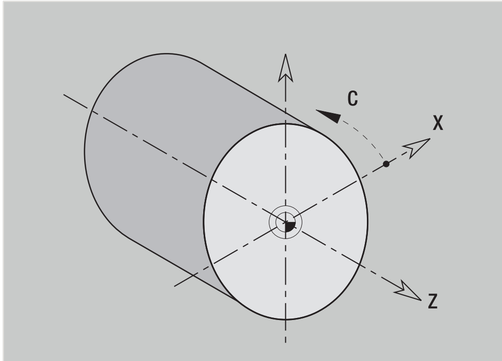
If required, for example for machining with the C axis or when programming with variables, you add further program sections.

> Use the **ICP editor** (Interactive Contour Programming) submode for describing the contours of workpiece blanks and finished parts.

Example: structured smart.Turn program

PROGRAM HEAD		
#UNIT	METRIC	
#MATERIAL	Steel	
#MACHINE	Automatic lathe	
#DRAWING	356_787.9	
#CLAMP. PRESS.	20	
#COMPANY	Turn & Co	
TURRET		
T1 ID"038_111_01"		
T2 ID"006_151_A"		
CHUCKING EQUIPMENT		
H0 D0 Z200 B20 O-100 X120 K12 Q4		
BLANK		
N1 G20 X120 Z120 K2		
FINISHED PART		
N2 G0 X0 Z0		
N3 G1 X20 BR3		
N4 G1 Z-24		
...		
MACHINING		
N50 UNIT ID"START"		[**Program beginning**]
N52 G26 S4000		
N53 G59 Z320		
N54 G14 Q0		
N25 END_OF_UNIT		
...		
		[Machining commands]
...		
N9900 UNIT ID"END"		[**Program end**]
N9902 M30		
N9903 END_OF_UNIT		
END		

Linear and rotary axes



Principal axes: Coordinates of the X, Y and Z axes refer to the workpiece datum.

C axis as reference axis:

- Angle data are given with respect to the **datum of the C axis**
- C-axis contours and C-axis operations:
  - Positions on the front/rear face are entered in Cartesian coordinates (**XK**, **YK**) or polar coordinates (**X**, **C**)
  - Positions on the lateral surface are entered in polar coordinates (**Z**, **C**). Instead of **C**, the linear value **CY** is used (**"unrolled"** reference diameter)

> **smart.Turn** mode of operation respects only address letters of the configured axes.

Units of measure

You write NC programs in **metric** or **inch** values. The unit of measure is defined in the **Unit** box.

Further information: "PROGRAM HEAD section", Page 82

> Once the unit of measure has been defined, it can no longer be edited.

Elements of an NC program

An NC program consists of the following elements:

- Program name
- Program section identifiers
- Units
- NC blocks
- Commands for program structuring
- Comment blocks

Program name

The **program name** starts with a digit or a letter, followed by up to 40 characters and the **.nc** extension for main programs and the **.ncs** extension for subprograms.

For program names, you can use any ASCII characters except:

~ * ? < > | / \ : " % #

The following characters have special meanings:

Character	Meaning
.	The last period (dot) in a file name is the extension separator
**** and **/**	Directory separators
:	Separates the drive name from the directory

Program section identifiers

For new NC programs, certain program section codes are already predefined. You can add new sections or delete existing codes, depending on your programming requirements. An NC program must contain at least the **MACHINING** and **END** section codes.

> If you create NC programs externally or edit them in **Text editor (character)** input mode, unknown words may result from typing errors (e.g., **THN** instead of **THEN**).
>
> In the NC mode of the editor, the control checks the DIN PLUS words and the information in the program head. The editor highlights lines containing unknown words in magenta. When you simulate or run the NC program, the control will indicate an error.

UNIT

The **UNIT** begins with this keyword followed by the identification of the **unit (ID"G...")**. The following lines contain the **G**, **M** and **T** functions of this machining block. The unit ends with **END_OF_UNIT**, followed by a check digit.

NC blocks

NC blocks start with an **N**, followed by a block number (with up to five digits). The block numbers do not affect the sequence in which the program blocks are executed. Their purpose is to identify the individual NC blocks. The NC blocks of the **PROGRAM HEAD** and **TURRET** or **MAGAZINE** sections are not included in the block number organization of the editor.

Commands for program structuring

You can use **program branches, repeats and subprograms** to structure a program (example: machining the front/back of a bar, etc.).

Input and output: With inputs, you influence the flow of the NC program. Outputs can be used to communicate information to the machine operator. Example: The machine operator is requested to check measuring points and update compensation values.

The **skip level** influences the execution of individual NC blocks.

On machines with multiple slides, the **slide code** is used to assign the NC blocks to a slide.

Comment blocks

Comments are enclosed in brackets **[...]**. They are located at the end of an NC block or in a separate NC block. Press the **CTRL** + **K** key combination to convert an existing block into a comment (and vice versa). You can also enclose multiple program lines in square brackets to mark them as a comment.

Creating a new NC program

To create a new NC program, proceed as follows:

- ▶ Select the **smart.Turn** operating mode
- ▶ Select the **Prog** menu item
- ▶ Select the **New** menu item
- ▶ Select the **New DIN PLUS program Ctrl+N** menu item
- > The control opens the **Save as** dialog box.
- ▶ Enter a program name
- Save ▶ Press the **Save** soft key
- > The control opens the **Program head (short)** dialog box.
- ▶ Define the program header, if required.
- OK ▶ Press the **OK** soft key

3.2 Fundamentals: smart.Turn editor

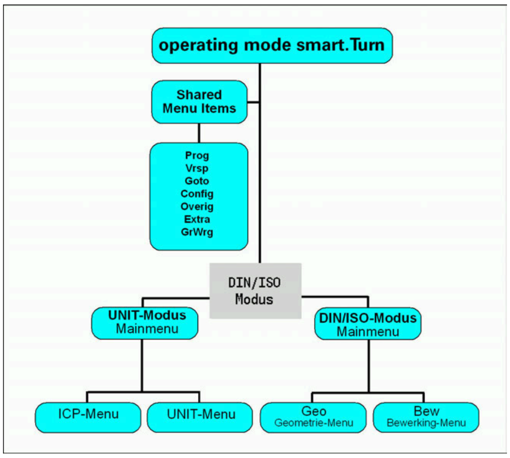
Menu structure

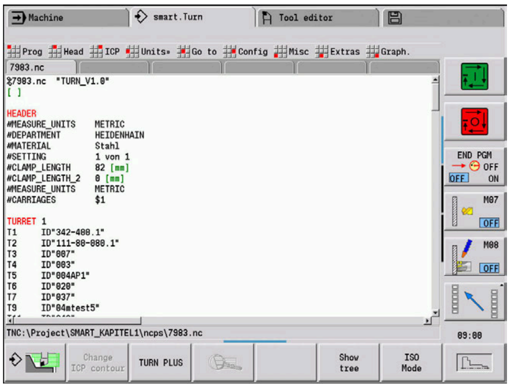
You can select the following editor modes in the **smart.Turn** operating mode:

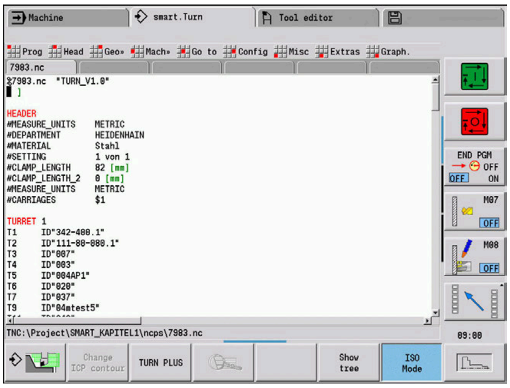
- Unit programming (standard)
- **ISO Mode** (DIN PLUS and DIN 66025)

The menu structure of the **smart.Turn** operating mode is shown in the figure at right. Many menu items are used in both modes. The menus differ with respect to the programming of geometries and machining operations. In **ISO Mode**, the **Geo»** (Geometry) and **Mach»** (Machining) menu items are displayed instead of the **ICP** and **Units»** menu items. You can switch between the editor modes by soft key.

- Switches between the **Unit** mode and the **ISO Mode**







For special cases, you can change to text-editor mode in order to make character-by-character changes without syntax checking. To make this setting, select the **Config Input mode** menu item.

For a description of the functions, please refer to the following chapters:

- ICP functions
  Further information: User's Manual
- Units for turning and C-axis machining
  Further information: "smart.Turn Units (Option 9)", Page 97
- Units for Y-axis machining
  Further information: "smart.Turn Units for the Y Axis (Option 9 and Option 70)", Page 235
- **G** codes for turning and C-axis machining (geometry and machining)
  Further information: "DIN/ISO Programming", Page 273
- **G** codes for Y-axis machining (geometry and machining)
  Further information: "ISO Programming for the Y Axis (Option 70)", Page 627

Parallel editing

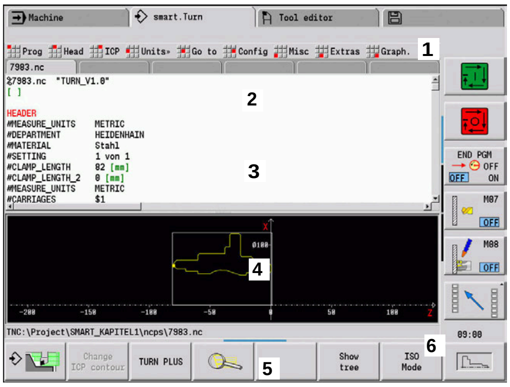
Up to six NC programs can be opened simultaneously in **smart.Turn** operating mode. The editor shows the names of the open programs in the tab bar. If you have changed the NC program, the editor displays the name in red.

You can program in the **smart.Turn** operating mode while the machine is running a program in the automatic mode.

- The **smart.Turn** operating mode saves all open programs every time the operating mode is switched
- The program running in the automatic mode cannot be edited

Screen layout

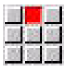
1 Menu bar
2 NC program bar with the names of the loaded NC programs. The selected program is marked
3 Program window
4 Contour display or large program window
5 Soft keys
6 Status bar



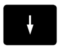
Selecting the editor functions

The functions of the **smart.Turn** operating mode are contained in the **main menu** and various **submenus**.

The submenus can be called by:

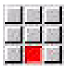


- Selecting the desired menu item



- Positioning the cursor in the respective program section

You can access the higher-level menu:

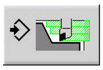


- By pressing the menu item

ESC

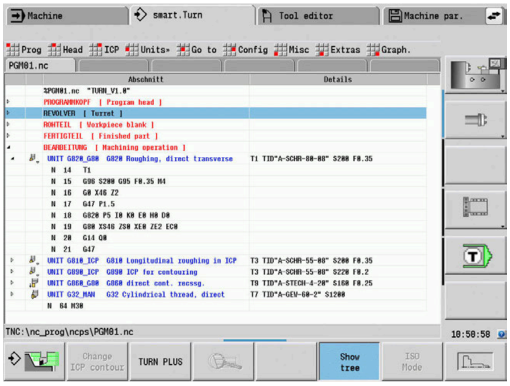
- Or alternatively by pressing the **ESC** key

Soft keys: Soft keys are available for fast switching to "neighboring operating modes," for changing the editing window or program view, and for activating the graphics.

Soft keys with active program window	
	Starts the current program in **Simulation** submode
Change ICP contour	Opens the contour, in which the cursor is located, in **ICP**
	Activates the zoom function in the contour display
Show tree	Switches between the NC editor and the tree view
ISO Mode	Switches between the Unit mode and **ISO Mode**
	Activates the contour display and starts redrawing the contour

Editing with active tree view

- Press the right arrow key to expand the program sections.
- Position the cursor on the program line you want to edit and press the right arrow key once again.
- > The control switches to the NC editor.
- Make the required changes
- Use the left cursor key to return to the tree view and to collapse the program section

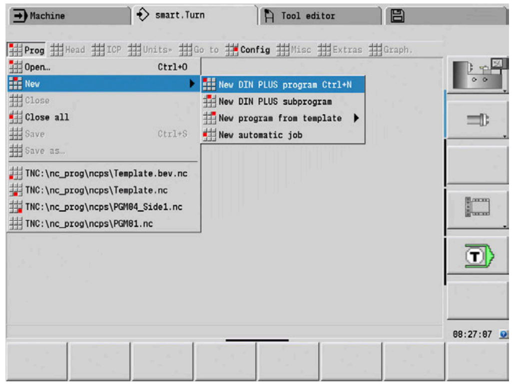


> In the **MACHINING** section, you can adapt the tree view to your requirements (e.g., by combining multiple units to create a custom range of blocks). Define the new range of blocks by inserting the DIN PLUS word **BLOCKSTART** at the beginning of the selected program section and the DIN PLUS word **BLOCKEND** at the end. The DIN PLUS words are available from the **Extras** menu in the **DIN PLUS word...** menu item.

Shared menu items

The menu items described below are used in both **smart.Turn** mode and **ISO Mode**.

Prog menu item



The **Prog** (Program management) menu item contains the following functions for NC main programs and subprograms:

- **Open ...**: Load existing NC programs
- **New**: Create new NC programs or **Automatic jobs**
- **Close**: The selected NC program is closed
- **Close all**: All open NC programs are closed
- **Save**: The selected NC program is saved
- **Save as...**: The selected NC program is saved under a new name
- Direct opening of the last four programs

When an NC program is opened or when a new NC program is created, the soft-key row is switched to the sorting and organization functions.

Further information: "Sorting and file organization", Page 78

Head (program head) menu item

The menu item **Head** (program head) contains functions for editing the program head and the tool list.

- **Program head**: Edit the program head
- **Go to turret list** (**Go to tool list**): Positions the cursor in the **TURRET** section
- **Set up the turret list** (**Setup the tool list**): Activates the "Set up turret list" function
  Further information: "Setting up a turret list", Page 91
- **Go to magazine**: Positions the cursor in the **MAGAZINE** section (machine-dependent)
- **Set up magazine list**: Activates the "Set up magazine list" function (machine-dependent)
- **Go to chucking equipment**: Positions the cursor in the **CHUCKING EQUIPMENT** section.
- **Insert chucking equipment**: Describe how the workpiece is clamped
- **Go to manual tool**: Positions the cursor in the **MANUAL TOOL** section.

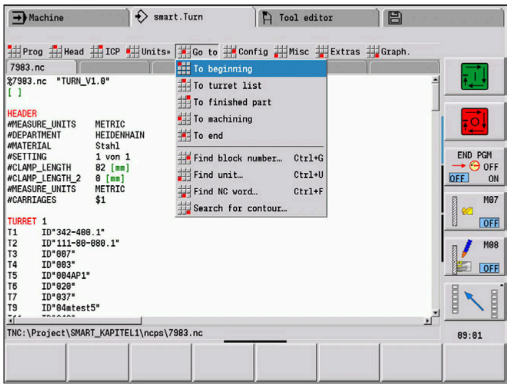
ICP menu item

The **ICP** (Interactive Contour Programming) menu item contains the following functions:

- **Contour editing**: Change the current contour (cursor position)
- **Workpiece blank**: Edit the description of the workpiece blank
- **Finished part**: Edit the description of the finished part
- **New auxiliary blank**: Create a new auxiliary workpiece blank
- **New aux. contour**: Create a new auxiliary contour
- **C axis**: Create patterns and milling contours on the front face and lateral surface
- **Y axis**: Create patterns and milling contours in the XY and YZ planes
- **Insert contour**: Insert the saved workpiece blank contour and finished part contour (only active if you have already saved a contour via **Simulation** submode)

Go to menu item

The **Go to** menu item contains the following jump and search functions:



- Jump targets—The editor positions the cursor to the selected jump target:
  - **To beginning**
  - **To turret list** (**To tool table**)
  - **To finished part**
  - **To machining**
  - **To end**
- Search functions
  - **Find block number... Ctrl+G**: You specify a certain block number. The editor jumps to this block number if it exists
  - **Find unit... Ctrl+U**: The editor opens the list of units available in the NC program. Select the desired unit
  - **Find NC word... Ctrl+F**: The editor opens the dialog for entering the desired NC word. You can use the soft keys to search forward or backward
  - **Search for contour...**: The editor opens the list of contours available in the NC program. Select the desired contour

Config menu item

The **Config** (configuration) menu item contains the following functions:

- **Display settings...** opens a dialog window that provides the following settings:
  - **Color display of technology**: The editor shows the technology values **T**, **S**, **F**, and **M** in violet
  - **Color display of FMAX**: The editor shows the command **G0** in brown
  - **Font size**: Permanent setting of the font size in the NC editor; the default value is identified by an asterisk (*)
  - **Font size for tree view**: Permanent setting of the font size in the tree view of the NC program; the default value is identified by an asterisk (*)
- **Reduce font**: Reduce the font size in the NC editor; as a result, the display settings will change
- **Enlarge font**: Enlarge the font size in the NC editor; as a result, the display settings will change
- **Input mode**: Define the mode
  - **NC editor (word-by-word)**: The editor checks for syntax errors in NC mode
  - **Text editor (character)**: The editor operates character by character (no syntax checking)

> If you create NC programs externally or edit them in **Text editor (character)** input mode, unknown words may result from typing errors (e.g., **THN** instead of **THEN**).
>
> In the NC mode of the editor, the control checks the DIN PLUS words and the information in the program head. The editor highlights lines containing unknown words in magenta. When you simulate or run the NC program, the control will indicate an error.

- **Settings**
  - **Save**: The editor memorizes the open NC programs and the respective cursor positions
  - **Load last saved** setting: Restores the last saved condition of the editor
- **Technology data**: Start **Technology editor** submode

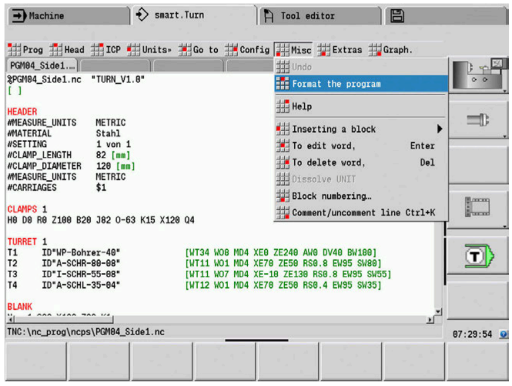
Misc menu item

The **Misc** (Miscellaneous) menu item provides the following functions:

- **Format the program**
  While checking the NC program, the control performs the following functions:
  - Adding of missing block numbers
  - Renumbering of NC blocks
  - Adding of missing indentations
  - Displaying of error message for detected syntax errors

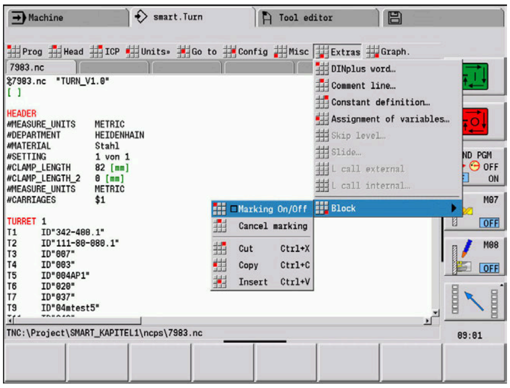
  > **i** You need to correct the syntax errors manually. Then you should use the **Format the program** function to re-check any corrections you have made.

- **Help**: The control opens TURNguide.
- **Inserting a block**:
  - **W/o block no. Alt-N**: The editor inserts an empty line at the cursor position
  - **With block no. Ins**: The editor inserts an empty line at the cursor position with a block number. Alternative: When you press the **INS** key, the editor inserts a block with a block number
  - **Comment at line end**: The editor inserts a comment at the end of the line in which the cursor is located.
- **To edit word, Enter**: You can edit the NC word at which the cursor is located.
- **To delete word, Del**: The editor deletes the NC parameter at the cursor position
- **Dissolve UNIT**: Position the cursor to the first line of a unit before selecting this menu item. The editor removes the brackets around the unit. The unit dialog can no longer be used for this machining block, but you can edit the machining block as desired
- **Block numbering...**: The block numbering is based on the **starting block number** and **block-number increment** settings. The first NC block receives the starting block number and the block-number increment is added for each further NC block. The settings for starting block number and block-number increment are specific to every NC program
- **Comment/uncomment line Ctrl+K**: You can hide the NC block or the unit on which the cursor is placed. The control will skip commented NC blocks.



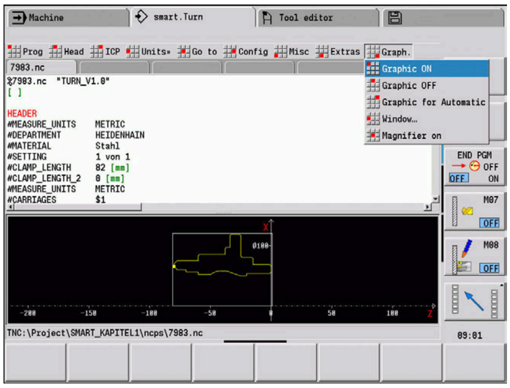
Extras menu item

The **Extras** menu item contains the following functions:



- **DIN PLUS word...**: The editor opens the selection list with all **DIN PLUS words** in alphabetical order. Select the desired instruction for program structuring or the input/output command. The editor inserts the **DIN PLUS word** at the cursor position
- **Comment line...**: The comment is inserted above the position of the cursor
- **Constant definition...**: The expression is inserted above the position of the cursor. If the **DIN PLUS word CONST** is not present yet, it will also be inserted
- **Assignment of variables...**: Inserts a variable instruction.
- **L call external** (the subprogram is in a separate file): The editor opens the file selection window for subprograms. Select the subprogram and fill out the subprogram dialog. The control searches for subprograms in the following sequence: current project, standard directory, and finally machine manufacturer directory
- **L call internal...** (the subprogram is contained in the main program): The editor opens the subprogram dialog
- **Block** functions. This menu item contains functions for marking, copying, and deleting sections
  - **Marking On/Off**: Activates/deactivates the marking mode during cursor movement
  - **Cancel marking**: After calling the menu item, no part of the program is marked
  - **Cut Ctrl+X**: Deletes the marked part of the program and copies it to the clipboard
  - **Copy Ctrl+C**: Copies the marked part of the program into the clipboard
  - **Insert Ctrl+V**: Inserts the contents of the clipboard at the cursor position. Any parts of the program that are marked will be replaced by the contents of the clipboard

Graph. menu item



The **Graph.** menu item provides the following functions:

- **Graphic ON**: Activates the graphic window or updates the displayed contour. As an alternative, you can use the soft key
- **Graphic OFF**: Closes the graphic window
- **Graphic for Automatic**: The graphic window is activated when the cursor is located in the contour description.
- **Window...**: Sets the graphic window. During editing, the control displays the programmed contours in up to four graphic windows. Set the desired windows
- **Magnifier on**: Activates the zoom function. As an alternative, you can use the soft key

The graphic window:

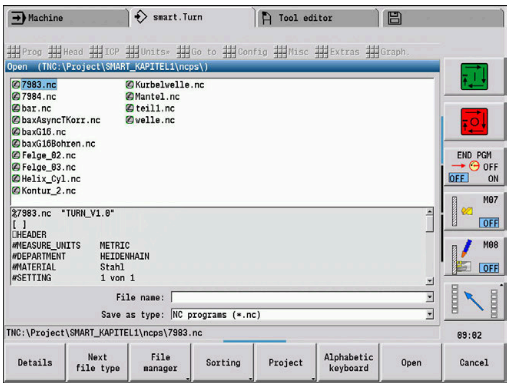
- Colors in contour graphics:
  - White: **Workpiece blank** and **Aux. workpiece blank**
  - Yellow: **Finished part**
  - Blue: **Auxiliary contour**
  - Red: contour element at the current cursor position. The arrow point indicates the direction of definition.
- When programming fixed cycles, you can use the displayed contour for establishing block references.
- Using the zoom functions, you can magnify, reduce or shift details
- If you are working with multiple contour groups, the control shows the number of the contour group at the top left in the graphic window

> **i**
> - Additions/changes to the contour will not be considered until **Graph.** is pressed again
> - Unambiguous NC block numbers are a prerequisite for the contour display

Soft keys with active program window	
	Activates the contour display and starts redrawing the contour
	Opens the soft-key menu for the zoom functions and displays the zoom frame

Sorting and file organization

When an NC program is opened or when a new NC program is created, the soft-key row is switched to the sorting and organization functions. Use the soft keys to select the order in which the programs are to be displayed, or use the functions for copying, deleting, etc.



File manager soft keys

Paths / files	Alternate between directory and file window
Cut out	Cut marked file
Copy	Copy marked file
Insert	Insert the file stored in the buffer memory
Rename	Rename marked file:
Delete all	Delete the marked file following confirmation prompt; program block display must not be open in any operating mode
Back	Return to program selection dialog

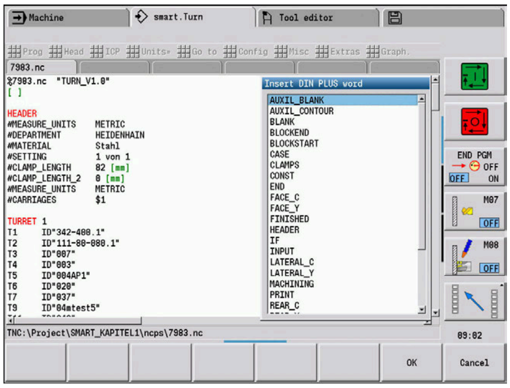
Other soft keys

Details	Display details
Mark all	Mark all files
Update	Update the marked program
Write protection	Activate/deactivate write protection for the marked program
Alphabetic keyboard	Open the **Alphabetic keyboard**.
Back	Return to program selection dialog

Soft keys for sorting

Details	Displays the file attributes: size, date, time
Sort by file name	Sorts by file name
Sort by size	Sorts by file size
Sort by date	Sorts by creation date or change date
Update	Updates the marked program
Reverse sorting	Reverses the sorting direction
Back	Return to program selection dialog

3.3 Program section code



A new NC program already contains some section codes. You can add new codes or delete existing ones, depending on your program requirements. An NC program must contain at least the **MACHINING** and **END** codes.

Further program section codes are available in the **DIN PLUS word...** list box (menu item **Extras** > **DIN PLUS word...**). The control enters the program section code at the correct position or at the current position.

German program section codes are used when German is set as the conversational language. All other languages use English program section codes.

> If you create NC programs externally or edit them in **Text editor (character)** input mode, unknown words may result from typing errors (e.g., **THN** instead of **THEN**).
>
> In the NC mode of the editor, the control checks the DIN PLUS words and the information in the program head. The editor highlights lines containing unknown words in magenta. When you simulate or run the NC program, the control will indicate an error.

Example: Program section codes

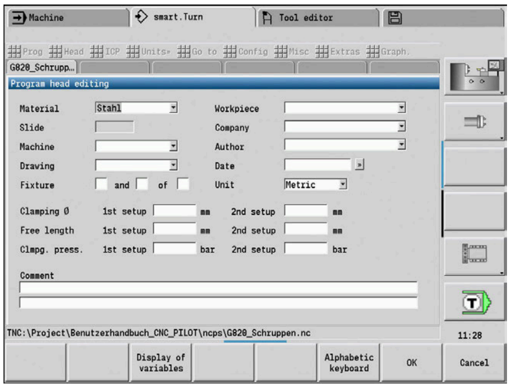
. . .	
BLANK	
N1 G20 X100 Z220 K1	
FINISHED PART	
N2 G0 X60 Z0	
N3 G1 Z-70	
. . .	
FACE_C Z-25	
N31 G308 ID“01“ P-10 O1	
N32 G402 Q5 K110 A0 Wi72 V2 XK0 YK0	
N33 G300 B5 P10 W118 A0	
N34 G309	
FACE_C Z0	
N35 G308 ID“02“ P-6 O1	
N36 G307 XK0 YK0 Q6 A0 K34.641	
N37 G309	
. . .	

Overview of program section codes

Meaning	DIN PLUS word	Description
Program head		
Program head	**PROGRAM HEAD**	Page 82
Chucking equipment	**CHUCKING EQUIP-MENT**	Page 84
Turret	**TURRET**	Page 85
Magazine	**MAGAZINE**	Page 85
Manual tool	**MANUAL TOOL**	Page 85
Contour definition		
Contour group	**CONTOUR GROUP**	Page 85
Workpiece blank	**BLANK**	Page 86
Finished part	**FINISHED PART**	Page 86
Auxiliary contour	**AUXIL_CONTOUR**	Page 86
Aux. workpiece blank	**AUX. BLANK**	Page 86
C-axis contours		
Front	**FRONT**	Page 86
REAR SIDE	**REAR SIDE**	Page 86
Lateral	**LATERAL**	Page 86
Y-axis contours		
Face Y	**FRONT Y**	Page 86
REAR SIDE Y	**REAR SIDE Y**	Page 86
Lateral surface in Y	**SURFACE Y**	Page 87
Workpiece machining		
Machining operation	**MACHINING**	Page 88
End	**END**	Page 88
Subprograms		
Subprogram	**SUBROUTINE**	Page 88
Return	**RETURN**	Page 88
Others		
CONST	**CONST**	Page 89
VAR	**VAR**	Page 89
SLIDE ALLOCATION	**SLIDE ALLOCATION**	Page 90

If there is more than one independent contour definition for drilling and milling, use the program section codes (**FRONT**, **LATERAL**, etc.) for each of them.

PROGRAM HEAD section



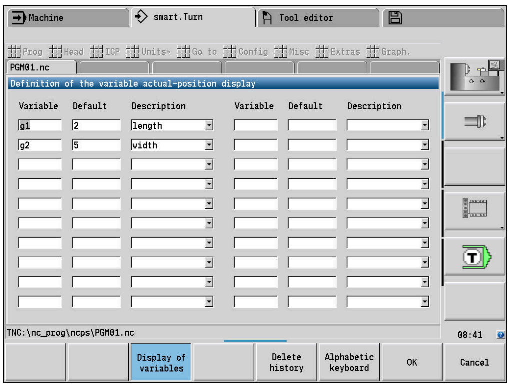
Instructions and information in the **PROGRAM HEAD**:

- **Unit**:
  - Select dimensional system in millimeters or inches
  - No entry: The unit set in the machine parameter is used
- The other fields contain **organizational information** and **set-up information**, which do not influence the machining process

Information contained in the program head is preceded by **#** in the NC program.

> You can select the **Unit** only when creating a new NC program. It is not possible to post-edit this entry.

Display of variables



To open the variable display in the **PROGRAM HEAD**, proceed as follows:

- ▶ Press the **Display of variables** soft key
- > The control opens the **Definition of the variable actual-position display** form.

You can define up to 20 variables. In **Program run** submode or **Simulation** submode, you define whether the variables are displayed during program run.

> Only use g variables:
> - g1 to g299 are freely available to the user
> - g5xx are reserved for the machine tool builder
> - g810 to g815 are used in measuring cycles
> - g950 to g955 are used for structure programming

For each variable you define the following:

- **Variable** – variable number
- **Default** – initialization value
- **Description** – text for displaying and requesting the variable during program run or simulation (max. 20 characters)

> Only global variables are currently supported.
> **Further information:** "Variable types", Page 515

Delete history

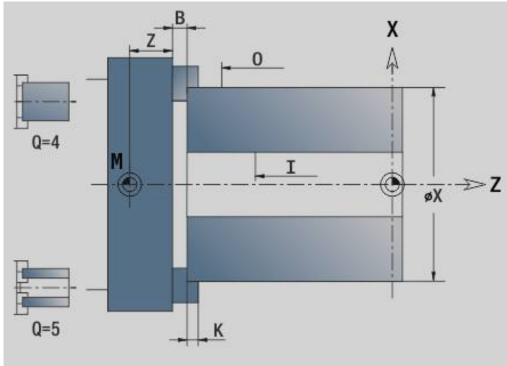
When **PROGRAM HEAD** is open, the **Delete history** soft key is available.

If you press the **Delete history** soft key, all old entries are deleted from the pull-down menu. The current entry is retained.

The following entries are deleted:

- Machine
- Drawing
- Workpiece
- Company
- Author
- Description of variables

CHUCKING EQUIPMENT section



In the **CHUCKING EQUIPMENT** program section, you can describe how the workpiece is clamped. This makes it possible to display the chucking equipment in the **Simulation** submode. In **TURN PLUS**, the chucking equipment information is used to calculate the datums and cutting limits during automatic program generation.

Parameters:

1 **H**: **No. of clamping**
2 **D**: **Spindle number AWG**
3 **R**: **Clamp type**
   - **0: J=Free length**
   - **1: J=Fixing length**
4 **Z**: **Edge of chuck** – position of chuck edge
5 **B**: **Chuck jaw reference**
6 **J**: **Unclamping length** – clamping or free length of the workpiece (depends on the **Clamp type R**)
7 **O**: **Cutting limit, outside** – cutting limit for outside machining
8 **I**: **Cutting limit, inside** – cutting limit for inside machining
9 **K**: **Overlap jaw/workpiece** (pay attention to sign!)
10 **X**: **Chuck diameter** of workpiece blank
11 **Q**: **Chuck form**
   - **5: Outside chucking**
   - **5: Inside chucking**
12 **V**: **Shaft machining AWG**
   - **0: Chuck** – automatic separation points at largest and smallest diameter
   - **1: Shaft/chuck** – machining also starting from the chuck
   - **2: Shaft/face driver** – outside contour can be machined completely

> If you do not define the parameters **Z** and **B**, **TURN PLUS** will use the following machine parameters during **AWG** (automatic working plan generation) submode:
> - Front Edge of chuck on spindle or counter spindle
> - Jaw width on spindle / counterspindle

Further information: User's Manual

TURRET / MAGAZINE section

The **TURRET** or **MAGAZINE** program section defines the assignment of the tool carrier. For every assigned pocket, the tool ID number is entered. For multipoint tools, every cutting edge is entered in the turret list.

> If you do not program the **TURRET** or **MAGAZINE**, the tools entered in the tool list of the **Machine** operating mode will be used.

Example: Turret table

. . .	
TURRET	
T1 ID"342-300.1"	
T2 ID"C44003"	
. . .	

Example: Magazine table

. . .	
MAGAZINE	
ID"342-300.1"	
ID"C44003"	
. . .	

MANUAL TOOL section

The **MANUAL TOOL** program section defines a usage list for manually changed tools.

This program section is only required if you use the automatic working plan generation (AWG) function on a machine equipped with a Multifix tool holder. The control will use these tools in connection with the AWG function.

During NC program generation, the control checks if this list only contains manually changed tools and issues an error message, if this is not the case.

Contour group section

In this program section, you describe the position of the workpiece in the working space.

The control supports up to four contour groups (**Workpiece blank**, **Finished part** and **Auxiliary contours**) in one NC program. The **Contour group** section code initiates the description of a contour group. **G99** assigns the machining operations to a contour group.

Parameters:

- **Q**: number of the **Contour group**
- **X**: **Contour pos. in graphic**
- **Z**: **Contour pos. in graphic**
- **V**: **Position**
  - **0**: Machine coordinate system
  - **2**: Mirrored coordinate system (opposite to Z direction)

BLANK section

In this program section, you describe the contour of the workpiece blank.

FINISHED PART section

In this program section, you describe the contour of the finished part. After the **FINISHED PART** section, you can use additional section codes such as **FRONT**, **LATERAL** etc.

AUX. BLANK section

In this program section you define additional workpiece blanks, which can be activated with **G702** when required.

AUXILIARY CONTOUR section

In this program section, you describe the auxiliary turning contours.

FRONT, REAR SIDE section

In this program section you describe the front and rear side contours to be machined with the C axis. The program section defines the position of the contour in Z direction.

Parameters:

- **Z**: **Position** of the front-face or rear-side contour

LATERAL section

In this program section you describe the lateral surface contours to be machined with the C axis. The program section defines the position of the contour in X direction.

Parameters:

- **X**: **Reference diameter** of lateral surface contours

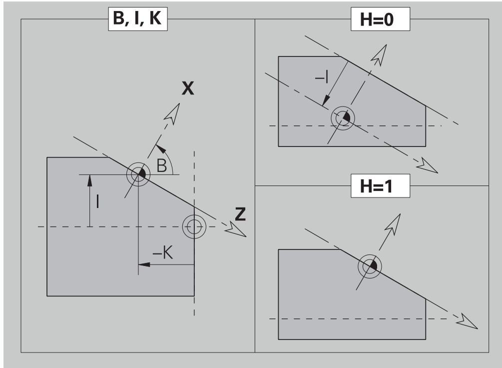
FRONT Y / REAR SIDE Y section

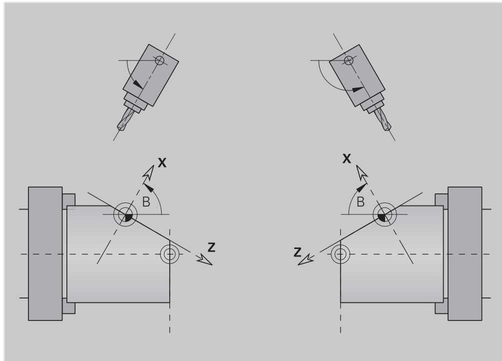
For lathes with Y axis, these program section codes define the XY plane (**G17**) and the position of the contour in Z direction. The **Spindle angle** (**C**) defines the spindle position.

Parameters:

- **X**: **Limit diameter** – area diameter as cutting limit
- **Z**: **Reference dimension** or **Position** – position of the reference plane (default: 0)
- **C**: **Spindle angle** or **Angle** (default: 0)

SURFACE Y section





The section code identifies the YZ plane (**G19**). For machines equipped with a B axis, it defines the tilted plane.

Without tilted plane: The reference diameter defines the contour position in the X direction; the C axis angle defines the position on the workpiece.

Parameters:

- **X**: **Reference diameter**
- **C**: **C axis angle** – defines the spindle position

With tilted plane: **SURFACE Y** additionally performs the following transformations and rotations for the tilted plane:

- Shifts the coordinate system to the position **I**, **K**
- Rotates the coordinate system by the **Plane angle B**; **Plane ref. in X**, **Plane ref. in Z**: **I**, **K**
- **H=0**: Shifts the rotated coordinate system by **-I**. The coordinate system is moved back

Parameters:

- **X**: **Reference diameter**
- **C**: **C axis angle** – defines the spindle position
- **B**: **Plane angle** (reference: positive Z axis)
- **I**: **Plane ref. in X** (radius value)
- **K**: **Plane ref. in Z**
- **H**: **Automatic shift** – automatic shift of the coordinate system (default: 0)
  - **0: Shift by -I** – the rotated coordinate system is shifted by **-I**
  - **1: Do not shift** – the coordinate system is not shifted

Shifting back coordinate system: The control evaluates the reference diameter for the cutting limit. This value is also used as the reference value for the depth that you program for drilling operations and milling contours.

Since the **Reference diameter** is referenced to the current datum, it is recommended, when working in a tilted plane, to shift the rotated coordinate system back by the distance **-I**. If the cutting limits are not needed, for example for drilling holes, you can disable the shift of the coordinate system (**H = 1**) and set the **Reference diameter** to 0.

> Please note:
>
> - X is the infeed axis in a tilted coordinate system. X coordinates are entered as diameter coordinates.
> - Mirroring the coordinate system has no effect on the reference axis of the tilt angle (B axis angle of the tool call)

Example: SURFACE Y

PROGRAM HEAD	
...	
CONTOUR Q1 X0 Z600	
BLANK	
...	
FINISHED PART	
...	
SURFACE Y X118 C0 B130 I59 K0	
...	
MACHINING	
...	

MACHINING section

In the **MACHINING** program section, you can program the required machining operations. This code is **mandatory**.

END code

The **END** code concludes the NC program. This code **must** be included.

SUBROUTINE section

If you define a subprogram within your NC program (within the same file), it is identified by **SUBROUTINE**, followed by the name of the subprogram (max. 40 characters).

RETURN code

The **RETURN** code concludes the subprogram.

CONST code

In the **CONST** section of the program you define constants. You use constants for the definition of a value.

You enter the value directly or you calculate it. If you use constants in the calculation you must first define them.

The length of the constant name must not exceed 20 characters. Lower case letters and numbers are allowed. Constants always begin with an underscore.

Further information: "Expanded variable syntax CONST – VAR", Page 529

Example: CONST

```
CONST
_nvr = 0
_sd=PARA("","CfgGlobalTechPara", "safetyDistWorkpOut")
_nws = _sd-_nvr
. . .
BLANK
N 1 G20 X120 Z_nws K2
. . .
MACHINING
N 6 G0 X100+_sd
. . .
```

VAR code

In the **VAR** program section, you assign names (descriptive text) to variables.

Further information: "Expanded variable syntax CONST – VAR", Page 529

The length of the variable name must not exceed 20 characters. Lower case letters and numbers are allowed. Variables always begin with **#**.

Example: VAR

```
VAR
#_inside_dm = #l2
#_length = #g3
. . .
BLANK
N 1 #_length=120
N 2 #_inside_dm=25
N 3 G20 X120 Z#_length+2 K2 I#_inside_dm
. . .
MACHINING
. . .
```

SLIDE ALLOCATION code

Refer to your machine manual.
This function is only available on a machine with multiple channels (option 153).

The **SLIDE ALLOCATION** code assigns the subsequent machining steps to the specified slide(s). If you specify more than one slide, the control performs the machining operation on each of the specified slides.

Parameters:

- **Slide**: Slide numbers

To reset the assignment, program **SLIDE ALLOCATION** again without entering a slide code. The control will then use all slides again as specified in the program header.

If you program a slide code in the NC block, the slides entered with a $ ... in the NC block will be effective.

3.4 Tool programming

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

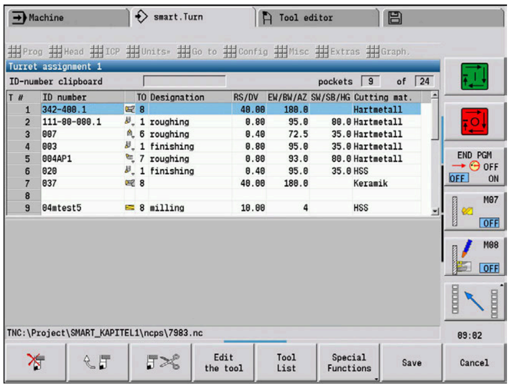
The designations of the tool pockets are fixed by the machine tool builder. Each tool holder has a unique **tool number**.

In the **T command** (**MACHINING** section), you can program the tool number, and therefore the position to which the tool carrier rotates. The control retrieves the assignment of the tools to the swivel position from the turret list of the **TURRET** section.

You can edit the tool entries individually, or you can call the turret list via the **Set up the turret list** menu item and then edit it.

Setting up a turret list

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

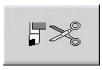


With the **Set up the turret list** function, the control provides the turret assignment as a tool list for editing.

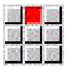
You have the following options:

- Editing the turret assignment: Transfer tools from the database, delete entries or move them to other positions
- Load the turret list from the **Machine** mode of operation
- Deleting the current turret assignment of the NC program

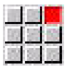
Soft keys in turret list

	Delete entry
	Paste entry from clipboard
	Cut out entry and save it in the clipboard
Tool List	Show entries in the tool database
Save	Save the turret assignment
Cancel	Close the tool list. You decide whether the changes made remain in effect
Edit the tool	The input window of the selected tool is opened for editing
Transfer machine	Load the turret list from the **Machine** mode of operation

Loading the turret list from the **Machine** mode of operation:



- Select the **Head** menu item

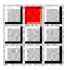


- Select the **Set up the turret list** menu item

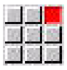
- Switch to **Special Functions** if needed

- Load the tool list from the **Machine** mode of operation into the NC program

Deleting the turret list:



- Select the **Head** menu item



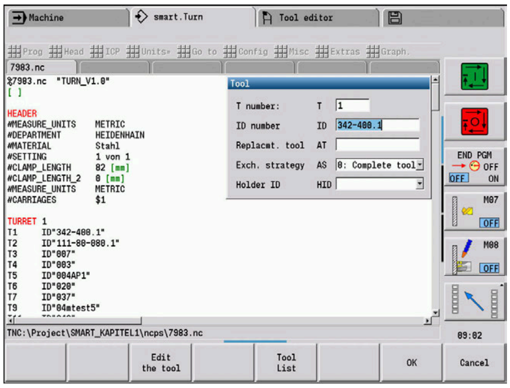
- Select the **Set up the turret list** menu item

- Switch to **Special Functions**

- Delete all entries of the turret list

Editing tool entries

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



For each entry in the **TURRET** section, call the **Tool** dialog box, enter the **ID no.**, or use the **ID no.** from the tool database.

Parameters of the **Tool** dialog box:

- **T**: **T number** – Position on the tool carrier
- **ID**: **ID number** – Reference to database
- **AT**: **Replacement tool** – Identification number of the tool to be used when the previous tool is worn out
- **AS**: **Replacement strategy**
  - **0: Complete tool**
  - **1: Secondary cutting edge or any**

Creating a new tool entry:

↓
- ▶ Position the cursor

INS
- ▶ Press the **INS** key
- > The editor opens the **Tool** dialog box.
- ▶ Enter the **ID no.** of the tool

Tool List
- ▶ Open the tool database

↓
- ▶ Place the cursor on the tool to be loaded.

Load tool
- ▶ Use the **ID no.** of the tool

Editing the tool data:

↓
- ▶ Position the cursor

ENT
- ▶ Press the **ENT** key
- ▶ Edit the **Tool** dialog box

Multipoint tools

A multipoint tool is a tool with multiple reference points or multiple cutting edges. During **T** call, the tool number is followed by an **.S** to identify the cutting edge.

Tool number.S (S=0..9)

S=0 identifies the main cutting edge. This does not need to be programmed.

Examples:

- **T3** or **T3.0**: Tilted position 3; main cutting edge
- **T12.2**: Tilted position 12; cutting edge 2

Replacement tools

During **simple** tool life monitoring the program run is stopped when a tool is worn out. However, the program run is then resumed and concluded.

If you use the **tool life monitoring with replacement tools** function (option 10), the control automatically inserts the replacement tool when a certain tool is worn out. The control does not stop the program run until the last tool of the tool change sequence is worn out.

You can define replacement tools when setting up the turret. The interchange chain can contain more than one replacement tool. The interchange chain is a part of the NC program. In the **T** commands, you program the **first tool** to be changed.

Defining replacement tools:

↓ ▶ Place the cursor on the previous tool

ENT ▶ Press the **ENT** key

▶ Enter the **ID no.** of the replacement tool (**Tool** dialog box)

▶ Define the replacement strategy

When using multipoint tools, you define in the replacement strategy whether the complete multipoint tool or only the worn-out cutting edge of the tool is to be replaced by a replacement tool:

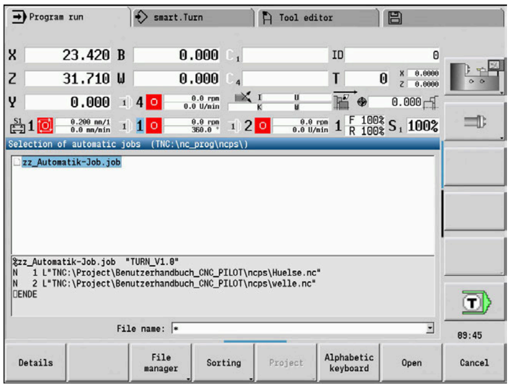
- **0: Complete tool** (default): If a cutting edge of the multipoint tool is worn out, the tool will no longer be used
- **1: Secondary cutting edge or any**: Only the cutting edge of the multipoint tool that is worn out will be replaced by another tool or another cutting edge. Any other cutting edges of the multipoint tool that are not worn out will continue to be used.

3.5 Automatic job

In **Program run** submode, the control can execute several main programs successively without you needing to reselect and start these programs. For this purpose you create a program list (Automatic jobs) that is run in **Program run** submode.

Enter the quantity, i.e. the number of repetitions, for each main program.

All program calls are saved with the complete path. In this way, you can also start project-dependent programs.



Opening a job

In **smart.Turn** operating mode, you create an automatic job with the file extension **.job**. **Automatic jobs** are project-independent and always stored in the default directory **TNC:\nc_prog_ncps**.

Creating a new automatic job:

- Select the **Prog** menu item
- Select the **New** menu item
- Select the **New automatic job** menu item
- Enter the file name.

Save

- Press the **Save** soft key

Opening an existing automatic job:

- Select the **Prog** menu item
- Select the **Open ...** menu item

Next file type

- Switch to the **.job** file type

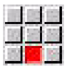
Open

- Press the **Open** soft key

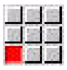
Editing a job

In an automatic job, you link main programs in order to execute them successively in **Program run** submode.

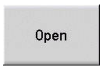
Creating a new automatic job:



- Select the **Extras** menu item



- Select the **Program call** menu item
- Select the main program.



- Press the **Open** soft key
- Enter the number of repetitions in the **Q** parameter, if required

> If you do not program a repetition the control runs the program once. The program is not run if you enter a 0.

Example: Automatic job

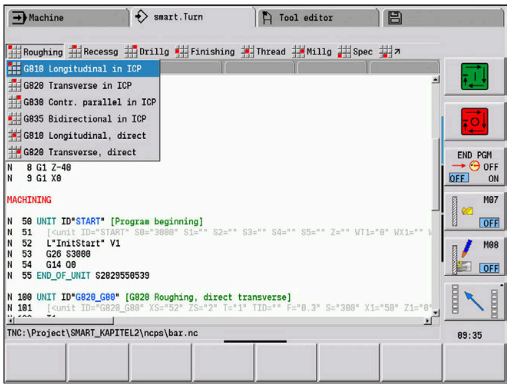
%autorun.job „TURN_V1.0“	
N1 L“TNC:\nc_prog\ncps\234.nc“ Q3	
N2 L“TNC:\Project\Project3\ncps\10785.nc“	
N3 L“TNC:\nc_prog\ncps\Huelse.nc“ Q12	
. . .	

4

smart.Turn Units (Option 9)

4.1 Units – smart.Turn units

Units menu item

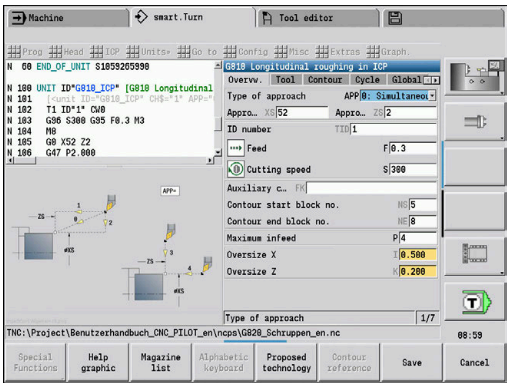


The **Units»** menu item contains the unit calls grouped by the type of machining operation. Select the **Units»** menu item to call the following menu items:

- **Roughing**
- **Recessg**
- **Drilling** (C axis and Y axis)
- **Finishing**
- **Thread**
- **Millg** (C axis and Y axis)
- **Spec** (special operations)

> Refer to your machine manual.
> Your machine tool builder can provide proprietary units. Select the **Spec** menu item to call these functions.

smart.Turn unit



A unit describes a complete working block.

The unit includes:

- Tool call
- Technology data
- Cycle call
- Approach and departure strategies
- Global data
- Safety clearance

These parameters are summarized in a dialog—simply and clearly.

Unit forms

The unit dialog is divided into fillable forms and the forms are divided again into groups. You can navigate between the forms and groups with the smart.Turn keys.

Forms in unit dialogs

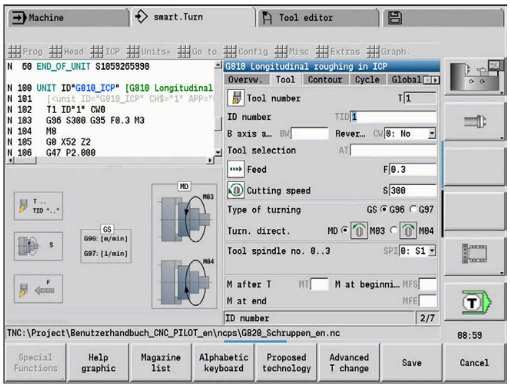
Fillable form	Function
Overvw.	Overview form with all necessary settings
Tool	Tool form with tool selection, technological settings and **M** functions
Contour	Description or selection of the contour to be machined
Cycle	Description of the machining operation
Global	View and settings of globally set values
AppDep	Definition of approach and departure behavior
Tool ext	Extended tool settings

Overview form

The overview form summarizes the most important settings of the unit. These parameters are repeated in the other forms.

Tool form

You program the technological information in this form.



Tool:

- **T**: **Tool number** – turret pocket number
- **TID**: **ID no.** – the tool name will be entered automatically
- **F**: **Feed** – feed per revolution (mm/rev) for machining
  The tool is moved by the programmed value at each spindle revolution.
- **S**: **Cutting speed** (m/min) or **Constant speed** (rev/min)
  Switchable with **Type of turning GS**.

Spindle:

- **GS**: **Type of turning**
  - **G96**: **Constant Cutting speed**
    The rotational speed changes in sync with the turning diameter.
  - **G97**: **Constant speed**
    Rotational speed is independent of the turning diameter.
- **MD**: **Turn. direct.**
  - **M03**: Clockwise (CW)
  - **M04**: Counterclockwise (CCW)
- **SPI**: **Workpc spindle no. 0..3** – spindle that is holding the workpiece (only on machines with more than one spindle)
- **SPT**: **Workpc spindle no. 0..3** – spindle of the driven tool

M functions:

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

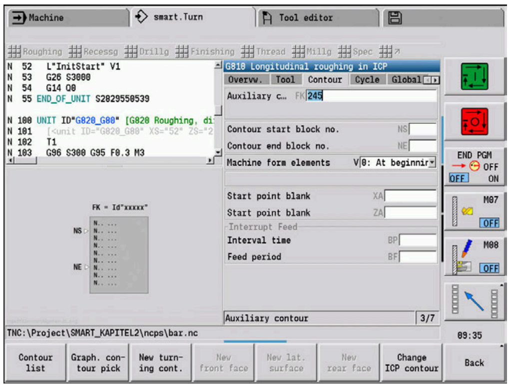
> A machining operation is assigned to each unit for access to the technology database. The following description shows the assigned machining mode and the unit parameters that were changed by the technology proposal.

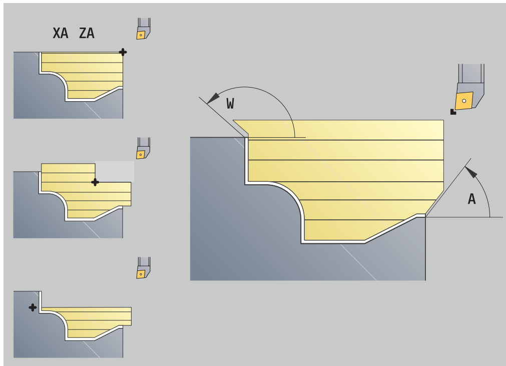
Soft keys in the Tool form

Turret list	Selects the tool number
Proposed technology	Loads the feed rate, cutting speed and infeed from the technology database

Contour form

In the contour form, you can define the contours to be machined. The control distinguishes between the direct contour definition (**G80**) and the reference to an **external** contour definition (**FINISHED PART** or **AUXILIARY CONTOUR** section).





ICP contour definition

- **FK**: **Auxiliary contour** – name of the contour to be machined
  You can select an existing contour or describe a new contour with **ICP**.
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
  - **NE** not programmed: The contour element **NS** is machined in the direction of contour definition
  - **NS** = **NE** programmed: The contour element **NS** is machined opposite to the direction of contour definition.
- **V**: **Machine form elements** (default: 0)

  A chamfer/rounding arc is machined
  - **0: At beginning and end**
  - **1: At beginning**
  - **2: At end**
  - **3: No machining**
  - **4: Only chamfer/rounding** is machined—not the basic element (requirement: the contour section consists of a single element)
- **BP**: **Interval time** – time during which the feed motion will be interrupted
  The chip is broken by the intermittent feed.
- **BF**: **Feed period** – time interval until the next break
  The chip is broken by the intermittent feed.

> The listed soft keys are only selectable if the input cursor is in the **FK** field, or on **NS** or **NE**.

Soft keys in the ICP contour form

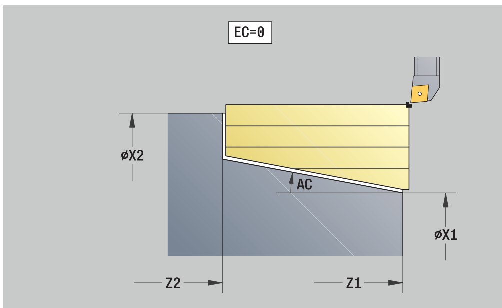
Contour list	Opens the selection list of the contours defined in the program
Graph. con- tour pick	Shows all defined contours in the graphics window. Use the arrow keys, touch gestures or a mouse for selection.
New turn- ing cont.	Starts the **ICP editor** submode. First, enter the desired contour name in **FK**.
Change ICP contour	Starts the **ICP editor** submode with the currently selected contour
Contour reference	Opens the graphics window for selection of a part of a contour for **NS** and **NE.** Use the arrow keys, touch gestures or a mouse for selection.
New front face	Starts the **ICP editor** submode. First, enter the desired contour name in **FK**
New lat. surface	Starts the **ICP editor** submode. First, enter the desired contour name in **FK**

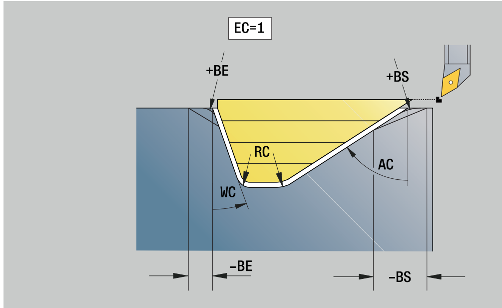
Navigating between contours

When working with multiple contour groups, you can press the **Contour reference** soft key to select the correct contour. The control shows the number of the **Contour group** and, if applicable, the name of the **Auxiliary contour** at top left in the graphic window.

Navigation keys

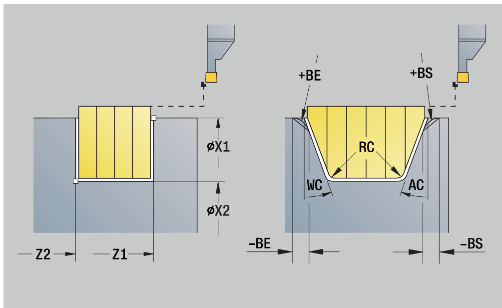
↓ ↑	Switch to the next or previous contour (**Contour group**/**Workpiece blank**/**Auxiliary contour**/**Finished part**)
← →	Switches to the next contour element
PG DN	Reduces the displayed workpiece (zoom –)
PG UP	Magnifies the displayed workpiece (zoom +)





Direct contour definition parameters for turning operations:

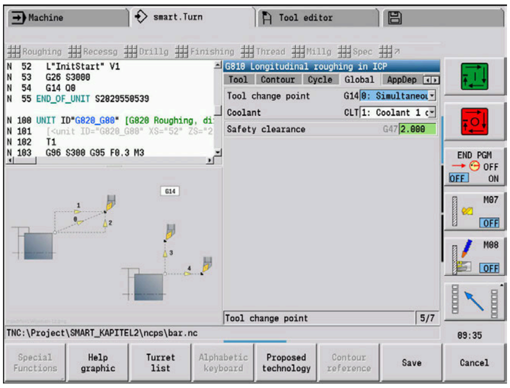
- **EC**: **Type of contour**
  - **0: Normal contour**
  - **1: Plunging contour**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **RC**: **Rounding** – radius of contour corner
- **AC**: **Start angle** – angle of the first contour element (range: 0°< **AC** < 90°)
- **WC**: **Final angle** – angle of the last contour element (range: 0° < **WC** < 90°)
- **BS**: **-Chamfer/+radius at start**
  - **BS** > 0: Radius of rounding arc
  - **BS** < 0: Width of chamfer
- **BE**: **-Chamfer/+radius at end**
  - **BE** > 0: Radius of rounding arc
  - **BE** < 0: Width of chamfer
- **BP**: **Interval time** – time during which the feed motion will be interrupted
  The chip is broken by the intermittent feed.
- **BF**: **Feed period** – time interval until the next break
  The chip is broken by the intermittent feed.



Direct contour definition parameters for recessing operations:

- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **RC**: **Rounding** – radii in the recess base
- **AC**: **Start angle** – angle of the first contour element (range: 0°< **AC** < 90°)
- **WC**: **Final angle** – angle of the last contour element (range: 0° < **WC** < 90°)
- **BS**: **-Chamfer/+radius at start**
  - **BS** > 0: Radius of rounding arc
  - **BS** < 0: Width of chamfer
- **BE**: **-Chamfer/+radius at end**
  - **BE** > 0: Radius of rounding arc
  - **BE** < 0: Width of chamfer

Global form



This form contains parameters that were defined as default values in the start unit. You can edit these parameters in the machining units.

Parameters:

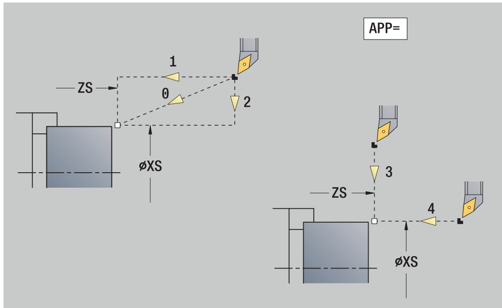
- **G14**: **Tool change point**
  - **No axis**
  - **0: Simultaneously**
  - **1: First X, then Z**
  - **2: First Z, then X**
  - **3: Only X**
  - **4: Only Z**
  - **5: Only Y** (machine-dependent)
  - **6: Simultaneous w/ Y** (machine-dependent)
- **CLT**: **Coolant**
  - **0: Without**
  - **1: Coolant 1 on**
  - **2: Coolant 2 on**
- **G47**: **Safety clearance** – indicates the distance to the current workpiece blank up to which the tool is not moved at rapid traverse during turning operations
- **SCK**: **Safety clearance** in infeed direction during drilling and milling operations
- **SCI**: **Safety clearance** in the machining plane during drilling and milling operations
- **G60**: **Protection zone** – protection zone monitoring during drilling
  - **0: Active**
  - **1: Inactive**

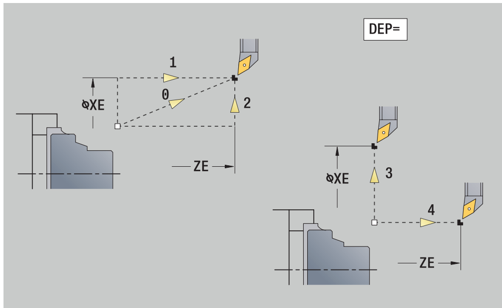
Programming notes:

- If, on controls without a configured Y axis, you set **G14** to the default setting **5: Only Y** or **6: Simultaneous w/ Y**, then the control will use **No axis** or **0: Simultaneously**.
- The **G840**, Contour milling, figures, and **G84X**, Pocket milling, figures units additionally provide the **Return plane RB** parameter as part of the **Global** form.

AppDep form

Positions and variants of the approach and departure movements are defined in this form.





Influence the approach strategy with the following parameters.

Approach:

- **APP**: **Type of approach**
  - **No axis** – switch off the approach function
  - **0: Simultaneously** – X and Z axes approach diagonally
  - **1: First X, then Z**
  - **2: First Z, then X**
  - **3: Only X**
  - **4: Only Z**
- **XS**, **ZS**: **Approach position X** and **Z** – position of the tool tip before cycle call

Additionally with C-axis operations:

- **CS**: **Approach position C** – C-axis position that is approached before cycle call with **G110**

Approach with Y axis:

- **APP**: **Type of approach**
  - **No axis** – switch off the approach function
  - **0: Simultaneously** – X and Z axes approach diagonally
  - **1: First X, then Z**
  - **2: First Z, then X**
  - **3: Only X**
  - **4: Only Z**
  - **5: Only Y**
  - **6: Simultaneous w/ Y** – X, Y and Z axes approach diagonally
- **XS**, **YS**, **ZS**: **Approach position X**, **Y** and **Z** – position of the tool tip before cycle call
- **CS**: **Approach position C** – C-axis position that is approached before cycle call with **G110**

Influence the departure strategy with the following parameters (also applies for Y-axis functions).

Departure:

- **DEP**: **Type of departure**
  - **No axis** – switch off the departure function
  - **0: Simultaneously** – X and Z axes depart diagonally
  - **1: First X, then Z**
  - **2: First Z, then X**
  - **3: Only X**
  - **4: Only Z**
- **XE**, **ZE**: **Departure position X** and **Z** – position of the tool tip before the movement to the tool change point

> The units **G890 ICP for contouring** and **G891 Simultaneous finishing** additionally provide the parameter **5: Simultaneous G1** in the function **DEP**.

Tool ext form

In this form you can program additional tool settings.

Tool:

- **T**: **Tool number** – turret pocket number
- **TID**: **ID no.** – the tool name will be entered automatically

B axis:

- **BW**: **B axis angle** – angle in the B axis (machine-dependent)
- **CW**: **Reverse the tool** (machine-dependent)
  - **0: No**
  - **1: Yes** (180°)

Misc. functions:

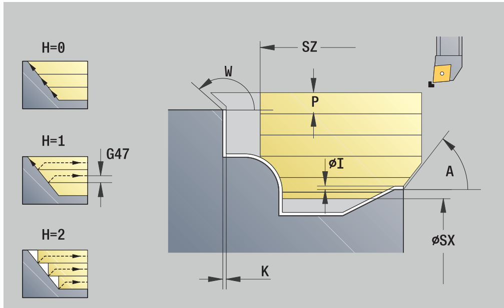
- **HC**: **Shoe brake** (machine-dependent)
  - **0:Automatic**
  - **1: Tighten**
  - **2: Don't tighten**
- **DF**: **Miscellaneous function** – can be evaluated by the machine tool builder in a subprogram (machine-dependent)
- **XL**, **YL**, **ZL**: Values can be evaluated by the machine manufacturer in a subprogram (machine-dependent)

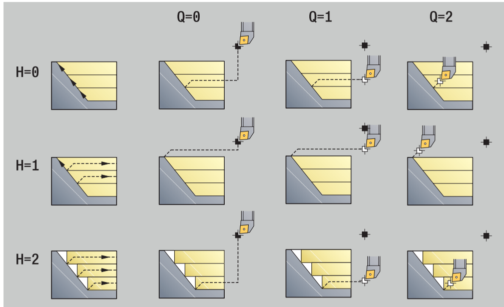
> With the **Advanced T change** soft key, you can switch quickly and easily between the **Tool** and **Tool ext** forms.

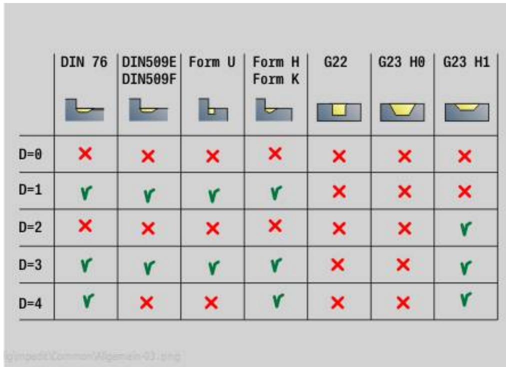
4.2 Units – Roughing

Unit G810 Longitudinal roughing in ICP

The unit machines the contour described in the **FINISHED PART** section from **NS** to **NE**. Any **Auxiliary contour** defined in **FK** will be used.







	DIN 76	DIN509E DIN509F	Form U	Form H Form K	G22	G23 H0	G23 H1
D=0	✗	✗	✗	✗	✗	✗	✗
D=1	✓	✓	✓	✓	✗	✗	✗
D=2	✗	✗	✗	✗	✗	✗	✓
D=3	✓	✓	✓	✓	✗	✗	✓
D=4	✓	✗	✗	✓	✗	✗	✓

Unit name: **G810_ICP** / cycle: **G810**

Further information: "Longitud. roughing G810", Page 356

Contour form:

- **RH**: **Wrkpc. blank contour** – evaluation only if no workpiece blank has been defined
  - **0:** (depends on defined parameters)
    - No parameters: workpiece blank from ICP contour and tool position
    - **XA** and **ZA**: workpiece blank from ICP contour and starting point of workpiece blank
    - **J**: workpiece blank from ICP contour and equidistant oversize
  - **1: From tool position** (workpiece blank from ICP contour and tool position)
  - **2: With blank start point** (workpiece blank from ICP contour and starting point of workpiece blank **XA** and **ZA**)
  - **3 Equidistant oversize** (workpiece blank from ICP contour and equidistant oversize **J**)
  - **4: Long.-transv. oversize** (workpiece blank from ICP contour, transverse oversize **XA** and longitudinal oversize **ZA**)
- **J**: **Workpiece blank oversize** (radius value – will only be evaluated if no workpiece blank has been defined)
- **XA**, **ZA**: **Start point blank** (Definition of the corner point of the workpiece blank contour—an evaluation will only take place if no workpiece blank has been defined beforehand)

Further information: "Contour form", Page 100

Cycle form:

- **I**, **K**: **O-size X** and **Z**
- **P**: **Maximum infeed**
- **E**: **Plunging behavior**
  - **E** = 0: Descending contours are not machined
  - **E** > 0: Plunging feed rate for descending contour elements. Descending contour elements are machined
  - No input: The plunging feed rate is reduced during machining of descending contour elements by up to 50 %. Descending contour elements are machined

- **SX**, **SZ**: **Cutting limit in X** and **Z** (default: no cutting limit; diameter value = **SX**)
- **A**: **Start angle** (reference: Z axis; default: parallel to Z axis)
- **W**: **Depart.angle** (reference: Z axis; default: orthogonal to Z axis)
- **Q**: **Kind of liber.** at end of cycle
  - **0: Back to beg., X before Z**
  - **1: Before finished contour**
  - **2: Retract by safety clear.**
- **H**: **Contour smoothing**
  - **0: With each cut** (within the infeed range)
  - **1: With the last cut** (entire contour) – retract at an angle below 45°
  - **2: No smoothing** – retract at an angle below 45°
- **D**: **Omit elements** (see figure)
- **U**: **Cut line on horiz. element**
  - **0: No** (uniform cutting segmentation)
  - **1: Yes** (irregular cutting segmentation, if applicable)
- **O**: **Hide undercutting**
  - **0: No**
  - **1: Yes**

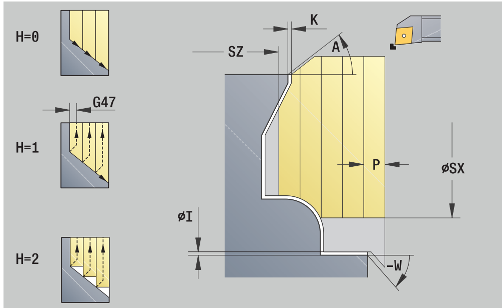
Further forms:

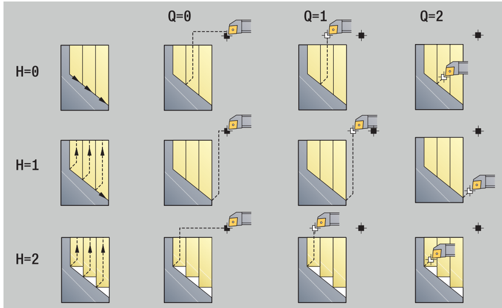
Further information: "smart.Turn unit", Page 98

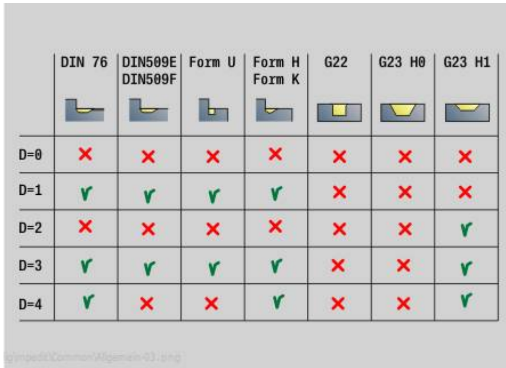
Access to the technology database:

- Machining operation: **Roughing**
- Affected parameters: **F**, **S**, **E**, **P**

G820 Roughing in ICP unit







	DIN 76	DIN509E DIN509F	Form U	Form H Form K	G22	G23 H0	G23 H1
D=0	✗	✗	✗	✗	✗	✗	✗
D=1	✓	✓	✓	✓	✗	✗	✗
D=2	✗	✗	✗	✗	✗	✗	✓
D=3	✓	✓	✓	✓	✗	✗	✓
D=4	✓	✗	✗	✓	✗	✗	✓

The unit machines the contour described in the **FINISHED PART** section from **NS** to **NE**. Any **Auxiliary contour** defined in **FK** will be used.

Unit name: **G820_ICP** / cycle: **G820**

Further information: "Face roughing G820", Page 359

Contour form:

- **RH**: **Wrkpc. blank contour** – evaluation only if no workpiece blank has been defined
  - **0:** (depends on defined parameters)
    - No parameters: workpiece blank from ICP contour and tool position
    - **XA** and **ZA**: workpiece blank from ICP contour and starting point of workpiece blank
    - **J**: workpiece blank from ICP contour and equidistant oversize
  - **1: From tool position** (workpiece blank from ICP contour and tool position)
  - **2: With blank start point** (workpiece blank from ICP contour and starting point of workpiece blank **XA** and **ZA**)
  - **3 Equidistant oversize** (workpiece blank from ICP contour and equidistant oversize **J**)
  - **4: Long.-transv. oversize** (workpiece blank from ICP contour, transverse oversize **XA** and longitudinal oversize **ZA**)
- **J**: **Workpiece blank oversize** (radius value – will only be evaluated if no workpiece blank has been defined)
- **XA**, **ZA**: **Start point blank** (Definition of the corner point of the workpiece blank contour—an evaluation will only take place if no workpiece blank has been defined beforehand)

Further information: "Contour form", Page 100

Cycle form:

- **I**, **K**: **O-size X** and **Z**
- **P**: **Maximum infeed**
- **E**: **Plunging behavior**
  - **E** = 0: Descending contours are not machined
  - **E** > 0: Plunging feed rate for descending contour elements. Descending contour elements are machined
  - No input: The plunging feed rate is reduced during machining of descending contour elements by up to 50 %. Descending contour elements are machined
- **SX**, **SZ**: **Cutting limit in X** and **Z** (default: no cutting limit; diameter value = **SX**)
- **A**: **Start angle** (reference: Z axis; default: orthogonal to Z axis)
- **W**: **Depart.angle** (reference: Z axis; default: parallel to Z axis)
- **Q**: **Kind of liber.** at end of cycle
  - **0: Back to beg., X before Z**
  - **1: Before finished contour**
  - **2: Retract by safety clear.**
- **H**: **Contour smoothing**

  - **0: With each cut** (within the infeed range)
  - **1: With the last cut** (entire contour) – retract at an angle below 45°
  - **2: No smoothing** – retract at an angle below 45°
- **D**: **Omit elements** (see figure)
- **U**: **Cut line on vertical element**
  - **0: No** (uniform cutting segmentation)
  - **1: Yes** (irregular cutting segmentation, if applicable)
- **O**: **Hide undercutting**
  - **0: No**
  - **1: Yes**

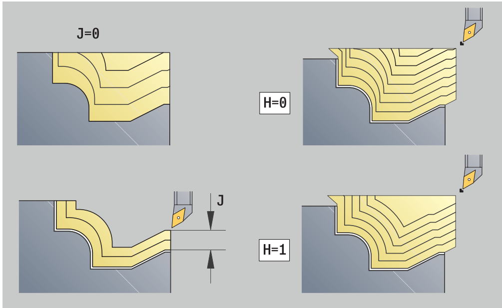
Further forms:

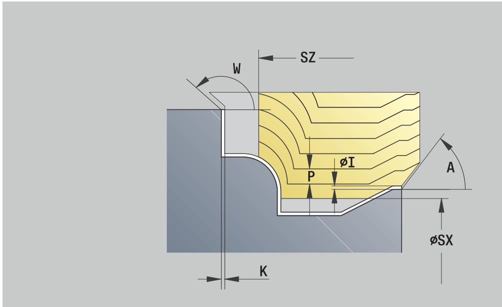
Further information: "smart.Turn unit", Page 98

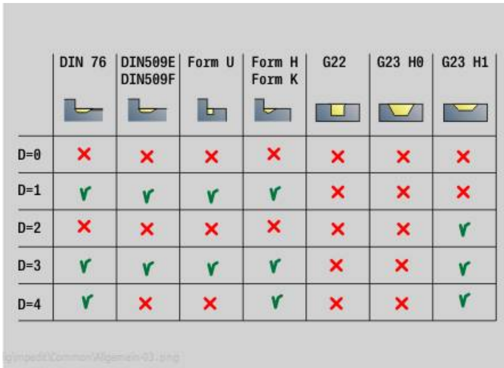
Access to the technology database:

- Machining operation: **Roughing**
- Affected parameters: **F**, **S**, **E**, **P**

Unit G830 Contr. parallel in ICP







	DIN 76	DIN509E DIN509F	Form U	Form H Form K	G22	G23 H0	G23 H1
D=0	×	×	×	×	×	×	×
D=1	✓	✓	✓	✓	×	×	×
D=2	×	×	×	×	×	×	✓
D=3	✓	✓	✓	✓	×	×	✓
D=4	✓	×	×	✓	×	×	✓

The unit machines the contour described in the **FINISHED PART** section from **NS** to **NE**, parallel to the contour. Any **Auxiliary contour** defined in **FK** will be used.

Unit name: **G830_ICP** / cycle: **G830**

Further information: "Contour-parallel roughing G830", Page 362

Contour form:

- **RH**: **Wrkpc. blank contour** – evaluation only if no workpiece blank has been defined
  - **0:** (depends on defined parameters)
    - No parameters: workpiece blank from ICP contour and tool position
    - **XA** and **ZA**: workpiece blank from ICP contour and starting point of workpiece blank
    - **J**: workpiece blank from ICP contour and equidistant oversize
  - **1: From tool position** (workpiece blank from ICP contour and tool position)
  - **2: With blank start point** (workpiece blank from ICP contour and starting point of workpiece blank **XA** and **ZA**)
  - **3 Equidistant oversize** (workpiece blank from ICP contour and equidistant oversize **J**)
  - **4: Long.-transv. oversize** (workpiece blank from ICP contour, transverse oversize **XA** and longitudinal oversize **ZA**)
- **J**: **Workpiece blank oversize** (radius value – will only be evaluated if no workpiece blank has been defined)
- **XA**, **ZA**: **Start point blank** (Definition of the corner point of the workpiece blank contour—an evaluation will only take place if no workpiece blank has been defined beforehand)
- **B**: **Contour calculation**

  B: **Contour calculation**
  - **0: Automatic**
  - **1: Tool left (G41)**
  - **2: Tool right(G42)**

Further parameters of the **Contour** form:

Further information: "Contour form", Page 100

Cycle form:

- **P**: **Maximum infeed**
- **I**, **K**: **O-size X** and **Z**
- **SX**, **SZ**: **Cutting limit in X** and **Z** (default: no cutting limit; diameter value = **SX**)
- **A**: **Start angle** (reference: Z axis; default: parallel to Z axis)
- **W**: **Depart.angle** (reference: Z axis; default: orthogonal to Z axis)
- **Q**: **Kind of liber.** at end of cycle
  - **0: Back to beg., X before Z**
  - **1: Before finished contour**
  - **2: Retract by safety clear.**

- **H**: **Type of cut lines**
  - **0: Constant mach.depth** – contour is shifted by a constant infeed value (paraxial)
  - **1: Equidistant cut lines** – cutting lines run at a constant distance from the contour (contour-parallel). The contour is scaled.
- **D**: **Omit elements** (see figure)
- **HR**: **Main machining direction**
  - **0: auto**
  - **1: +Z**
  - **2: +X**
  - **3: -Z**
  - **4: -X**

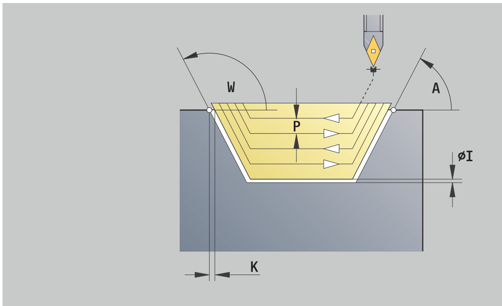
Further forms:

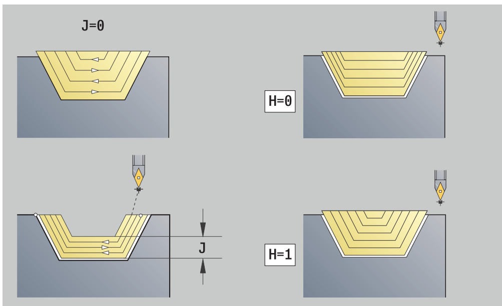
Further information: "smart.Turn unit", Page 98

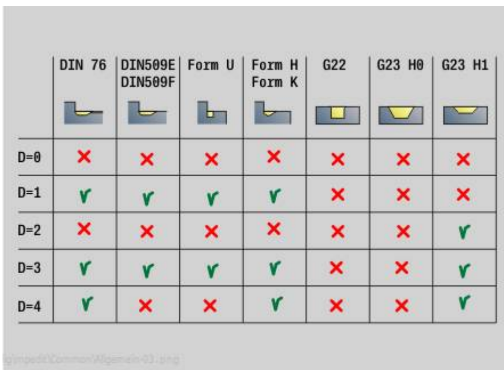
Access to the technology database:

- Machining operation: **Roughing**
- Affected parameters: **F**, **S**, **E**, **P**

Unit G835 Bidirectional in ICP







	DIN 76	DIN509E DIN509F	Form U	Form H Form K	G22	G23 H0	G23 H1
D=0	✗	✗	✗	✗	✗	✗	✗
D=1	✓	✓	✓	✓	✗	✗	✗
D=2	✗	✗	✗	✗	✗	✗	✓
D=3	✓	✓	✓	✓	✗	✗	✓
D=4	✓	✗	✗	✓	✗	✗	✓

The unit machines the contour described in the **FINISHED PART** section from **NS** to **NE** parallel to the contour and bidirectionally. Any **Auxiliary contour** defined in **FK** will be used.

Unit name: **G835_ICP** / cycle: **G835**

Further information: "Contour cycle, bidirectional G835 (contour-parallel with neutral tool)", Page 364

Contour form:

- **RH**: **Wrkpc. blank contour** – evaluation only if no workpiece blank has been defined
  - **0:** (depends on defined parameters)
    - No parameters: workpiece blank from ICP contour and tool position
    - **XA** and **ZA**: workpiece blank from ICP contour and starting point of workpiece blank
    - **J**: workpiece blank from ICP contour and equidistant oversize
  - **1: From tool position** (workpiece blank from ICP contour and tool position)
  - **2: With blank start point** (workpiece blank from ICP contour and starting point of workpiece blank **XA** and **ZA**)
  - **3 Equidistant oversize** (workpiece blank from ICP contour and equidistant oversize **J**)
  - **4: Long.-transv. oversize** (workpiece blank from ICP contour, transverse oversize **XA** and longitudinal oversize **ZA**)
- **J**: **Workpiece blank oversize** (radius value – will only be evaluated if no workpiece blank has been defined)
- **XA**, **ZA**: **Start point blank** (Definition of the corner point of the workpiece blank contour—an evaluation will only take place if no workpiece blank has been defined beforehand)

- **B**: **Contour calculation**

  B: **Contour calculation**
  - **0: Automatic**
  - **1: Tool left (G41)**
  - **2: Tool right(G42)**

Further parameters of the **Contour** form:

Further information: "Contour form", Page 100

Cycle form:

- **P**: **Maximum infeed**
- **I**, **K**: **O-size X** and **Z**
- **SX**, **SZ**: **Cutting limit in X** and **Z** (default: no cutting limit; diameter value = **SX**)
- **A**: **Start angle** (reference: Z axis; default: parallel to Z axis)
- **W**: **Depart.angle** (reference: Z axis; default: orthogonal to Z axis)
- **Q**: **Kind of liber.** at end of cycle
  - **0: Back to beg., X before Z**
  - **1: Before finished contour**
  - **2: Retract by safety clear.**
- **H**: **Type of cut lines**
  - **0: Constant mach.depth** – contour is shifted by a constant infeed value (paraxial)
  - **1: Equidistant cut lines** – cutting lines run at a constant distance from the contour (contour-parallel). The contour is scaled.
- **D**: **Omit elements** (see figure)

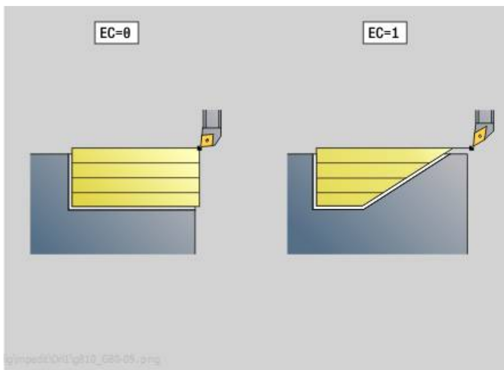
Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **Roughing**
- Affected parameters: **F**, **S**, **E**, **P**

Unit G810 Roughing, direct longitudnl.



The unit machines the contour defined by the parameters. In **EC** you define whether you want to machine a normal or a plunging contour.

Unit name: **G810_G80** / cycle: **G810**

Further information: "Longitud. roughing G810", Page 356

Contour form:

- **EC**: **Type of contour**
  - **0: Normal contour**
  - **1: Plunging contour**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **RC**: **Rounding** – radius of contour corner
- **AC**: **Start angle** – angle of the first contour element (range: 0°< **AC** < 90°)
- **WC**: **Final angle** – angle of the last contour element (range: 0° < **WC** < 90°)
- **BS**: **-Chamfer/+radius at start**

- **BS** > 0: Radius of rounding arc
- **BS** < 0: Width of chamfer
- **BE**: **-Chamfer/+radius at end**
  - **BE** > 0: Radius of rounding arc
  - **BE** < 0: Width of chamfer
- **BP**: **Interval time** – time during which the feed motion will be interrupted
  The chip is broken by the intermittent feed.
- **BF**: **Feed period** – time interval until the next break
  The chip is broken by the intermittent feed.

Cycle form:

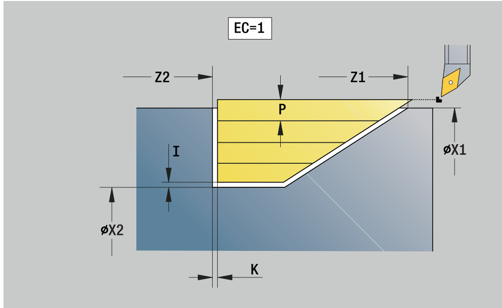
- **P**: **Maximum infeed**
- **I**, **K**: **O-size X** and **Z**
- **E**: **Plunging behavior**
  - **E** = 0: Descending contours are not machined
  - **E** > 0: Plunging feed rate for descending contour elements. Descending contour elements are machined
  - No input: The plunging feed rate is reduced during machining of descending contour elements by up to 50 %. Descending contour elements are machined
- **H**: **Contour smoothing**
  - **0: With each cut** (within the infeed range)
  - **1: With the last cut** (entire contour) – retract at an angle below 45°
  - **2: No smoothing** – retract at an angle below 45°

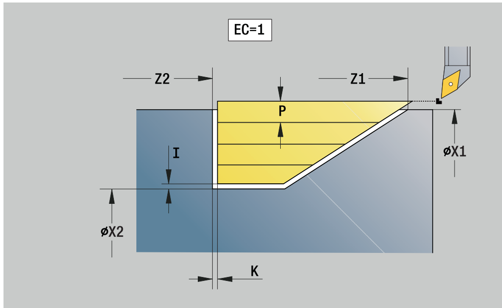
Further forms:

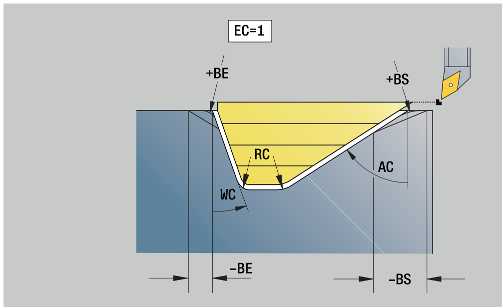
Further information: "smart.Turn unit", Page 98

Access to the technology database:

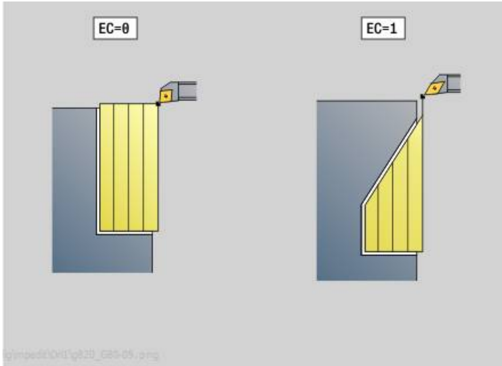
- Machining operation: **Roughing**
- Affected parameters: **F**, **S**, **E**, **P**

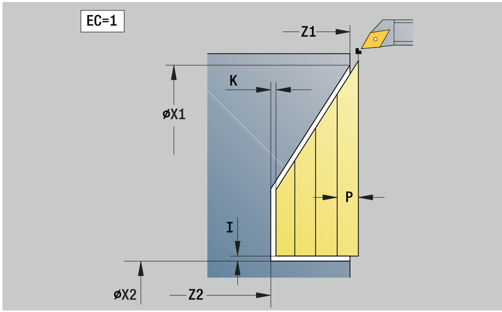


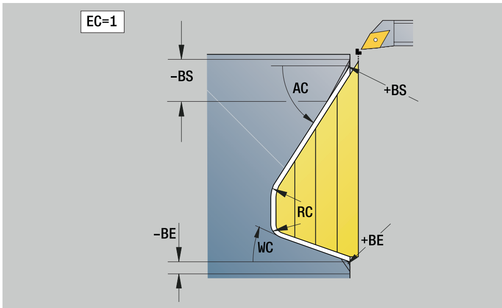




Unit G820 Roughing, direct transverse







The unit machines the contour defined by the parameters. In **EC** you define whether you want to machine a normal or a plunging contour.

Unit name: **G820_G80** / cycle: **G820**

Further information: "Face roughing G820", Page 359

Contour form:

- **EC**: **Type of contour**
  - **0: Normal contour**
  - **1: Plunging contour**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **RC**: **Rounding** – radius of contour corner
- **AC**: **Start angle** – angle of the first contour element (range: 0°< **AC** < 90°)
- **WC**: **Final angle** – angle of the last contour element (range: 0° < **WC** < 90°)
- **BS**: **-Chamfer/+radius at start**
  - **BS** > 0: Radius of rounding arc
  - **BS** < 0: Width of chamfer
- **BE**: **-Chamfer/+radius at end**
  - **BE** > 0: Radius of rounding arc
  - **BE** < 0: Width of chamfer
- **BP**: **Interval time** – time during which the feed motion will be interrupted
  The chip is broken by the intermittent feed.
- **BF**: **Feed period** – time interval until the next break
  The chip is broken by the intermittent feed.

Cycle form:

- **P**: **Maximum infeed**
- **I**, **K**: **O-size X** and **Z**
- **E**: **Plunging behavior**
  - **E** = 0: Descending contours are not machined
  - **E** > 0: Plunging feed rate for descending contour elements. Descending contour elements are machined
  - No input: The plunging feed rate is reduced during machining of descending contour elements by up to 50 %. Descending contour elements are machined
- **H**: **Contour smoothing**
  - **0: With each cut** (within the infeed range)
  - **1: With the last cut** (entire contour) – retract at an angle below 45°
  - **2: No smoothing** – retract at an angle below 45°

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **Roughing**
- Affected parameters: **F**, **S**, **E**, **P**

Unit G895 Simultaneous roughing (option 54)

G895 enables three-axis simultaneous roughing of the defined contour area in several steps.

> In order to enable realistic collision monitoring through the cycle, you need to assign the corresponding tool holder to the tool to be used.
>
> The real holder must be within the defined holder dimensions.
>
> In addition to the holder, the machine tool builder can also describe another component of the tilting axis as a collision object (e.g., the B-axis head). If this description is available as a 2-D view in the plane of rotation, then this object will be displayed in the 2-D simulation and will automatically be included in collision monitoring.

NOTICE

Danger of collision!

Collision monitoring is performed only in the two-dimensional machining plane XZ. The cycle does not check whether an area in the Y coordinate of the cutting edge, tool holder, or tilting body will lead to a collision.

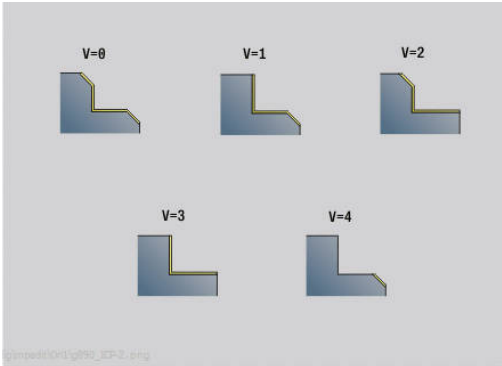
- Perform a first run of the NC program in **Single block** mode
- Limit the machining area

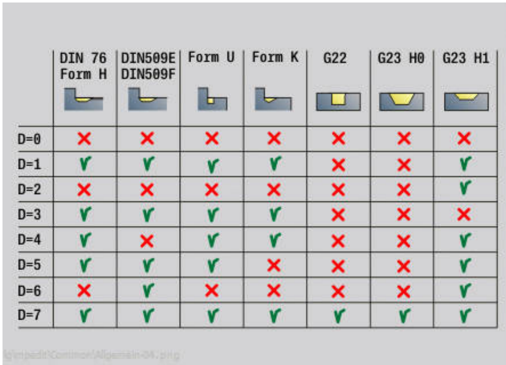
Unit name: **G895_ICP** / Cycle: **G895**

Further information: "Simultaneous roughing G895 (option 54)", Page 378

Contour form:

- **FK**: **Auxiliary contour** – name of the contour to be machined
  You can select an existing contour or describe a new contour with **ICP**.
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
  - **NE** not programmed: The contour element **NS** is machined in the direction of contour definition
  - **NS** = **NE** programmed: The contour element **NS** is machined opposite to the direction of contour definition.
- **V**: **Machine form elements** (default: 0)
  A chamfer/rounding arc is machined
  - **0: At beginning and end**
  - **1: At beginning**
  - **2: At end**
  - **3: No machining**
  - **4: Only chamfer/rounding** is machined—not the basic element (requirement: the contour section consists of a single element)
- **D**: **Omit elements** (see figure)





	DIN 76 Form H	DIN509E DIN509F	Form U	Form K	G22	G23 H0	G23 H1
D=0	✗	✗	✗	✗	✗	✗	✗
D=1	✓	✓	✓	✓	✗	✗	✓
D=2	✗	✗	✗	✗	✗	✗	✓
D=3	✓	✓	✓	✓	✗	✗	✗
D=4	✓	✗	✓	✓	✗	✗	✓
D=5	✓	✓	✓	✗	✗	✗	✓
D=6	✗	✓	✗	✗	✗	✗	✓
D=7	✓	✓	✓	✓	✓	✓	✓

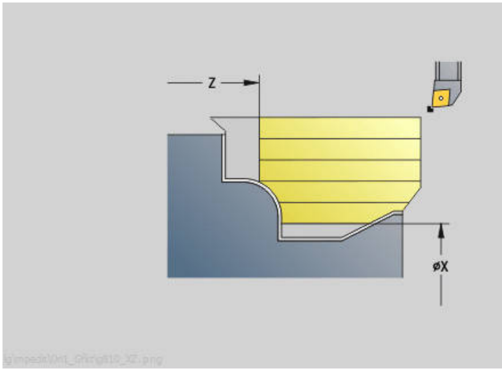
Codes for omitting recesses and undercuts

G call	Function	D code
G22	Recess for sealing ring	512
G22	Recess for circlip	1.024
G23 H0	General recess	256
G23 H1	Relief turn	2.048
G25 H4	Undercut type U	32.768
G25 H5	Undercut type E	65.536
G25 H6	Undercut type F	131.072
G25 H7	Undercut type G	262.144
G25 H8	Undercut type H	524.288
G25 H9	Undercut type K	1.048.576

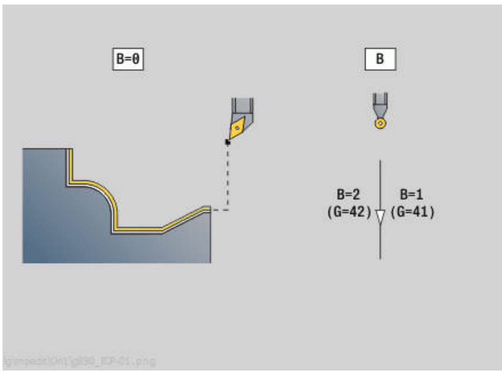
In order to omit multiple elements, you add the D codes from the table or use the D values from the graphic.

Example for omitting the undercut types **E** and **F**:

65.536 + 131.072 = 196.608



- **SX**, **SZ**: **Cutting limit in X** and **Z** (default: no cutting limit; diameter value = **SX**)
- **A**: **Start angle** (reference: Z axis; default: parallel to Z axis)
- **W**: **Depart.angle** (reference: Z axis; default: orthogonal to Z axis)
- **I**: **O-size X**
- **K**: **O-size Z**
- **B**: **Contour calculation**
  - **0: Automatic**
  - **1: Tool left (G41)**
  - **2: Tool right(G42)**



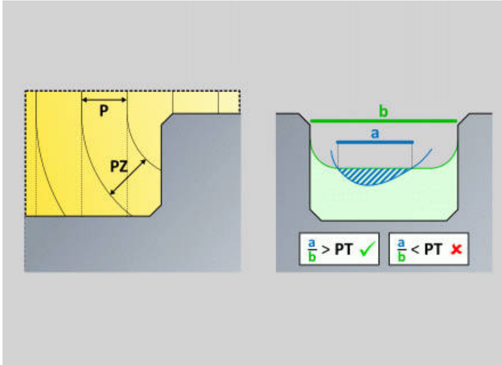
Cycle form:

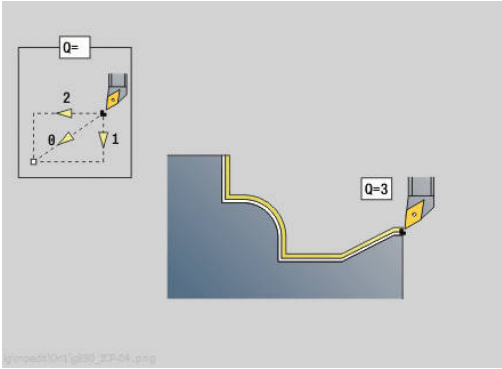
- **P**: **Desired infeed** – basis for calculating the infeed
- **PZ**: **Maximum infeed**

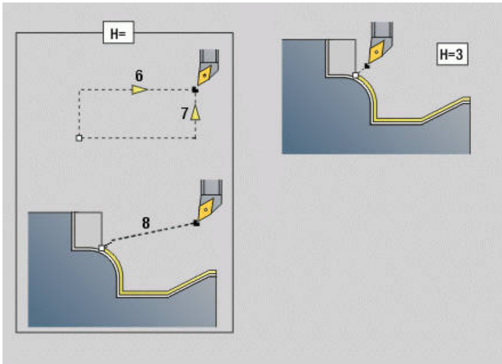
  Further information: "Material removal:", Page 380
- **PT**: **Minimum removal** – maintaining the infeed **P** in %
- **Q**: **Type of approach** (default: 0)
  - **0: automatic (with B)** – the control checks:
    - Diagonal approach
    - First X, then Z direction
    - Equidistant around the workpiece blank, taking the safety clearance into account
    - Omission of the first contour elements if the starting position is inaccessible
  - **1: First X, then Z**
  - **2: First Z, then X**
  - **3: No approach** – tool is located near the starting point of the contour area
- **H**: **Kind of liber.**
  - **3: Retract by safety clear.**
  - **6: X then Z to start pos.**
  - **7: Z then X to start pos.**
  - **8: with B-axis motion to start pos.**
- **U** : **Use of soft clearance angles** – defines how the soft clearance angles **IC** and **JC** are used

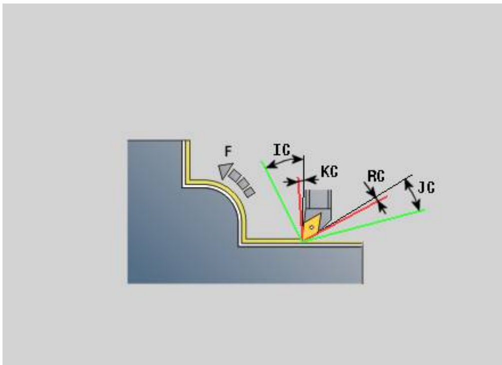
The parameter **U Use of soft clearance angles** provides the following settings options:

- **0: very hard**
- **1: hard**
- **2: medium**
- **3: soft**
- **4: very soft**
- **IC**: **Primary clearance angle - soft**: Desired clearance area in front of the cutting edge
- **JC** : **Secondary clearance angle - soft**: Desired clearance area behind the cutting edge
- **KC**: **Primary clearance angle - hard**: Safe clearance area in front of the cutting edge
- **RC**: **Primary clearance angle - hard**: Safe clearance area behind the cutting edge









Cycle 2 form:

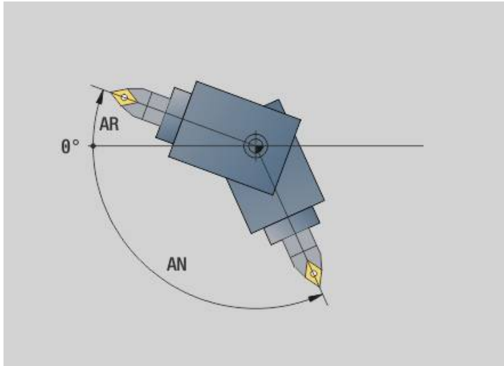
- **AR**: **Minimum angle of incidence** – minimum permissible angle of the tilting axis (range: –359.999° < **AR** < 359.999°)
- **AN**: **Maximum angle of incidence** – maximum permissible angle of the tilting axis (range: –359.999° < **AN** < 359.999°)
- **C**: **Cutting strategy** – type of cut lines (cutting paths)
  - **0: automatic** – the control automatically combines transverse and longitudinal turning
  - **1: longitudinal (outside)**
  - **2: transverse (front)**
  - **3: longitudinal (inside)**
  - **4: transverse (back)**
  - **5: parallel to blank**
- **EC**: **Cutting direction**
  - **0: Unidirectional** – every cut is performed in the direction of contour definition
  - **1: Bidirectional** – the cuts are performed along the optimum cutting path with respect to the machining time, and can be performed in the direction or opposite to the direction of contour definition
- **AS**: **Strategy for sequence** – machining sequence for separate pockets
  - **0: transverse (preferred)** – the machining sequence is chosen so that the center of gravity of the workpiece is always as close to the chucking equipment as possible
  - **1: longitudinal (preferred)** – the machining sequence is chosen so that the moment of inertia of the workpiece is as low as possible
- **SL**: **Oversize of tool holder** – oversize for calculating collisions between the workpiece and the tool holder
- **E**: **Fmax in compensating motion** – speed limit during the compensation movement of the linear axes
- **EW**: **Plunging feed rate** – feed rate for plunging into the material in mm/min
- **BP**: **Interval time** – time during which the feed motion will be interrupted
  The chip is broken by the intermittent feed.
- **BF**: **Feed period** – time interval until the next break
  The chip is broken by the intermittent feed.

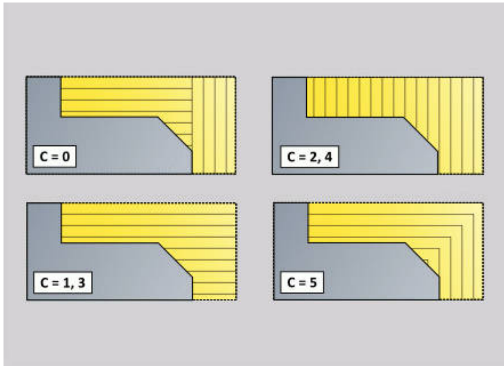
  Further forms:

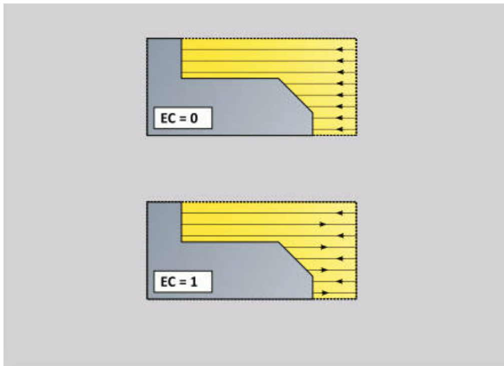
  Further information: "smart.Turn unit", Page 98

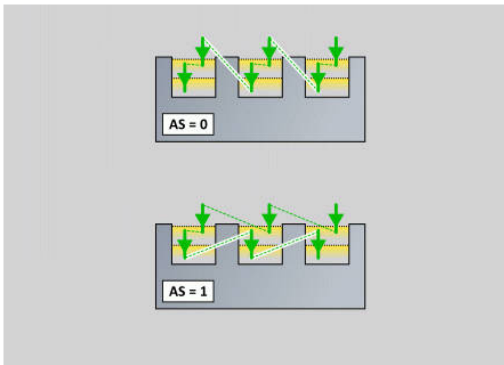
  Access to the technology database:

  - Machining operation: **Roughing**
  - Affected parameters: **F**, **S**, **E**, **P**



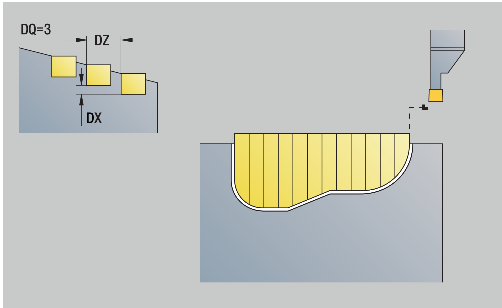


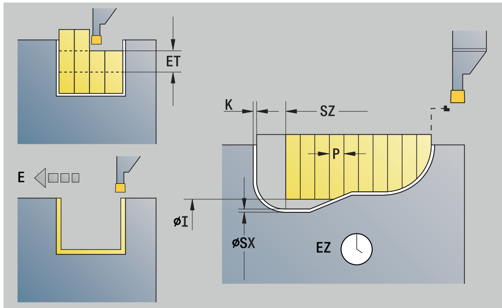




4.3 Units – Recessg

Unit G860 Contour recess in ICP





The unit machines the contour described in the **FINISHED PART** section axially/radially from **NS** to **NE**. Any **Auxiliary contour** defined in **FK** will be used.

Unit name: **G860_ICP** / cycle: **G860**

Further information: "Recessing G860", Page 366

Contour form:

- **SX**, **SZ**: **Cutting limit in X** and **Z** (default: no cutting limit; diameter value = **SX**)
- **DQ**: **Number of recess cycles**
- **DX**, **DZ**: **Dist.to subsequent recess** in X, Z direction (**DX** = radius value)
- **DO**: **Flow** (with parameters **Q** = 0 and **DQ** > 1)
  - **0: Complete roughing/finishing** – rough-machine all recesses, then finish-machine all recesses
  - **1: Single roughing/finishing** – machine each recess completely before machining the next one

Cycle form:

- **I**, **K**: **O-size X** and **Z**
- **ET**: **Recess depth** per infeed
- **P**: **Cut. width** – Infeeds <= **P** (no input: **P** = 0.8 * cutting width of the tool)
- **E**: **Finishing feed**
- **EW**: **Recessing feed**
- **EZ**: **Delay** after recessing path (default: time for one spindle revolution)
- **D**: **Rev. on recessing floor**
- **Q**: **Roughing/Finish** – process variants
  - **0: Roughing and finishing**
  - **1: Only roughing**
  - **2: Only finishing**
- **KS**: **Multiple plunging** (default: 0)
  - 0: No
  - 1: Yes – Rough grooving is performed by means of full-section cuts—the ridges in between are machined centrally relative to the recessing tool
- **H**: **Kind of liber.** at end of cycle
  - **0: Back to start point**
    - Axial recess: First Z, then X direction
    - Radial recess: First X, then Z direction
  - **1: Before finished contour**
  - **2: Stops at safety clear.**

- **O**: **End of rough cut**
  - **0: Lift-up at rapid**
  - **1: Half recessng width 45°**
- **U**: **End of finishing cut**
  - **0: Value from glob. param.**
  - **1: Parting horizntl. elem.**
  - **2: Complete horzntl. elem.**

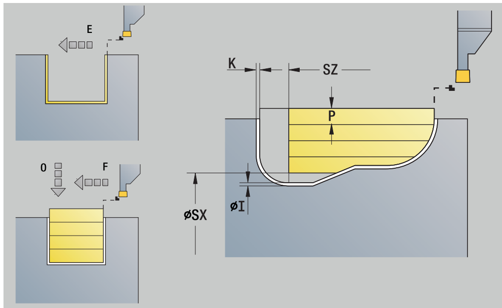
Further forms:

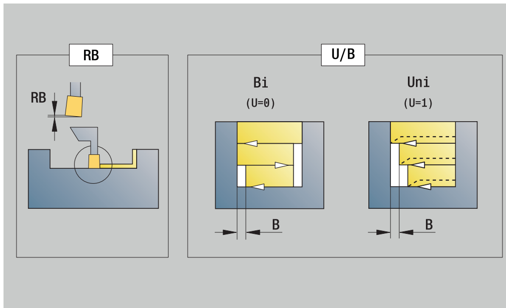
Further information: "smart.Turn unit", Page 98

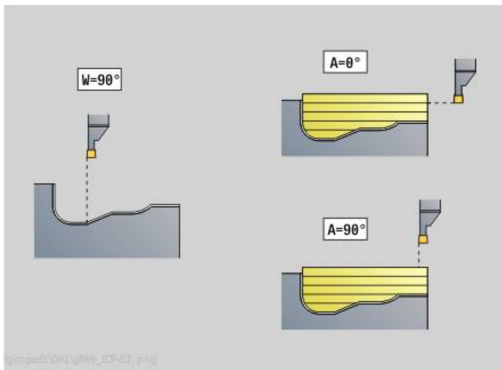
Access to the technology database:

- Machining operation: **Contour recessing**
- Affected parameters: **F**, **S**, **E**

Unit G869 Recess turning in ICP







The unit machines the contour described by **ICP** axially/radially from **NS** to **NE**. The workpiece is machined by alternate recessing and roughing movements.

The unit machines the contour described in the **FINISHED PART** section axially/radially from **NS** to **NE**. Any **Auxiliary contour** defined in **FK** will be used.

Unit name: **G869_ICP** / cycle: **G869**

Further information: "Recess turning cycle G869", Page 370

Contour form:

- **X1**, **Z1**: **Start point blank** – evaluation only if no blank was defined
- **RI**, **RK**: **Oversize blank X** and **Z**
- **SX**, **SZ**: **Cutting limit in X** and **Z** (default: no cutting limit; diameter value = **SX**)

Further parameters of the **Contour** form:

Further information: "Contour form", Page 100

Cycle form:

- **P**: **Maximum infeed**
- **I**, **K**: **O-size X** and **Z**
- **RB**: **Depth compens.** for finishing
- **B**: **Offset width** (default: 0)
- **U**: **Direction:** – cutting direction
  - **0**: Bidirectional (in both directions)
  - **1**: Unidirectional (in direction of contour)
- **Q**: **Roughing/Finish** – process variants
  - **0: Roughing and finishing**
  - **1: Only roughing**
  - **2: Only finishing**
- **A**: **Approach angle** (default: opposite to recessing direction)
- **W**: **Depart.angle** (default: opposite to recessing direction)
- **O**: **Recessing feed** (default: active feed rate)
- **E**: **Finishing feed**

- **H**: **Kind of liber.** at end of cycle
  - **0: Back to start point**
    - Axial recess: First Z, then X direction
    - Radial recess: First X, then Z direction
  - **1: Before finished contour**
  - **2: Stops at safety clear.**

Further forms:

Further information: "smart.Turn unit", Page 98

The control uses the tool definition to distinguish between radial and axial recessing.

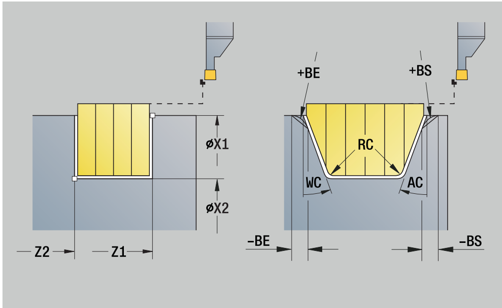
Depth compens. RB: Depending on factors such as workpiece material or feed rate, the cutting edge is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation factor. The value is usually determined empirically.

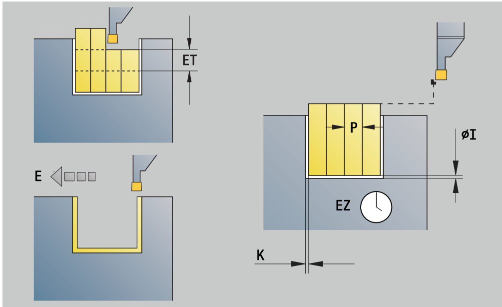
Offset width B: After the second infeed movement, during the transition from turning to recessing, the length to be machined is reduced by **Offset width B**. For each other transition at this edge, the path is reduced by **B**—in addition to the previous offset. The total offset is limited to 80 % of the effective cutting width (effective cutting width = cutting width – 2*cutting radius). If required, the control reduces the programmed offset width. After precutting (rough grooving), the remaining material is machined with a single cut.

Access to the technology database:

- Machining operation: **Recessing**
- Affected parameters: **F**, **S**, **O**, **P**

Unit G860 Contour recess, direct





The unit machines the contour defined by the parameters axially or radially.

Unit name: **G860_G80** / cycle: **G860**

Further information: "Recessing G860", Page 366

Contour form:

- **DQ**: **Number of recess cycles**
- **DX**, **DZ**: **Dist.to subsequent recess** in X, Z direction (**DX** = radius value)
- **DO**: **Flow** (with parameters **Q** = 0 and **DQ** > 1)

Further parameters of the **Contour** form:

Further information: "Contour form", Page 100

Cycle form:

- **Q**: **Roughing/Finish** – process variants
  - **0: Roughing and finishing**
  - **1: Only roughing**
  - **2: Only finishing**
- **KS**: **Multiple plunging** (default: 0)
  - 0: No
  - 1: Yes – Rough grooving is performed by means of full-section cuts—the ridges in between are machined centrally relative to the recessing tool
- **I**, **K**: **O-size X** and **Z**
- **ET**: **Recess depth** per infeed
- **P**: **Cut. width** – Infeeds <= **P** (no input: **P** = 0.8 * cutting width of the tool)
- **E**: **Finishing feed**
- **EW**: **Recessing feed**
- **EZ**: **Delay** after recessing path (default: time for one spindle revolution)
- **D**: **Rev. on recessing floor**
  - **0: Complete roughing/finishing** – rough-machine all recesses, then finish-machine all recesses
  - **1: Single roughing/finishing** – machine each recess completely before machining the next one

Further forms:

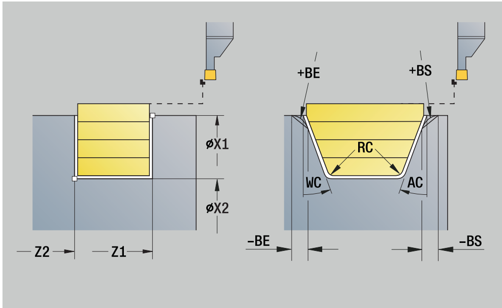
Further information: "smart.Turn unit", Page 98

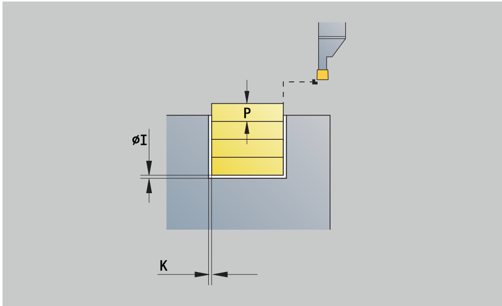
The control uses the tool definition to distinguish between radial and axial recessing.

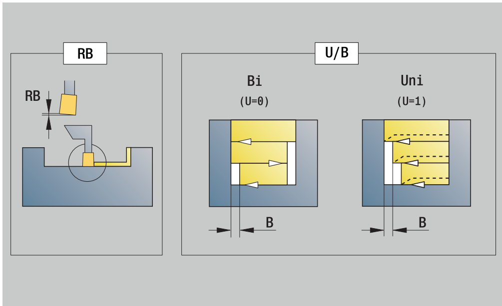
Access to the technology database:

- Machining operation: **Contour recessing**
- Affected parameters: **F**, **S**, **E**

Unit G869 Recess turning, direct







The unit machines the contour defined by the parameters axially or radially. The workpiece is machined by alternate recessing and roughing movements. The machining process requires a minimum of retraction and infeed movements.

Unit name: **G869_G80** / cycle: **G869**

Further information: "Recess turning cycle G869", Page 370

Contour form:

- **RI**, **RK**: **Oversize blank X** and **Z**

Further parameters of the **Contour** form:

Further information: "Contour form", Page 100

Cycle form:

- **P**: **Maximum infeed**
- **I**, **K**: **O-size X** and **Z**
- **RB**: **Depth compens.** for finishing
- **B**: **Offset width** (default: 0)
- **U**: **Direction:** – cutting direction
  - **0**: Bidirectional (in both directions)
  - **1**: Unidirectional (in direction of contour)
- **Q**: **Roughing/Finish** – process variants
  - **0: Roughing and finishing**
  - **1: Only roughing**
  - **2: Only finishing**

Further forms:

Further information: "smart.Turn unit", Page 98

The control uses the tool definition to distinguish between radial and axial recessing.

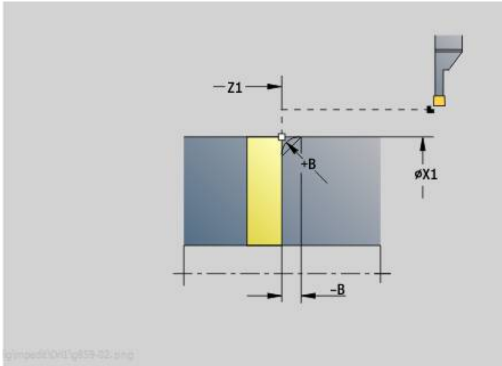
Depth compens. RB: Depending on factors such as workpiece material or feed rate, the cutting edge is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation factor. The value is usually determined empirically.

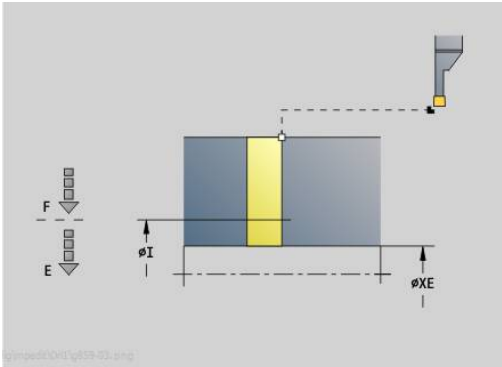
Offset width B: After the second infeed movement, during the transition from turning to recessing, the length to be machined is reduced by **Offset width B**. For each other transition at this edge, the path is reduced by **B**—in addition to the previous offset. The total offset is limited to 80 % of the effective cutting width (effective cutting width = cutting width – 2*cutting radius). If required, the control reduces the programmed offset width. After precutting (rough grooving), the remaining material is machined with a single cut.

Access to the technology database:

- Machining operation: **Recessing**
- Affected parameters: **F**, **S**, **O**, **P**

Unit G859 Parting





The unit parts the workpiece. If programmed, a chamfer or rounding arc is machined on the outside diameter. At the end of cycle, the tool returns to the starting point. You can define a feed rate reduction, which becomes effective as soon as the position **I** is reached.

Unit name: **G859_CUT_OFF** / cycle: **G859**

Further information: "Cut-off cycle G859", Page 413

Cycle form:

- **X1**, **Z1**: **Start point contour**
- **B**: **-B cham./+B round.**
  - **B** > 0: Radius of rounding
  - **B** < 0: Width of chamfer
- **D**: **Maximum speed**
- **XE**: **Inner diameter (pipe)**
- **I**: **Diam. feed reduction** – Limit diameter above which the workpiece will be machined with a reduced feed rate
- **E**: **Reduced feed**
- **SD**: **Speed limit from I up**
- **U**: **Collector active diameter** (machine-dependent)
- **K**: **Retraction distance** after parting – Lift off the tool laterally from the plane surface before retraction

Further forms:

Further information: "smart.Turn unit", Page 98

> The limit to the **Maximum speed D** is only effective in the cycle. After the cycle ends, the speed limit that was valid before the cycle becomes effective again.

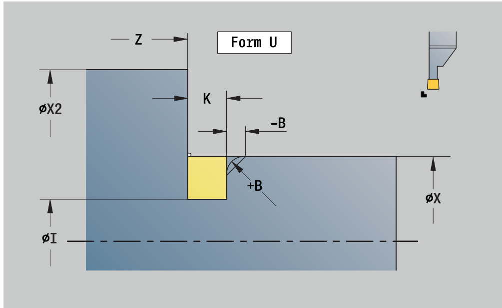
Access to the technology database:

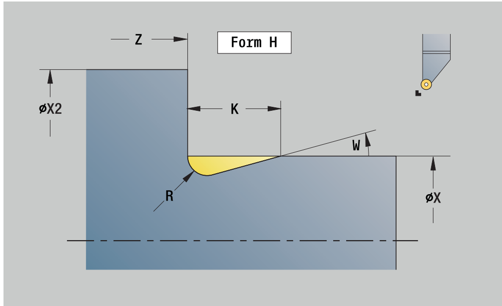
- Machining operation: **Contour recessing**
- Affected parameters: **F**, **S**, **E**

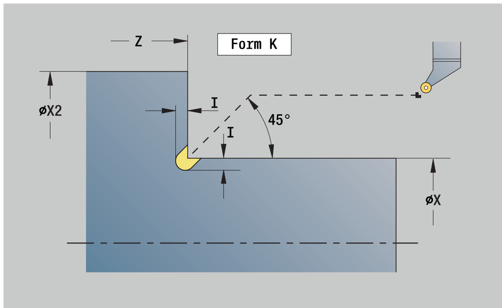
Unit G85X Undercutting (H,K,U)

Depending on **KG**, the unit machines one of the following undercuts:

- **Type U**: The unit machines an undercut and finishes the adjoining plane surface. Either a chamfer or a rounding arc can be machined
- **Type H**: The end point of the undercut is determined from the plunging angle
- **Type K**: Only one linear cut at an angle of 45° is performed. The resulting contour geometry therefore depends on the tool used







> - First, select the **Type of undercut KG** and then, enter the values for the selected type of undercut
> - Changes are also reflected in other undercuts where the same parameter letters are used. Do not change these values

Unit name: **G85x_H_K_U** / cycle: **G85**

Further information: "Undercut cycle G85", Page 414

Contour form:

- **KG**: **Type of undercut**
  - **Form U G856**
    Further information: "Undercut type U G856", Page 419
  - **Form H G857**
    Further information: "Undercut type H G857", Page 420
  - **Form K G858**
    Further information: "Undercut type K G858", Page 420
- **X1**, **Z1**: **Contour corner**

Undercut **type U**:

- **X2**: **End point transv.**
- **I**: **Undercut diameter**
- **K**: **Undercut lgth.**
- **B**: **-B cham./+B round.**
  - **B** > 0: Radius of rounding
  - **B** < 0: Width of chamfer

Undercut **type H**:

- **K**: **Undercut lgth.**
- **R**: **Radius** in the undercut corner
- **W**: **Plunging angle**

Undercut **type K**:

- **I**: **Undercut depth**

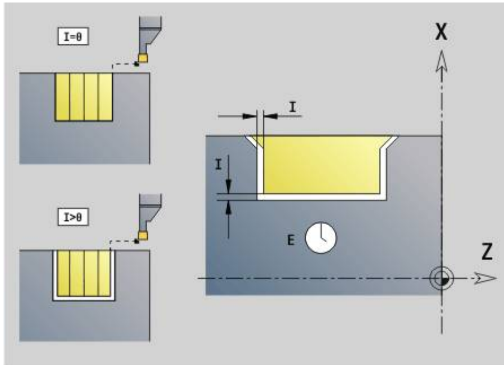
Further forms:

Further information: "smart.Turn Units (Option 9)", Page 97

Access to the technology database:

- Machining operation: **Finishing**
- Affected parameters: **F**, **S**

Unit G870 ICP Recessing – Recess cycle



G870 generates a recess defined by **G22**-Geo. The control uses the tool definition to distinguish between external and internal machining, or between radial and axial recesses.

Unit name: **G870_ICP** / cycle: **G870**

Further information: "Recessing cycle G870", Page 373

Contour form:

- **I**: **Allowance**
- **EZ**: **Delay** after recessing path (default: time for one spindle revolution)

Further parameters of the **Contour** form:

Further information: "Contour form", Page 100

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **Recessg**
- Affected parameters: **F**, **S**

4.4 Units – Drilling / Centric

Unit G74 Centric drilling

The unit uses stationary tools to drill axial holes in several passes. Suitable tools can be positioned up to +/– 2 mm outside the turning center.

Unit name: **G74_ZENTR** / cycle: **G74**

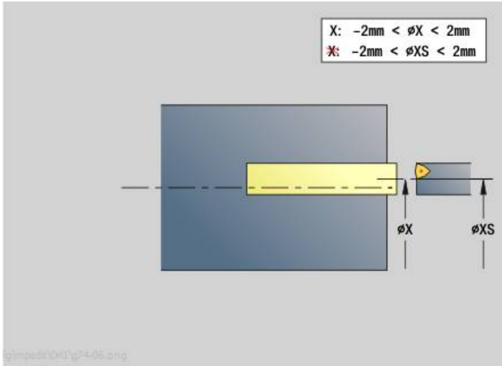
Further information: "Deep boring G74", Page 428

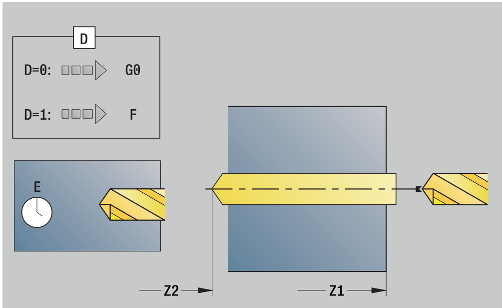
Cycle form:

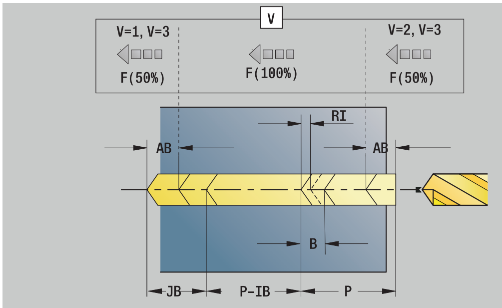
- **Z1**: **Start point drill**
- **Z2**: **End point drill**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **X**: **Start point drill** (diameter value; range: –2 mm < **X** < 2 mm; default: 0)
- **E**: **Period of dwell** at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **DFF**: **Retraction feed rate**
- **V**: **Feed reduction**
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **AB**: **Drilling lengths** (default: 0)
- **P**: **1. Boring depth**
- **IB**: **Hole depth reduct. val** – value by which the hole depth decreases after every infeed.
- **JB**: **Minimum hole depth**
  If you have entered a hole depth reduction value, the hole depth is reduced only to the value entered in **JB**.
- **B**: **Return distance** – value by which the tool is retracted after reaching the respective hole depth
- **RI**: **Safety clearance** internal – distance for reapproach inside the hole (default: **Safety clearance SCK**)

Global form:

- **G14**: **Tool change point**
  - **No axis**
  - **0: Simultaneously**
  - **1: First X, then Z**
  - **2: First Z, then X**
  - **3: Only X**
  - **4: Only Z**
  - **5: Only Y** (machine-dependent)
  - **6: Simultaneous w/ Y** (machine-dependent)







- **CLT**: **Coolant**
  - **0: Without**
  - **1: Coolant 1 on**
  - **2: Coolant 2 on**
- **SCK**: **Safety clearance** in infeed direction during drilling and milling operations
- **G60**: **Protection zone** – protection zone monitoring during drilling
  - **0: Active**
  - **1: Inactive**
- **BP**: **Interval time** – time during which the feed motion will be interrupted
  The chip is broken by the intermittent feed.
- **BF**: **Feed period** – time interval until the next break
  The chip is broken by the intermittent feed.

Further forms:

Further information: "smart.Turn unit", Page 98

> If **X** is not programmed or **XS** is in the range of –2 mm < **XS** < 2 mm, then the control drills at **XS**.

Access to the technology database:

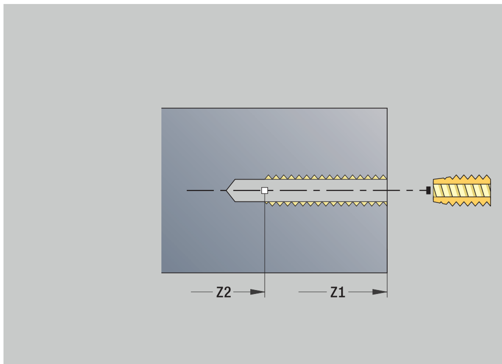
- Machining operation: **Drilling**
- Affected parameters: **F**, **S**

Unit G73 Centric tapping

The unit cuts axial threads using stationary tools.

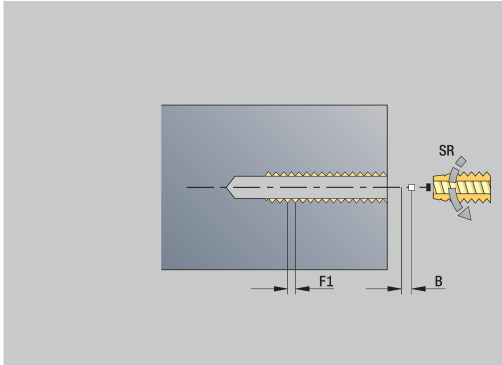
Unit name: **G73_ZENTR** / cycle: **G73**

Further information: "Tapping G73", Page 426



Cycle form:

- **Z1**: **Start point drill**
- **Z2**: **End point drill**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **X**: **Start point drill** (diameter value; range: –2 mm < **X** < 2 mm; default: 0)
- **F1**: **Thread pitch**
- **B**: **Run-in lgth** to obtain the programmed spindle speed and feed rate (default: 2 * **Thread pitch F1**)
- **L**: **Retract length** when using floating tap holders (default: 0)
- **SR**: **Return speed** (default: tapping speed)
- **SP**: **Chip breaking depth**
- **SI**: **Retraction distance**



Global form:

- **G14**: **Tool change point**
  - **No axis**
  - **0: Simultaneously**
  - **1: First X, then Z**
  - **2: First Z, then X**
  - **3: Only X**
  - **4: Only Z**
  - **5: Only Y** (machine-dependent)
  - **6: Simultaneous w/ Y** (machine-dependent)
- **CLT**: **Coolant**
  - **0: Without**
  - **1: Coolant 1 on**
  - **2: Coolant 2 on**
- **SCK**: **Safety clearance** in infeed direction during drilling and milling operations
- **G60**: **Protection zone** – protection zone monitoring during drilling
  - **0: Active**
  - **1: Inactive**

Further forms:

Further information: "smart.Turn unit", Page 98

Retract length L: 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 tap is pulled away from the chuck by the retraction length. With this method, you can achieve higher service life of the taps.

Access to the technology database:

- Machining operation: **Tapping**
- Affected parameter: **S**

> If you interrupt program run during a tapping cycle, you can manually retract the tool from the hole in the Z axis. The control moves the spindle on a path matching the traverse.
>
> If the optional machine parameter **CfgBackTrack** (no. 122000) is active, use the **Start blck search** soft key to resume program run after the manual traverse.

Unit G72 Boring, cntrsnkg.

The unit uses stationary tools to drill axial holes in several passes.

Unit name: **G72_ZENTR** / cycle: **G72**

Further information: "Boring/cnt-sink G72", Page 425

Cycle form:

- **NS**: **Starting block no. of contour** – beginning of contour section
- **E**: **Period of dwell** at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **DFF**: **Retraction feed rate**
- **RB**: **Return plane**

Global form:

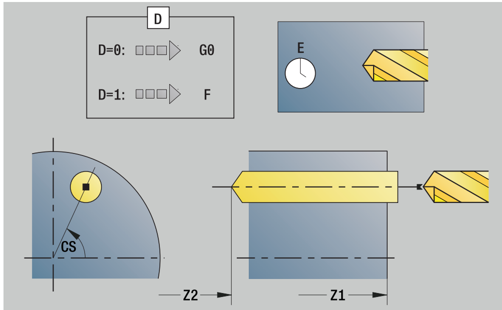
- **G14**: **Tool change point**
  - **No axis**
  - **0: Simultaneously**
  - **1: First X, then Z**
  - **2: First Z, then X**
  - **3: Only X**
  - **4: Only Z**
  - **5: Only Y** (machine-dependent)
  - **6: Simultaneous w/ Y** (machine-dependent)
- **CLT**: **Coolant**
  - **0: Without**
  - **1: Coolant 1 on**
  - **2: Coolant 2 on**
- **SCK**: **Safety clearance** in infeed direction during drilling and milling operations
- **G60**: **Protection zone** – protection zone monitoring during drilling
  - **0: Active**
  - **1: Inactive**

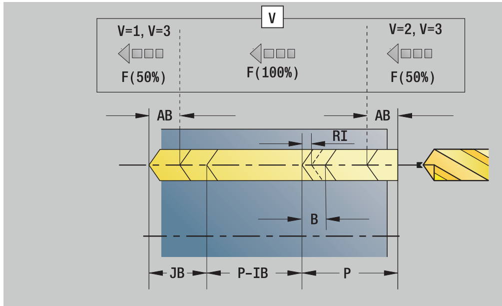
Further forms:

Further information: "smart.Turn unit", Page 98

4.5 Units – Drillg / Face C, Lateral C, and ICP C

Unit G74 Single hole, front face C





This unit machines a hole on the face of the workpiece.

Unit name: **G74_Bohr_Stirn_C** / cycle: **G74**

Further information: "Deep boring G74", Page 428

Cycle form:

- **Z1**: **Start point drill**
- **Z2**: **End point drill**
- **CS**: **Spindle angle**
- **E**: **Period of dwell** at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **DFF**: **Retraction feed rate**
- **V**: **Feed reduction**
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **AB**: **Drilling lengths** (default: 0)
- **P**: **1. Boring depth**
- **IB**: **Hole depth reduct. val** – value by which the hole depth decreases after every infeed.
- **JB**: **Minimum hole depth**
  If you have entered a hole depth reduction value, the hole depth is reduced only to the value entered in **JB**.
- **B**: **Return distance** – value by which the tool is retracted after reaching the respective hole depth
- **RI**: **Safety clearance** internal – distance for reapproach inside the hole (default: **Safety clearance SCK**)

Global form:

- **G14**: **Tool change point**
  - **No axis**
  - **0: Simultaneously**
  - **1: First X, then Z**
  - **2: First Z, then X**
  - **3: Only X**
  - **4: Only Z**
  - **5: Only Y** (machine-dependent)
  - **6: Simultaneous w/ Y** (machine-dependent)
- **CLT**: **Coolant**
  - **0: Without**
  - **1: Coolant 1 on**
  - **2: Coolant 2 on**

- **SCK**: **Safety clearance** in infeed direction during drilling and milling operations
- **G60**: **Protection zone** – protection zone monitoring during drilling
  - **0: Active**
  - **1: Inactive**
- **BP**: **Interval time** – time during which the feed motion will be interrupted
  The chip is broken by the intermittent feed.
- **BF**: **Feed period** – time interval until the next break
  The chip is broken by the intermittent feed.

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **Drilling**
- Affected parameters: **F**, **S**

Unit G74 Drilling lin. pattern, front face C

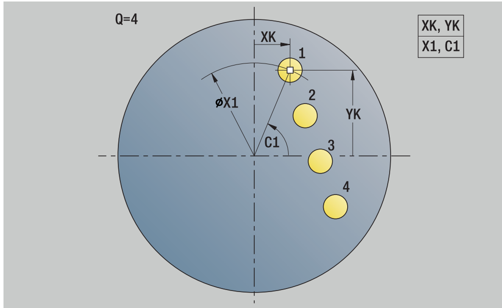
The unit machines a linear drilling pattern in which the individual features are arranged at a regular spacing on the front face.

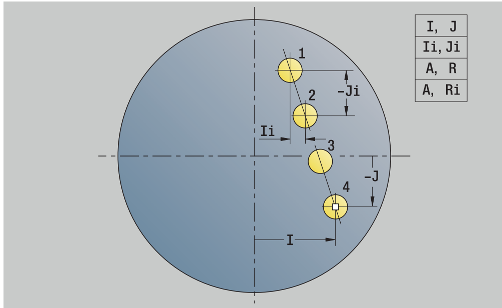
Unit name: **G74_Lin_Stirn_C** / cycle: **G74**

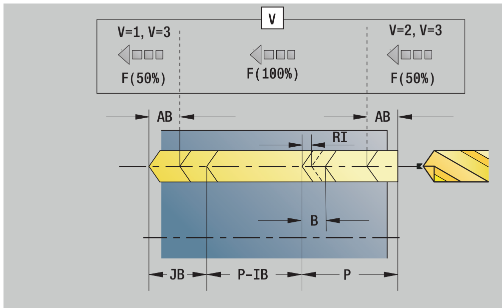
Further information: "Deep boring G74", Page 428

Pattern form:

- **Q**: **Number of holes**
- **X1**, **C1**: **Polar starting point** – Starting point of the pattern
- **XK**, **YK**: **Cartesian starting pnt**
- **I**, **J**: **End point (XK)** and **(YK)** – End point of the pattern (in Cartesian coordinates)
- **Ii**, **Ji**: **Distance (XKi)** and **(YKi)** – Incremental pattern spacing
- **R**: **Distance to first/last hole**
- **Ri**: **Length** – **Incremental distance**
- **A**: **Pattern ang.** (reference: XK axis)







Cycle form:

- **Z1**: **Start point drill**
- **Z2**: **End point drill**
- **E**: **Period of dwell** at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **DFF**: **Retraction feed rate**
- **V**: **Feed reduction**
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **AB**: **Drilling lengths** (default: 0)
- **P**: **1. Boring depth**
- **IB**: **Hole depth reduct. val** – value by which the hole depth decreases after every infeed.
- **JB**: **Minimum hole depth**
  If you have entered a hole depth reduction value, the hole depth is reduced only to the value entered in **JB**.
- **B**: **Return distance** – value by which the tool is retracted after reaching the respective hole depth
- **RI**: **Safety clearance** internal – distance for reapproach inside the hole (default: **Safety clearance SCK**)
- **RB**: **Return plane** (default: back to start position)

Global form:

- **G14**: **Tool change point**
  - **No axis**
  - **0: Simultaneously**
  - **1: First X, then Z**
  - **2: First Z, then X**
  - **3: Only X**
  - **4: Only Z**
  - **5: Only Y** (machine-dependent)
  - **6: Simultaneous w/ Y** (machine-dependent)
- **CLT**: **Coolant**
  - **0: Without**
  - **1: Coolant 1 on**
  - **2: Coolant 2 on**
- **SCK**: **Safety clearance** in infeed direction during drilling and milling operations
- **G60**: **Protection zone** – protection zone monitoring during drilling
  - **0: Active**
  - **1: Inactive**
- **BP**: **Interval time** – time during which the feed motion will be interrupted
  The chip is broken by the intermittent feed.
- **BF**: **Feed period** – time interval until the next break
  The chip is broken by the intermittent feed.

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

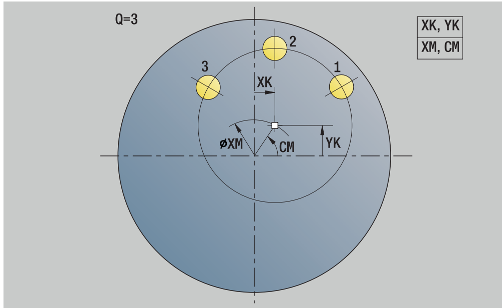
- Machining operation: **Drilling**
- Affected parameters: **F**, **S**

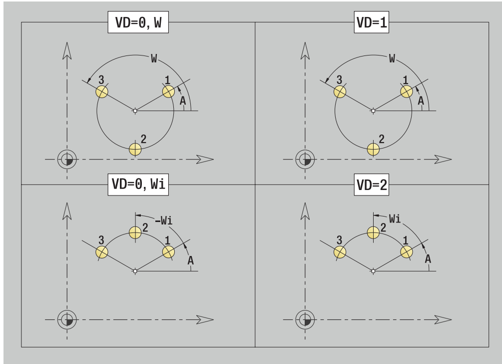
Unit G74 Drilling circ. pattern, front face C

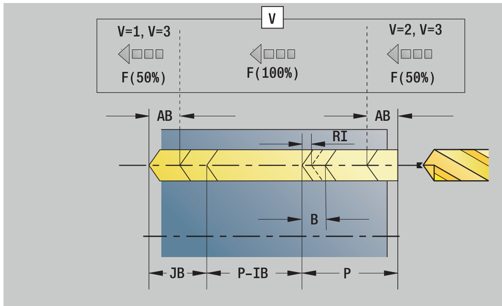
This unit machines a circular drilling pattern on the face of the workpiece.

Unit name: **G74_Bohr_Stirn_C** / cycle: **G74**

Further information: "Deep boring G74", Page 428







Pattern form:

- **Q**: **Number of holes**
- **XM**, **CM**: **Polar center point**
- **XK**, **YK**: **Cartesian center point**
- **A**: **Start angle**
- **Wi**: **End angle** – **Angle increment**
- **K**: **Pattern diameter**
- **W**: **Final angle**
- **VD**: **Rotation dir.** (default: 0)
  - **VD** = 0, without **W**: Figures are arranged on a full circle
  - **VD** = 0, with **W**: Figures are arranged on the longer circular arc
  - **VD** = 0, with **Wi**: The algebraic sign of **Wi** defines the direction (**Wi** < 0: clockwise)
  - **VD** = 1, with **W**: Clockwise
  - **VD** = 1, with **Wi**: Clockwise (algebraic sign of **Wi** has no effect)
  - **VD** = 2, with **W**: Counterclockwise
  - **VD** = 2, with **Wi**: Counterclockwise (algebraic sign of **Wi** has no effect)

Cycle form:

- **Z1**: **Start point drill**
- **Z2**: **End point drill**
- **E**: **Period of dwell** at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **DFF**: **Retraction feed rate**
- **V**: **Feed reduction**
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **AB**: **Drilling lengths** (default: 0)
- **P**: **1. Boring depth**
- **IB**: **Hole depth reduct. val** – value by which the hole depth decreases after every infeed.
- **JB**: **Minimum hole depth**
  If you have entered a hole depth reduction value, the hole depth is reduced only to the value entered in **JB**.
- **B**: **Return distance** – value by which the tool is retracted after reaching the respective hole depth
- **RI**: **Safety clearance** internal – distance for reapproach inside the hole (default: **Safety clearance SCK**)
- **RB**: **Return plane** (default: back to start position)

Global form:

- **G14**: **Tool change point**
  - **No axis**
  - **0: Simultaneously**
  - **1: First X, then Z**
  - **2: First Z, then X**
  - **3: Only X**
  - **4: Only Z**
  - **5: Only Y** (machine-dependent)
  - **6: Simultaneous w/ Y** (machine-dependent)
- **CLT**: **Coolant**
  - **0: Without**
  - **1: Coolant 1 on**
  - **2: Coolant 2 on**
- **SCK**: **Safety clearance** in infeed direction during drilling and milling operations
- **G60**: **Protection zone** – protection zone monitoring during drilling
  - **0: Active**
  - **1: Inactive**
- **BP**: **Interval time** – time during which the feed motion will be interrupted
  The chip is broken by the intermittent feed.
- **BF**: **Feed period** – time interval until the next break
  The chip is broken by the intermittent feed.

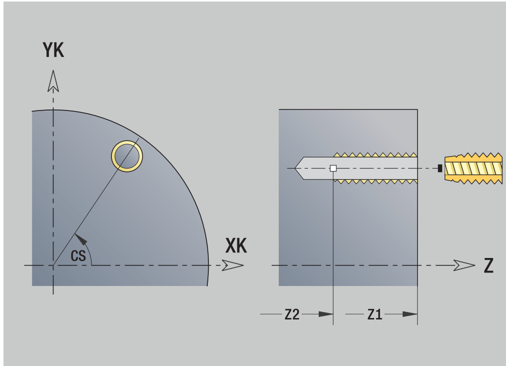
Further forms:

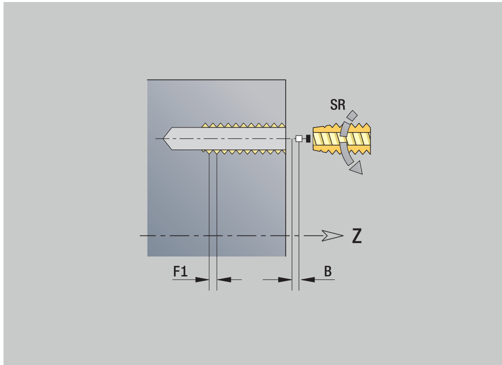
Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **Drilling**
- Affected parameters: **F**, **S**

Unit G73 Tapping, front face C





This unit machines a single tap hole on the face of the workpiece.

Unit name: **G73_Gew_Stirn_C** / cycle: **G73**

Further information: "Tapping G73", Page 426

Cycle form:

- **Z1**: **Start point drill**
- **Z2**: **End point drill**
- **CS**: **Spindle angle**
- **F1**: **Thread pitch**
- **B**: **Run-in lgth** to obtain the programmed spindle speed and feed rate (default: 2 * **Thread pitch F1**)
- **L**: **Retract length** when using floating tap holders (default: 0)
- **SR**: **Return speed** (default: tapping speed)
- **SP**: **Chip breaking depth**
- **SI**: **Retraction distance**

Further forms:

Further information: "smart.Turn unit", Page 98

Use the **Retract length** 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 tap is pulled away from the chuck by the retraction length. With this method, you can achieve higher service life of the taps.

Access to the technology database:

- Machining operation: **Tapping**
- Affected parameter: **S**

> If you interrupt program run during a tapping cycle, you can manually retract the tool from the hole in the Z axis. The control moves the spindle on a path matching the traverse.
>
> If the optional machine parameter **CfgBackTrack** (no. 122000) is active, use the **Start blck search** soft key to resume program run after the manual traverse.

Unit G73 Tapping linear pattern, front face C

The unit machines a linear tapping pattern in which the individual features are arranged at a regular spacing on the face.

Unit name: **G73_Lin_Stirn_C** / cycle: **G73**

Further information: "Tapping G73", Page 426

Pattern form:

- **Q**: **Number of holes**
- **X1**, **C1**: **Polar starting point** – Starting point of the pattern
- **XK**, **YK**: **Cartesian starting pnt**
- **I**, **J**: **End point (XK)** and **(YK)** – End point of the pattern (in Cartesian coordinates)
- **Ii**, **Ji**: **Distance (XKi)** and **(YKi)** – Incremental pattern spacing
- **R**: **Distance to first/last hole**
- **Ri**: **Length** – **Incremental distance**
- **A**: **Pattern ang.** (reference: XK axis)

Cycle form:

- **Z1**: **Start point drill**
- **Z2**: **End point drill**
- **F1**: **Thread pitch**
- **B**: **Run-in lgth** to obtain the programmed spindle speed and feed rate (default: 2 * **Thread pitch F1**)
- **L**: **Retract length** when using floating tap holders (default: 0)
- **SR**: **Return speed** (default: tapping speed)
- **SP**: **Chip breaking depth**
- **SI**: **Retraction distance**
- **RB**: **Return plane** (default: back to start position)

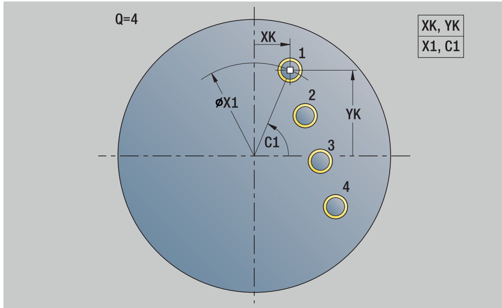
Further forms:

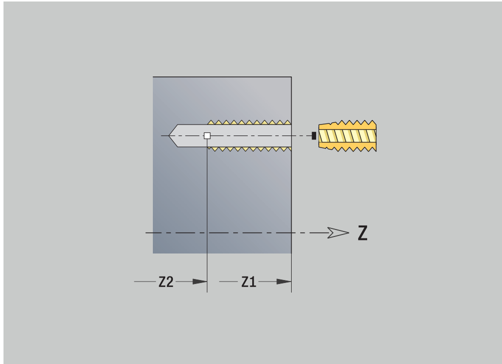
Further information: "smart.Turn unit", Page 98

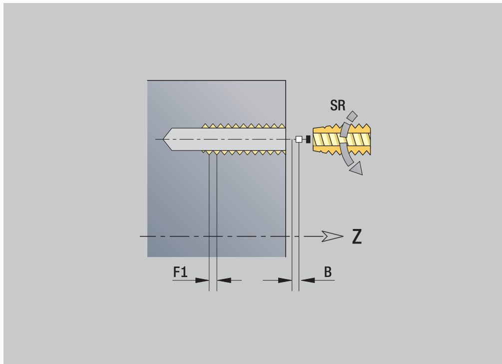
Use the **Retract length** for floating tap holders. The cycle calculates a new nominal pitch on the basis of the thread depth, the programmed pitch, and the retraction length. The nominal pitch is somewhat smaller than the pitch of the tap. During tapping, the tap is pulled away from the chuck by the retraction length. With this method, you can achieve higher service life of the taps.

Access to the technology database:

- Machining operation: **Tapping**
- Affected parameter: **S**

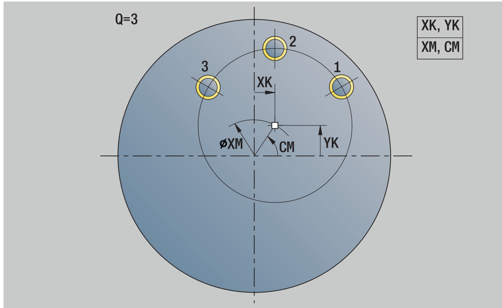


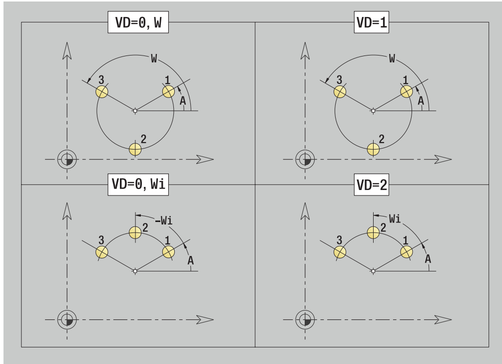


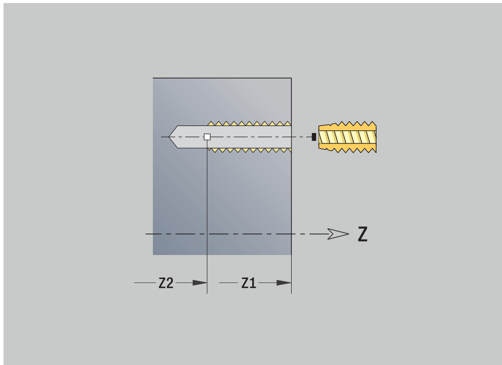


> If you interrupt program run during a tapping cycle, you can manually retract the tool from the hole in the Z axis. The control moves the spindle on a path matching the traverse.
>
> If the optional machine parameter **CfgBackTrack** (no. 122000) is active, use the **Start blck search** soft key to resume program run after the manual traverse.

Unit G73 Tapping circular pattern, front face C







This unit machines a circular tapping pattern on the front face of the workpiece.

Unit name: **G73_Cir_Stirn_C** / cycle: **G73**

Further information: "Tapping G73", Page 426

Pattern form:

- **Q**: **Number of holes**
- **XM**, **CM**: **Polar center point**
- **XK**, **YK**: **Cartesian center point**
- **A**: **Start angle**
- **Wi**: **End angle – Angle increment**
- **K**: **Pattern diameter**
- **W**: **Final angle**
- **VD**: **Rotation dir.** (default: 0)
  - **VD** = 0, without **W**: Figures are arranged on a full circle
  - **VD** = 0, with **W**: Figures are arranged on the longer circular arc
  - **VD** = 0, with **Wi**: The algebraic sign of **Wi** defines the direction (**Wi** < 0: clockwise)
  - **VD** = 1, with **W**: Clockwise
  - **VD** = 1, with **Wi**: Clockwise (algebraic sign of **Wi** has no effect)
  - **VD** = 2, with **W**: Counterclockwise
  - **VD** = 2, with **Wi**: Counterclockwise (algebraic sign of **Wi** has no effect)

Cycle form:

- **Z1**: **Start point drill**
- **Z2**: **End point drill**
- **F1**: **Thread pitch**
- **B**: **Run-in lgth** to obtain the programmed spindle speed and feed rate (default: 2 * **Thread pitch F1**)
- **L**: **Retract length** when using floating tap holders (default: 0)
- **SR**: **Return speed** (default: tapping speed)
- **SP**: **Chip breaking depth**
- **SI**: **Retraction distance**
- **RB**: **Return plane** (default: back to start position)

Further forms:

Further information: "smart.Turn unit", Page 98

Use the **Retract length** 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 tap is pulled away from the chuck by the retraction length. With this method, you can achieve higher service life of the taps.

Access to the technology database:

- Machining operation: **Tapping**
- Affected parameter: **S**

If you interrupt program run during a tapping cycle, you can manually retract the tool from the hole in the Z axis. The control moves the spindle on a path matching the traverse.

If the optional machine parameter **CfgBackTrack** (no. 122000) is active, use the **Start blck search** soft key to resume program run after the manual traverse.

Unit G74 Single hole, lat. surface C

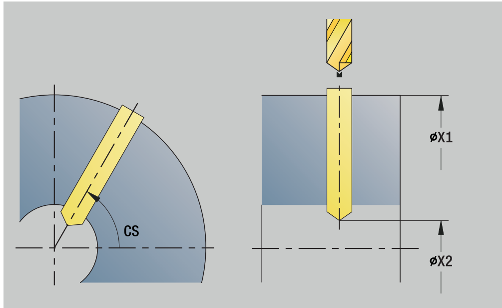
This unit machines a hole on the lateral surface of the workpiece.

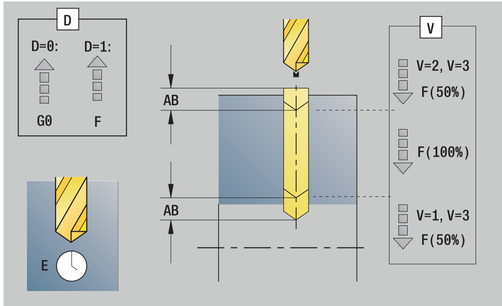
Unit name: **G74_Bohr_Mant_C** / cycle: **G74**

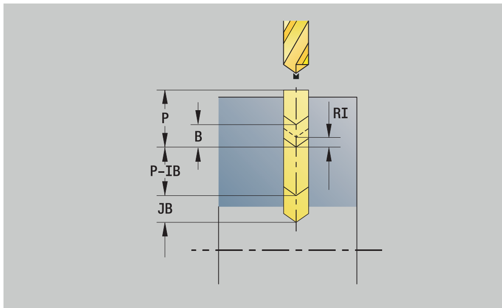
Further information: "Deep boring G74", Page 428

Cycle form:

- **X1**: **Start point drill** (diameter value)
- **X2**: **End point drill**
- **CS**: **Spindle angle**
- **E**: **Period of dwell** at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **DFF**: **Retraction feed rate**
- **V**: **Feed reduction**
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **AB**: **Drilling lengths** (default: 0)
- **P**: **1. Boring depth**
- **IB**: **Hole depth reduct. val** – value by which the hole depth decreases after every infeed.
- **JB**: **Minimum hole depth**
  If you have entered a hole depth reduction value, the hole depth is reduced only to the value entered in **JB**.
- **B**: **Return distance** – value by which the tool is retracted after reaching the respective hole depth
- **RI**: **Safety clearance** internal – distance for reapproach inside the hole (default: **Safety clearance SCK**)







Global form:

- **G14**: **Tool change point**
  - **No axis**
  - **0: Simultaneously**
  - **1: First X, then Z**
  - **2: First Z, then X**
  - **3: Only X**
  - **4: Only Z**
  - **5: Only Y** (machine-dependent)
  - **6: Simultaneous w/ Y** (machine-dependent)
- **CLT**: **Coolant**
  - **0: Without**
  - **1: Coolant 1 on**
  - **2: Coolant 2 on**
- **SCK**: **Safety clearance** in infeed direction during drilling and milling operations
- **BP**: **Interval time** – time during which the feed motion will be interrupted
  The chip is broken by the intermittent feed.

- **BF**: **Feed period** – time interval until the next break
  The chip is broken by the intermittent feed.
- **CB**: **Brake off (1)**

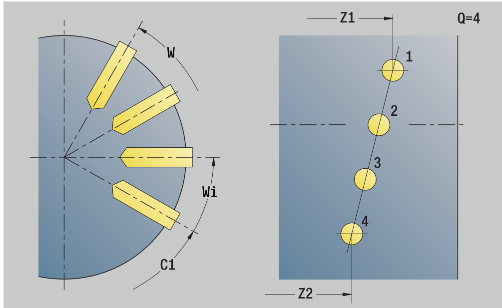
Further forms:

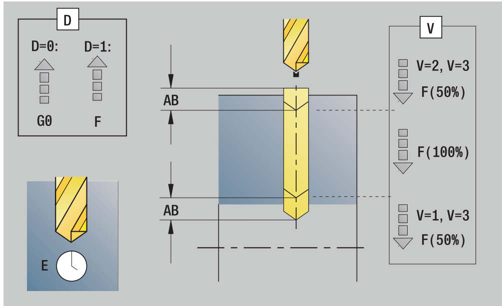
Further information: "smart.Turn unit", Page 98

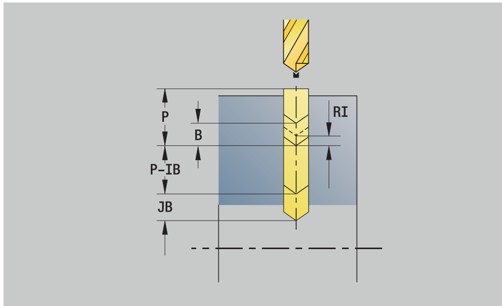
Access to the technology database:

- Machining operation: **Drilling**
- Affected parameters: **F**, **S**

Unit G74 Drilling lin. patt., lat. surface C







The unit machines a linear drilling pattern in which the individual features are arranged at a regular spacing on the lateral surface.

Unit name: **G74_Lin_Mant_C** / cycle: **G74**

Further information: "Deep boring G74", Page 428

Pattern form:

- **Q**: **Number of holes**
- **Z1**: **Start point pattern** – Position of first hole
- **C1**: **Start. angle**
- **Wi**: **End angle** – **Angle increment**
- **W**: **Final angle**
- **Z2**: **End point pattern**

Cycle form:

- **X1**: **Start point drill** (diameter value)
- **X2**: **End point drill**
- **E**: **Period of dwell** at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **DFF**: **Retraction feed rate**
- **V**: **Feed reduction**
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **AB**: **Drilling lengths** (default: 0)
- **P**: **1. Boring depth**
- **IB**: **Hole depth reduct. val** – value by which the hole depth decreases after every infeed.
- **JB**: **Minimum hole depth**
  If you have entered a hole depth reduction value, the hole depth is reduced only to the value entered in **JB**.
- **B**: **Return distance** – value by which the tool is retracted after reaching the respective hole depth
- **RI**: **Safety clearance** internal – distance for reapproach inside the hole (default: **Safety clearance SCK**)
- **RB**: **Return plane** (default: back to start position)

Global form:

- **G14**: **Tool change point**
  - **No axis**
  - **0: Simultaneously**
  - **1: First X, then Z**
  - **2: First Z, then X**
  - **3: Only X**
  - **4: Only Z**
  - **5: Only Y** (machine-dependent)
  - **6: Simultaneous w/ Y** (machine-dependent)

- **CLT**: **Coolant**
  - **0: Without**
  - **1: Coolant 1 on**
  - **2: Coolant 2 on**
- **SCK**: **Safety clearance** in infeed direction during drilling and milling operations
- **BP**: **Interval time** – time during which the feed motion will be interrupted
  The chip is broken by the intermittent feed.
- **BF**: **Feed period** – time interval until the next break
  The chip is broken by the intermittent feed.
- **CB**: **Brake off (1)**

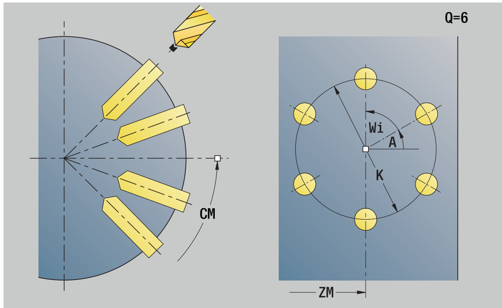
Further forms:

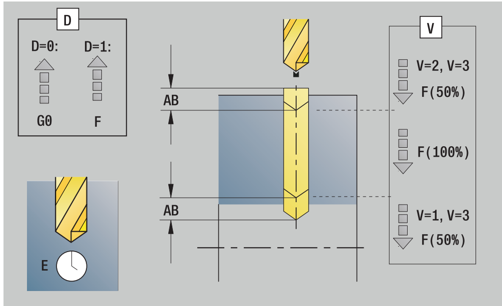
Further information: "smart.Turn unit", Page 98

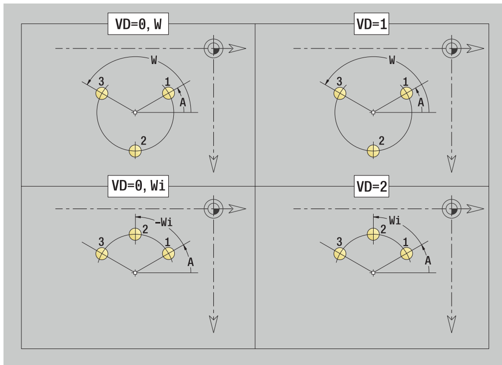
Access to the technology database:

- Machining operation: **Drilling**
- Affected parameters: **F**, **S**

Unit G74 Drilling circ. patt., lat. surface C







This unit machines a circular hole pattern on the lateral surface of the workpiece.

Unit name: **G74_Cir_Mant_C** / cycle: **G74**

Further information: "Deep boring G74", Page 428

Pattern form:

- **Q**: **Number of holes**
- **ZM**: **Center** of pattern
- **CM**: **Angle of pattern center**
- **A**: **Start angle**
- **Wi**: **End angle** – **Angle increment**
- **K**: **Pattern diameter**
- **W**: **Final angle**
- **VD**: **Rotation dir.** (default: 0)
  - **VD** = 0, without **W**: Figures are arranged on a full circle
  - **VD** = 0, with **W**: Figures are arranged on the longer circular arc
  - **VD** = 0, with **Wi**: The algebraic sign of **Wi** defines the direction (**Wi** < 0: clockwise)
  - **VD** = 1, with **W**: Clockwise
  - **VD** = 1, with **Wi**: Clockwise (algebraic sign of **Wi** has no effect)
  - **VD** = 2, with **W**: Counterclockwise
  - **VD** = 2, with **Wi**: Counterclockwise (algebraic sign of **Wi** has no effect)

Cycle form:

- **X1**: **Start point drill** (diameter value)
- **X2**: **End point drill**
- **E**: **Period of dwell** at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **DFF**: **Retraction feed rate**
- **V**: **Feed reduction**
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **AB**: **Drilling lengths** (default: 0)
- **P**: **1. Boring depth**
- **IB**: **Hole depth reduct. val** – value by which the hole depth decreases after every infeed.
- **JB**: **Minimum hole depth**
  If you have entered a hole depth reduction value, the hole depth is reduced only to the value entered in **JB**.
- **B**: **Return distance** – value by which the tool is retracted after reaching the respective hole depth
- **RI**: **Safety clearance** internal – distance for reapproach inside the hole (default: **Safety clearance SCK**)
- **RB**: **Return plane** (default: back to start position)

Global form:

- **G14**: **Tool change point**
  - **No axis**
  - **0: Simultaneously**
  - **1: First X, then Z**
  - **2: First Z, then X**
  - **3: Only X**
  - **4: Only Z**
  - **5: Only Y** (machine-dependent)
  - **6: Simultaneous w/ Y** (machine-dependent)
- **CLT**: **Coolant**
  - **0: Without**
  - **1: Coolant 1 on**
  - **2: Coolant 2 on**
- **SCK**: **Safety clearance** in infeed direction during drilling and milling operations
- **BP**: **Interval time** – time during which the feed motion will be interrupted
  The chip is broken by the intermittent feed.
- **BF**: **Feed period** – time interval until the next break
  The chip is broken by the intermittent feed.
- **CB**: **Brake off (1)**

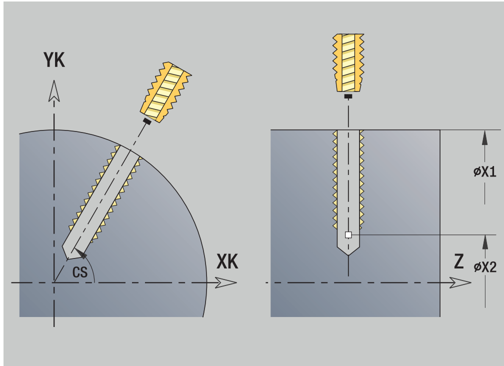
Further forms:

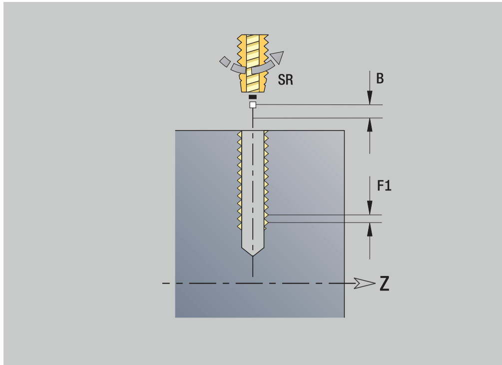
Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **Drilling**
- Affected parameters: **F**, **S**

Unit G73 Tap hole, lateral surface C





This unit machines a tap hole on the lateral surface of the workpiece.

Unit name: **G73_Gew_Mant_C** / cycle: **G73**

Further information: "Tapping G73", Page 426

Cycle form:

- **X1**: **Start point drill** (diameter value)
- **X2**: **End point drill**
- **CS**: **Spindle angle**
- **F1**: **Thread pitch**
- **B**: **Run-in lgth** to obtain the programmed spindle speed and feed rate (default: 2 * **Thread pitch F1**)
- **L**: **Retract length** when using floating tap holders (default: 0)
- **SR**: **Return speed** (default: tapping speed)
- **SP**: **Chip breaking depth**
- **SI**: **Retraction distance**

Further forms:

Further information: "smart.Turn unit", Page 98

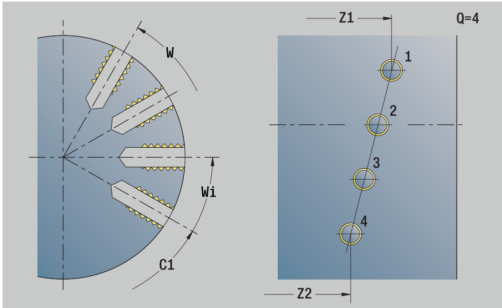
Use the **Retract length** for floating tap holders. The cycle calculates a new nominal pitch on the basis of the thread depth, the programmed pitch, and the retraction length. The nominal pitch is somewhat smaller than the pitch of the tap. During tapping, the tap is pulled away from the chuck by the retraction length. With this method, you can achieve higher service life of the taps.

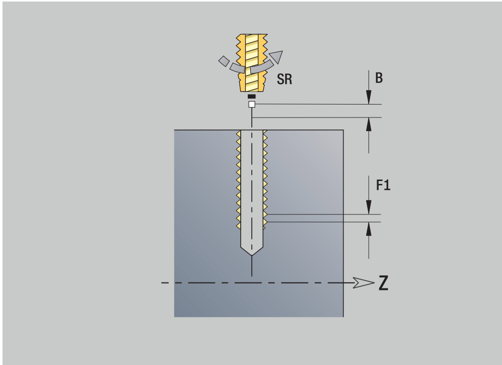
Access to the technology database:

- Machining operation: **Tapping**
- Affected parameter: **S**

> If you interrupt program run during a tapping cycle, you can manually retract the tool from the hole in the Z axis. The control moves the spindle on a path matching the traverse.
>
> If the optional machine parameter **CfgBackTrack** (no. 122000) is active, use the **Start blck search** soft key to resume program run after the manual traverse.

Unit G73 Tapping linear pattern, cylindrical C





The unit machines a linear tapping pattern in which the individual features are arranged at a regular spacing on the lateral surface.

Unit name: **G73_Lin_Mant_C** / cycle: **G73**

Further information: "Tapping G73", Page 426

Pattern form:

- **Q**: **Number of holes**
- **Z1**: **Start point pattern** – Position of first hole
- **C1**: **Start. angle**
- **Wi**: **End angle** – **Angle increment**
- **W**: **Final angle**
- **Z2**: **End point pattern**

Cycle form:

- **X1**: **Start point drill** (diameter value)
- **X2**: **End point drill**
- **F1**: **Thread pitch**
- **B**: **Run-in lgth** to obtain the programmed spindle speed and feed rate (default: 2 * **Thread pitch F1**)
- **L**: **Retract length** when using floating tap holders (default: 0)
- **SR**: **Return speed** (default: tapping speed)
- **SP**: **Chip breaking depth**
- **SI**: **Retraction distance**
- **RB**: **Return plane** (default: back to start position)

Further forms:

Further information: "smart.Turn unit", Page 98

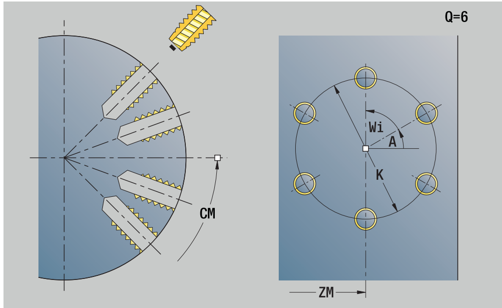
Use the **Retract length** for floating tap holders. The cycle calculates a new nominal pitch on the basis of the thread depth, the programmed pitch, and the retraction length. The nominal pitch is somewhat smaller than the pitch of the tap. During tapping, the tap is pulled away from the chuck by the retraction length. With this method, you can achieve higher service life of the taps.

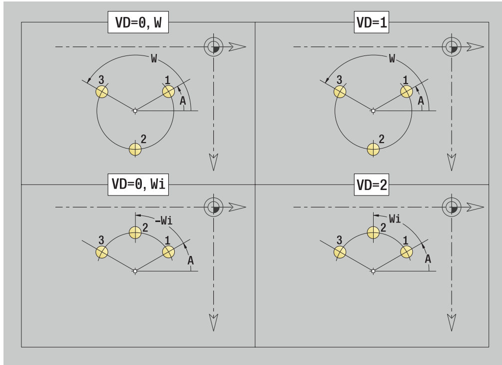
Access to the technology database:

- Machining operation: **Tapping**
- Affected parameter: **S**

> If you interrupt program run during a tapping cycle, you can manually retract the tool from the hole in the Z axis. The control moves the spindle on a path matching the traverse.
>
> If the optional machine parameter **CfgBackTrack** (no. 122000) is active, use the **Start blck search** soft key to resume program run after the manual traverse.

Unit G73 Tapping circ. pattern, lateral surf. C





This unit machines a circular tapping pattern on the lateral surface of the workpiece.

Unit name: **G73_Cir_Mant_C** / cycle: **G73**

Further information: "Tapping G73", Page 426

Pattern form:

- **Q**: **Number of holes**
- **ZM**: **Center** of pattern
- **CM**: **Angle of pattern center**
- **A**: **Start angle**
- **Wi**: **End angle** – **Angle increment**
- **K**: **Pattern diameter**
- **W**: **Final angle**
- **VD**: **Rotation dir.** (default: 0)
  - **VD** = 0, without **W**: Figures are arranged on a full circle
  - **VD** = 0, with **W**: Figures are arranged on the longer circular arc
  - **VD** = 0, with **Wi**: The algebraic sign of **Wi** defines the direction (**Wi** < 0: clockwise)
  - **VD** = 1, with **W**: Clockwise
  - **VD** = 1, with **Wi**: Clockwise (algebraic sign of **Wi** has no effect)
  - **VD** = 2, with **W**: Counterclockwise
  - **VD** = 2, with **Wi**: Counterclockwise (algebraic sign of **Wi** has no effect)

Cycle form:

- **X1**: **Start point drill** (diameter value)
- **X2**: **End point drill**
- **F1**: **Thread pitch**
- **B**: **Run-in lgth** to obtain the programmed spindle speed and feed rate (default: 2 * **Thread pitch F1**)
- **L**: **Retract length** when using floating tap holders (default: 0)
- **SR**: **Return speed** (default: tapping speed)
- **SP**: **Chip breaking depth**
- **SI**: **Retraction distance**
- **RB**: **Return plane**

Further forms:

Further information: "smart.Turn unit", Page 98

Use the **Retract length** 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 tap is pulled away from the chuck by the retraction length. With this method, you can achieve higher service life of the taps.

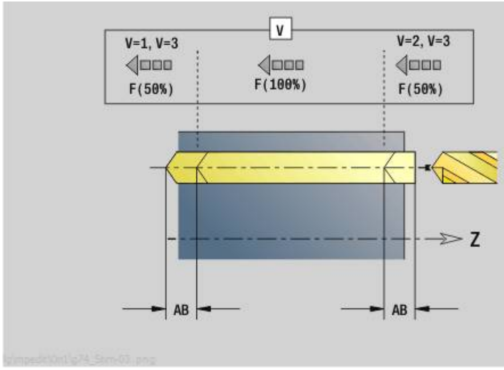
Access to the technology database:

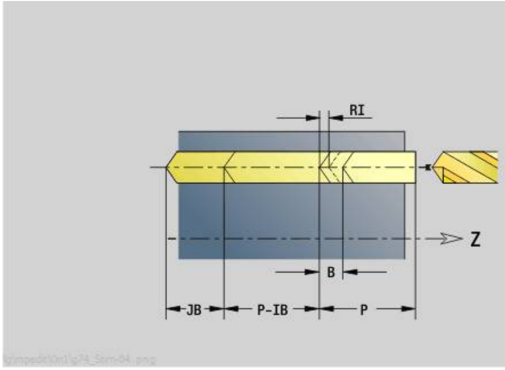
- Machining operation: **Tapping**
- Affected parameter: **S**

If you interrupt program run during a tapping cycle, you can manually retract the tool from the hole in the Z axis. The control moves the spindle on a path matching the traverse.

If the optional machine parameter **CfgBackTrack** (no. 122000) is active, use the **Start blck search** soft key to resume program run after the manual traverse.

Unit G74 Drilling in ICP C (option 55)





The unit machines a single hole or a hole pattern on the face or lateral surface. Using **ICP**, you define the positions of the holes as well as further details.

Unit name: **G74_ICP_C** / cycle: **G74**

Further information: "Deep boring G74", Page 428

Pattern form:

- **FK**: **No. of ICP finished part** – name of the contour to be machined
- **NS**: **Starting block no. of contour** – beginning of contour section

Cycle form:

- **E**: **Period of dwell** at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **DFF**: **Retraction feed rate**
- **V**: **Feed reduction**
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **AB**: **Drilling lengths** (default: 0)
- **P**: **1. Boring depth**
- **IB**: **Hole depth reduct. val** – value by which the hole depth decreases after every infeed.
- **JB**: **Minimum hole depth**
  If you have entered a hole depth reduction value, the hole depth is reduced only to the value entered in **JB**.
- **B**: **Return distance** – value by which the tool is retracted after reaching the respective hole depth
- **RI**: **Safety clearance** internal – distance for reapproach inside the hole (default: **Safety clearance SCK**)
- **RB**: **Return plane** (default: back to start position)

Global form:

- **G14**: **Tool change point**
  - **No axis**
  - **0: Simultaneously**
  - **1: First X, then Z**
  - **2: First Z, then X**
  - **3: Only X**
  - **4: Only Z**
  - **5: Only Y** (machine-dependent)
  - **6: Simultaneous w/ Y** (machine-dependent)
- **CLT**: **Coolant**
  - **0: Without**
  - **1: Coolant 1 on**
  - **2: Coolant 2 on**

- **SCK**: **Safety clearance** in infeed direction during drilling and milling operations
- **CB**: **Brake off (1)**

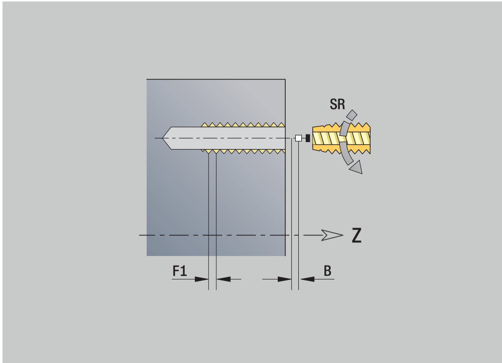
Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **Drilling**
- Affected parameters: **F**, **S**

Unit G73 Tapping in ICP C (option 55)



The unit machines a single tap hole or a tapping pattern on the face or lateral surface. Using **ICP**, you define the positions of the tap holes as well as further details.

Unit name: **G73_ICP_C** / cycle: **G73**

Further information: "Tapping G73", Page 426

Pattern form:

- **FK**: **No. of ICP finished part** – name of the contour to be machined
- **NS**: **Starting block no. of contour** – beginning of contour section

Cycle form:

- **F1**: **Thread pitch**
- **B**: **Run-in lgth** to obtain the programmed spindle speed and feed rate (default: 2 * **Thread pitch F1**)
- **L**: **Retract length** when using floating tap holders (default: 0)
- **SR**: **Return speed** (default: tapping speed)
- **SP**: **Chip breaking depth**
- **SI**: **Retraction distance**
- **RB**: **Return plane**

Further forms:

Further information: "smart.Turn unit", Page 98

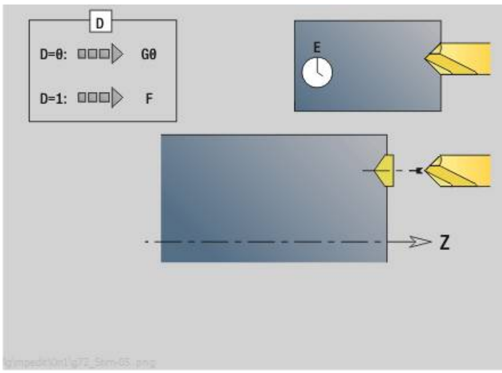
Use the **Retract length** for floating tap holders. The cycle calculates a new nominal pitch on the basis of the thread depth, the programmed pitch, and the retraction length. The nominal pitch is somewhat smaller than the pitch of the tap. During tapping, the tap is pulled away from the chuck by the retraction length. With this method, you can achieve higher service life of the taps.

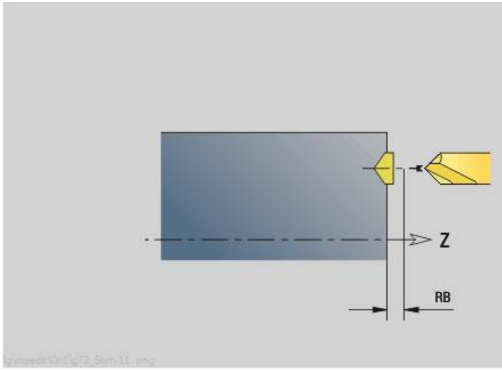
Access to the technology database:

- Machining operation: **Tapping**
- Affected parameter: **S**

> If you interrupt program run during a tapping cycle, you can manually retract the tool from the hole in the Z axis. The control moves the spindle on a path matching the traverse.
>
> If the optional machine parameter **CfgBackTrack** (no. 122000) is active, use the **Start blck search** soft key to resume program run after the manual traverse.

Unit G72 Boring, cntrsinkg ICP C (option 55)





The unit machines a single hole or a hole pattern on the face or lateral surface. Using **ICP**, you define the hole positions as well as further details for boring or countersinking.

Unit name: **G72_ICP_C** / cycle: **G72**

Further information: "Boring/cnt-sink G72", Page 425

Pattern form:

- **FK**: **No. of ICP finished part** – name of the contour to be machined
- **NS**: **Starting block no. of contour** – beginning of contour section

Cycle form:

- **E**: **Period of dwell** at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **DFF**: **Retraction feed rate**
- **RB**: **Return plane** (default: back to start position)

Further forms:

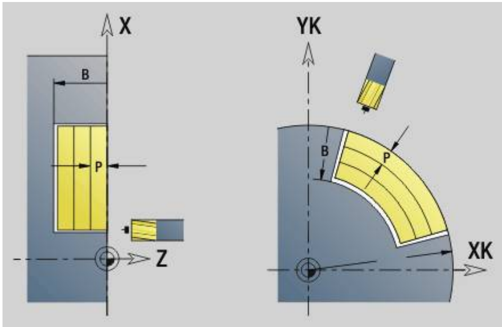
Further information: "smart.Turn unit", Page 98

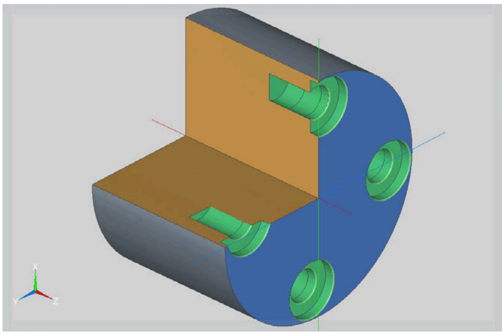
Access to the technology database:

- Machining operation: **Drilling**
- Affected parameters: **F**, **S**

Units G75 Bore milling ICP C (option 55)

Unit G75 Bore milling ICP C face





The unit machines a single hole or a hole pattern on the face. Using **ICP**, you define the positions of the holes as well as further details.

Unit name: **G75_BF_ICP_C** / cycle: **G75**

Further information: "Bore milling G75", Page 431

Contour form:

- **FK**: **Finished part contour** – name of the contour to be machined
- **NS**: **Starting block no. of contour** – beginning of contour section
- **FZ**: **Approach feed** (default: active feed rate)
- **B**: **Milling depth** (default: depth from the contour description)

Cycle form:

- **QK**: **Machining operation**
  - **0: Roughing**
  - **1: Finishing**
  - **2: Roughing and finishing**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Max. approach** (default: milling in one infeed)
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **WB**: **Diameter of the helix** (default: helix diameter = 1.5 * milling diameter)
- **EW**: **Plunging angle**
- **U**: **Overlap factor** – overlap of milling paths = **U** * milling diameter (default: 0.5)
- **RB**: **Return plane** (default: retract to starting position or to safety clearance; diameter value with radial holes and holes in the YZ plane)

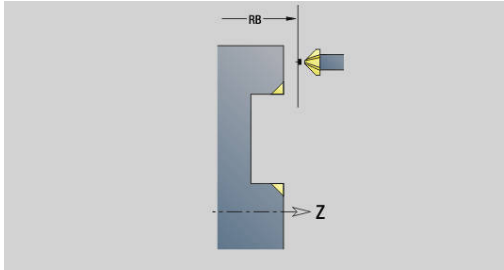
Further forms:

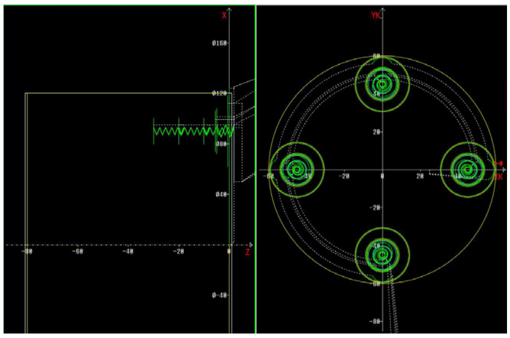
Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G75 Deburring ICP C face





The unit deburs a single hole or a hole pattern on the face. Using **ICP**, you define the positions of the holes as well as further details.

Unit name: **G75_EN_ICP_C** / cycle: **G75**

Further information: "Bore milling G75", Page 431

Contour form:

- **FK**: **Finished part contour** – name of the contour to be machined
- **NS**: **Starting block no. of contour** – beginning of contour section
- **B**: **Milling depth** (default: countersinking depth from the contour definition)

Cycle form:

- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **RB**: **Return plane** (default: retract to starting position or to safety clearance; diameter value with radial holes and holes in the YZ plane)

Further forms:

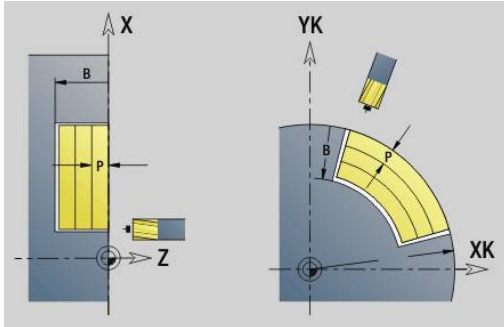
Further information: "smart.Turn unit", Page 98

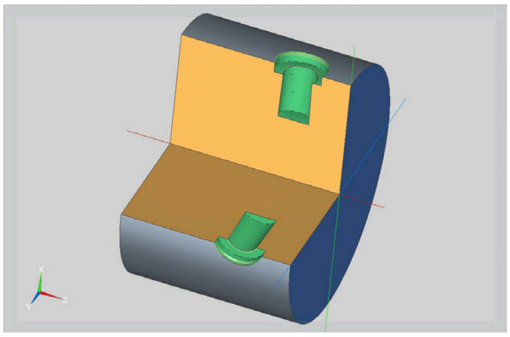
Access to the technology database:

- Machining operation: **Deburring**
- Affected parameters: **F**, **S**

Unit G75 Bore milling ICP C lateral

The unit machines a single hole or a hole pattern on the lateral surface. Using **ICP**, you define the positions of the holes as well as further details.





> This cycle produces oval contours on the lateral surface, and not circles.
> Circles are machined when the Y axis is used.
> **Further information:** "Units G75 Bore milling Y", Page 239

Unit name: **G75_BF_ICP_C_MANT** / cycle: **G75**

Further information: "Bore milling G75", Page 431

Contour form:

- **FK**: **Finished part contour** – name of the contour to be machined
- **NS**: **Starting block no. of contour** – beginning of contour section
- **FZ**: **Approach feed** (default: active feed rate)
- **B**: **Milling depth** (default: depth from the contour description)

Cycle form:

- **QK**: **Machining operation**
  - **0: Roughing**
  - **1: Finishing**
  - **2: Roughing and finishing**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Max. approach** (default: milling in one infeed)
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **WB**: **Diameter of the helix** (default: helix diameter = 1.5 * milling diameter)
- **EW**: **Plunging angle**
- **U**: **Overlap factor** – overlap of milling paths = **U** * milling diameter (default: 0.5)
- **RB**: **Return plane** (default: retract to starting position or to safety clearance; diameter value with radial holes and holes in the YZ plane)

Further forms:

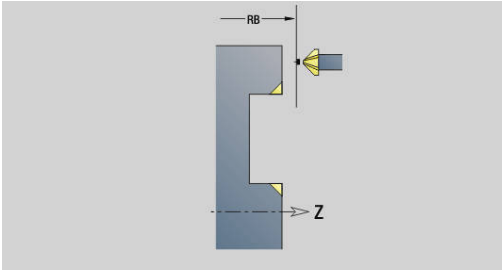
Further information: "smart.Turn unit", Page 98

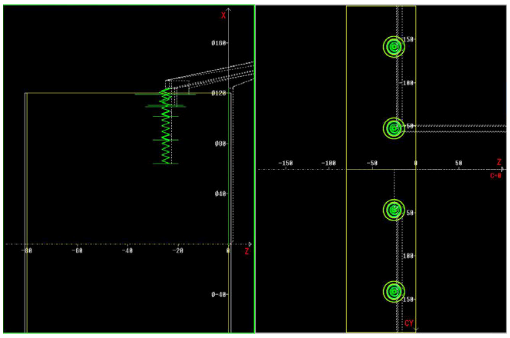
Access to the technology database:

- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G75 Deburring ICP C lateral

The unit deburs a single hole or a hole pattern on the lateral surface. Using **ICP**, you define the positions of the holes as well as further details.





> This cycle produces oval contours on the lateral surface, and not circles.
>
> Circles are machined when the Y axis is used.
> **Further information:** "Units G75 Bore milling Y", Page 239

Unit name: **G75_EN_ICP_C_MANT** / cycle: **G75**

Further information: "Bore milling G75", Page 431

Contour form:

- **FK**: **Finished part contour** – name of the contour to be machined
- **NS**: **Starting block no. of contour** – beginning of contour section
- **B**: **Milling depth** (default: countersinking depth from the contour definition)

Cycle form:

- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **RB**: **Return plane** (default: retract to starting position or to safety clearance; diameter value with radial holes and holes in the YZ plane)

Further forms:

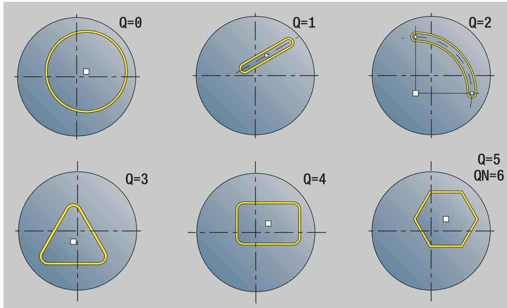
Further information: "smart.Turn unit", Page 98

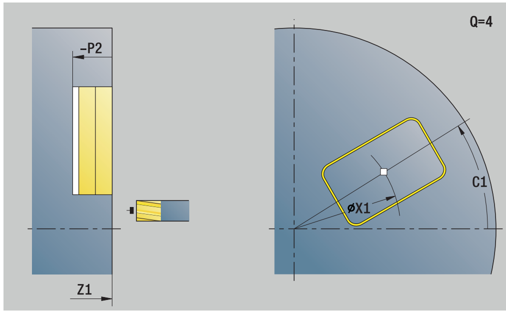
Access to the technology database:

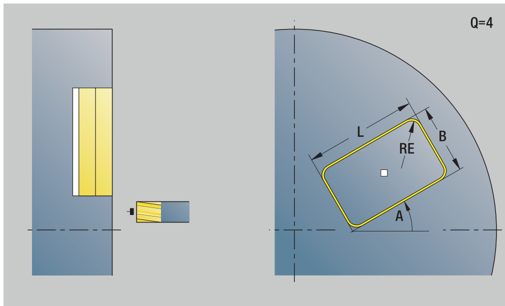
- Machining operation: **Deburring**
- Affected parameters: **F**, **S**

4.6 Units – Drillg / Predrilling, milling in C (option 55)

Unit G840 Predrill, contour mill, figure on face in C







The unit determines the hole position and machines the hole. The subsequent milling cycle obtains the hole position from the reference stored in **NF**.

Unit name: **DRILL_STI_KON_C** / cycles: **G840 A1**; **G71**

Further information: "G840 – calculating hole positions", Page 464

Further information: "Simple drilling G71", Page 423

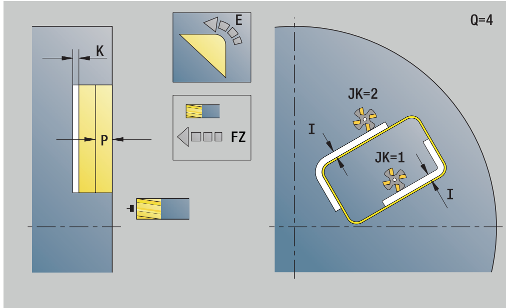
Figure form:

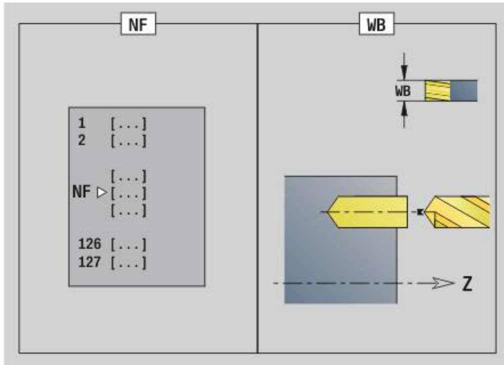
- **Q**: **Type of figure**
  - **0: Complete circle**
  - **1: Linear slot**
  - **2: Circular slot**
  - **3: Triangle**
  - **4: Rectangle / square**
  - **5: Polygon**
- **QN**: **No. of polygon corners** (only with **Q** = **5: Polygon**)
- **X1**: **Dia. of figure center**
- **C1**: **Angle of figure center** (default: **Spindle angle C**)
- **Z1**: **Millg. top edge** (default: **Starting point Z**)
- **P2**: **Depth of figure**
- **L**: **+edge lngth/-width a. flats**
  - **L** > 0: **Edge length**
  - **L** < 0: **Width across** (inside diameter) for polygon
- **B**: **Width of rectangle**
- **RE**: **Rounding radius** (default: 0)
- **A**: **Angle to X axis** (default: 0°)
- **Q2**: **Rot. direction of slot** (only with **Q** = **2: Circular slot**)
  - **cw**: In clockwise direction
  - **ccw**: In counterclockwise direction
- **W**: **Angle of slot end point** (only with **Q** = **2: Circular slot**)

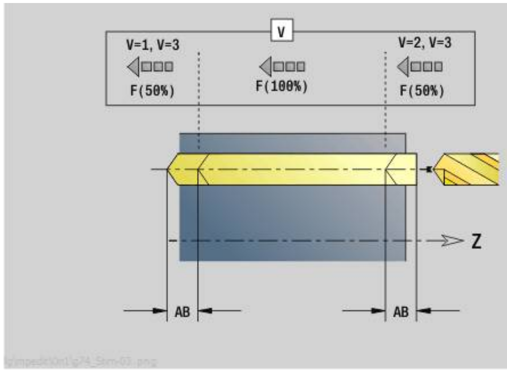
> ℹ Program only the parameters relevant to the selected figure type.

Cycle form:

- **JK**: **Cutter position**
  - **0: On the contour**
  - **1: Within the contour**
  - **2: Outside the contour**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **R**: **Apprch angle** (default: 0)
- **WB**: **Milling diameter**
- **NF**: **Position mark** – reference at which the cycle stores the predrill hole positions (range: 1 to 127)
- **E**: **Period of dwell** at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **V**: **Feed reduction**
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **AB**: **Drilling lengths** (default: 0)
- **RB**: **Return plane** (default: back to start position)







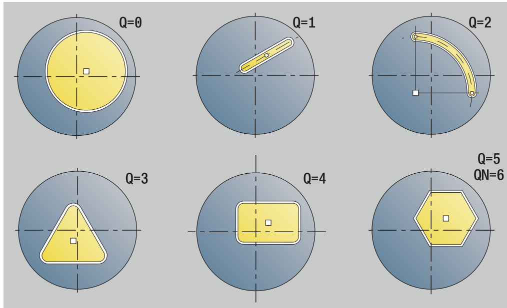
Further forms:

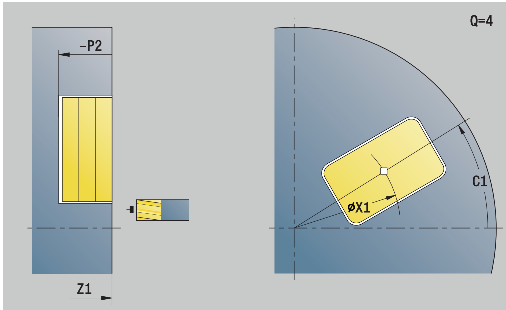
Further information: "smart.Turn unit", Page 98

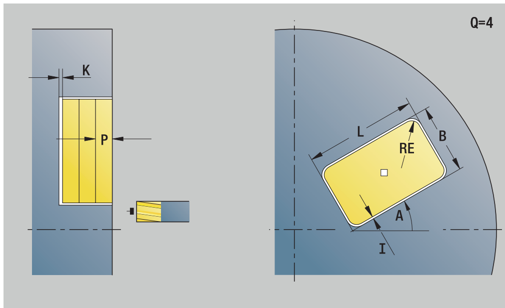
Access to the technology database:

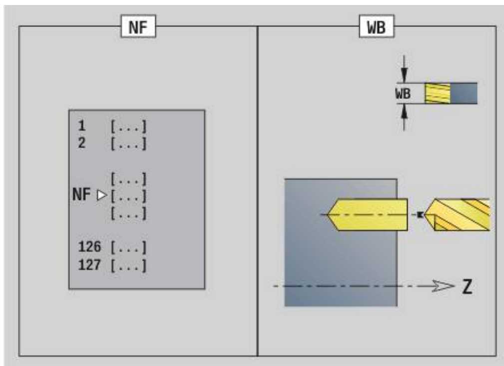
- Machining operation: **Drilling**
- Affected parameters: **F**, **S**

Unit G845
Predrill, pocket mill, figure on face in C









The unit determines the hole position and machines the hole. The subsequent milling cycle obtains the hole position from the reference stored in **NF**.

Unit name: **DRILL_STI_TASC** / cycles: **G845**; **G71**

Overvw. form:

- **AP**: **Predrilling position**
  - **1: Determine predrilling pos**
  - **2: predrill. pos. fig. center**

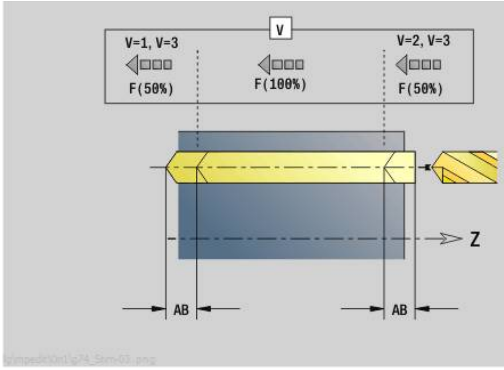
Further information: "G845 – calculating hole positions", Page 473

Further information: "Simple drilling G71", Page 423

Figure form:

- **Q**: **Type of figure**
  - **0: Complete circle**
  - **1: Linear slot**
  - **2: Circular slot**
  - **3: Triangle**
  - **4: Rectangle / square**
  - **5: Polygon**
- **QN**: **No. of polygon corners** (only with **Q** = **5: Polygon**)
- **X1**: **Dia. of figure center**
- **C1**: **Angle of figure center** (default: **Spindle angle C**)
- **Z1**: **Millg. top edge** (default: **Starting point Z**)
- **P2**: **Depth of figure**
- **L**: **+edge lngth/-width a. flats**
  - **L** > 0: **Edge length**
  - **L** < 0: **Width across** (inside diameter) for polygon
- **B**: **Width of rectangle**
- **RE**: **Rounding radius** (default: 0)
- **A**: **Angle to X axis** (default: 0°)
- **Q2**: **Rot. direction of slot** (only with **Q** = **2: Circular slot**)
  - **cw**: In clockwise direction
  - **ccw**: In counterclockwise direction
- **W**: **Angle of slot end point** (only with **Q** = **2: Circular slot**)

> Program only the parameters relevant to the selected figure type.



Cycle form:

- **JT**: **Machining direction**
  - **0: From the inside out**
  - **1: From the outside in**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
- **WB**: **Milling diameter**
- **NF**: **Position mark** – reference at which the cycle stores the predrill hole positions (range: 1 to 127)
- **E**: **Period of dwell** at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **V**: **Feed reduction**
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **AB**: **Drilling lengths** (default: 0)
- **RB**: **Return plane** (default: back to start position)

Further forms:

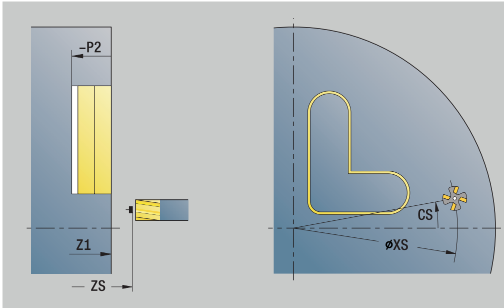
Further information: "smart.Turn unit", Page 98

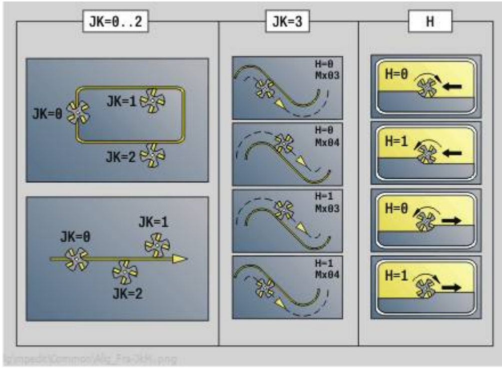
Access to the technology database:

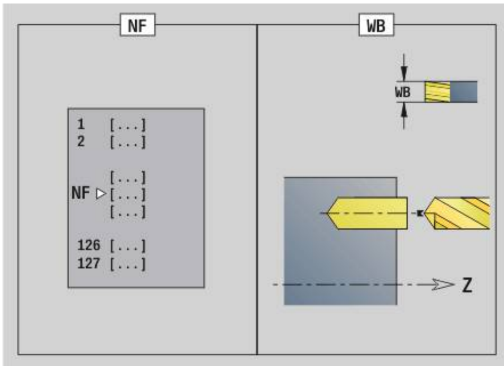
- Machining operation: **Drilling**
- Affected parameters: **F**, **S**

Unit G840 Predrill, contour mill, ICP on face in C

The unit determines the hole position and machines the hole. The subsequent milling cycle obtains the hole position from the reference stored in **NF**. If the milling contour consists of multiple sections, the unit machines a hole for each section.







Unit name: **DRILL_STI_840_C** / cycles: **G840 A1**; **G71**

Further information: "G840 – calculating hole positions", Page 464

Further information: "Simple drilling G71", Page 423

Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
- **Z1**: **Millg. top edge** (default: **Starting point Z**)
- **P2**: **Depth of contour**

Cycle form:

- **JK**: **Cutter position**
  - **0: On the contour**
  - **1: Within/left of contour**
  - **2: Outside/right of contour**
  - **3: Depending on H and MD**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **R**: **Apprch angle** (default: 0)
- **WB**: **Milling diameter**
- **NF**: **Position mark** – reference at which the cycle stores the predrill hole positions (range: 1 to 127)
- **E**: **Period of dwell** at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **V**: **Feed reduction**
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **AB**: **Drilling lengths** (default: 0)
- **RB**: **Return plane** (default: back to start position)

Further forms:

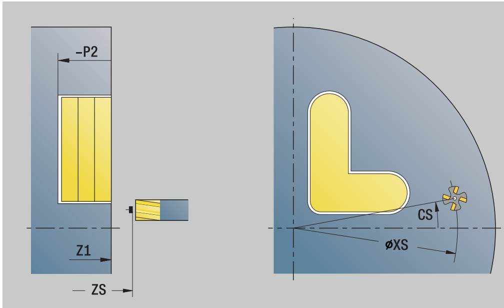
Further information: "smart.Turn unit", Page 98

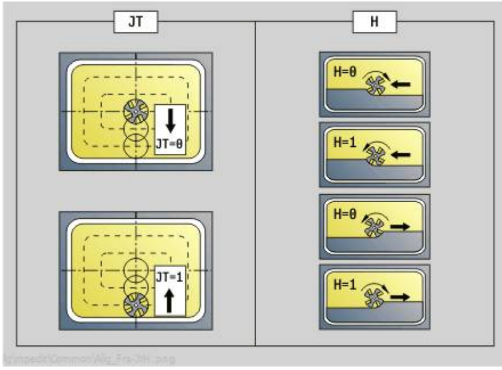
Access to the technology database:

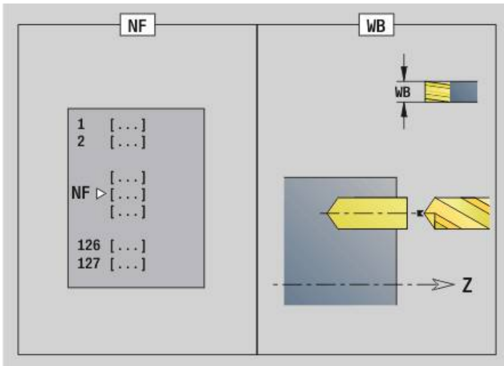
- Machining operation: **Drilling**
- Affected parameters: **F**, **S**

Unit G845 Predrill, pocket mill, ICP on face in C

The unit determines the hole position and machines the hole. The subsequent milling cycle obtains the hole position from the reference stored in **NF**. If the pocket consists of multiple sections, the unit machines a hole for each section.







Unit name: **DRILL_STI_845_C** / cycles: **G845**; **G71**

Overvw. form:

- **AP**: **Predrilling position**
  - **1: Determine predrilling pos**
  - **2: predrill. pos. fig. center**

Further information: "G845 – calculating hole positions", Page 473

Further information: "Simple drilling G71", Page 423

Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
- **Z1**: **Millg. top edge** (default: **Starting point Z**)
- **P2**: **Depth of contour**

Cycle form:

- **JT**: **Machining direction**
  - **0: From the inside out**
  - **1: From the outside in**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
- **WB**: **Milling diameter**
- **NF**: **Position mark** – reference at which the cycle stores the predrill hole positions (range: 1 to 127)
- **E**: **Period of dwell** at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **V**: **Feed reduction**
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **AB**: **Drilling lengths** (default: 0)
- **RB**: **Return plane** (default: back to start position)

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **Drilling**
- Affected parameters: **F**, **S**

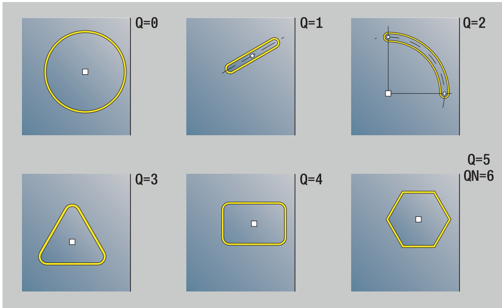
Unit G840
Predrill, contour mill, fig. on lateral in C

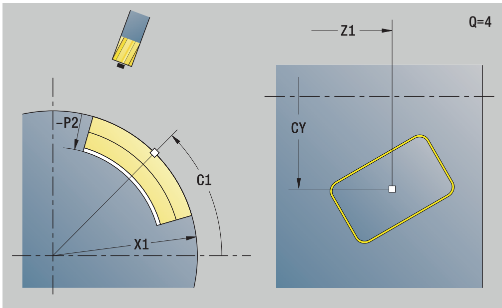
The unit determines the hole position and machines the hole. The subsequent milling cycle obtains the hole position from the reference stored in **NF**.

Unit name: **DRILL_MAN_KON_C** / cycles: **G840 A**; **G71**

Further information: "G840 – calculating hole positions", Page 464

Further information: "Simple drilling G71", Page 423





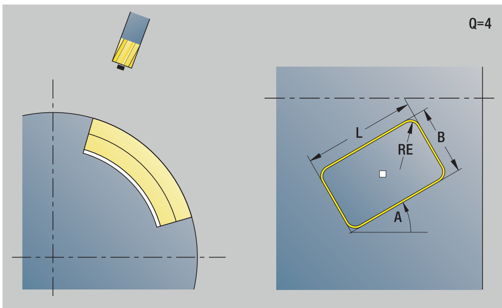
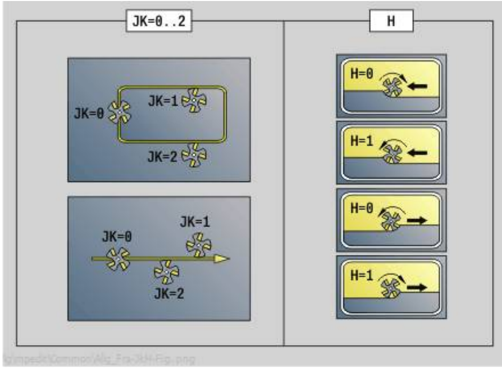
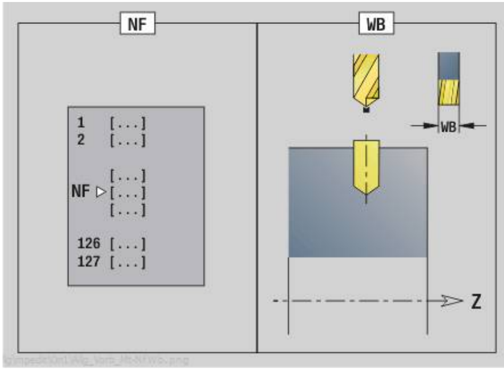


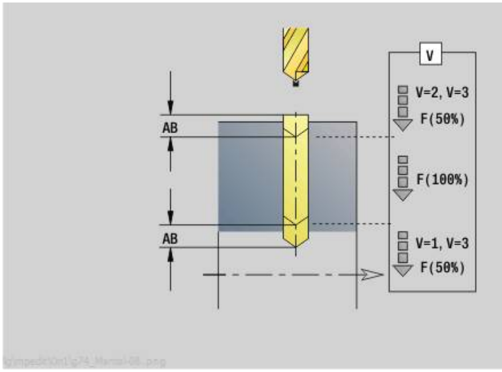
Figure form:

- **Q**: **Type of figure**
  - **0: Complete circle**
  - **1: Linear slot**
  - **2: Circular slot**
  - **3: Triangle**
  - **4: Rectangle / square**
  - **5: Polygon**
- **QN**: **No. of polygon corners** (only with **Q** = **5: Polygon**)
- **Z1**: **Figure center**
- **C1**: **Angle of figure center** (default: **Spindle angle C**)
- **CY**: **Fig. center, unr.lat. surf.**
- **X1**: **Upper edge of milling**
- **P2**: **Depth of figure**
- **L**: **+edge lngth/-width a. flats**
  - **L** > 0: **Edge length**
  - **L** < 0: **Width across** (inside diameter) for polygon
- **B**: **Width of rectangle**
- **RE**: **Rounding radius** (default: 0)
- **A**: **Angle to Z axis** (default: 0°)
- **Q2**: **Rot. direction of slot** (only with **Q** = **2: Circular slot**)
  - **cw**: In clockwise direction
  - **ccw**: In counterclockwise direction
- **W**: **Angle of slot end point** (only with **Q** = **2: Circular slot**)

> Program only the parameters relevant to the selected figure type.







Cycle form:

- **JK**: **Cutter position**
  - **0: On the contour**
  - **1: Within the contour**
  - **2: Outside the contour**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **R**: **Apprch angle** (default: 0)
- **WB**: **Milling diameter**
- **NF**: **Position mark** – reference at which the cycle stores the predrill hole positions (range: 1 to 127)
- **E**: **Period of dwell** at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **V**: **Feed reduction**
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **AB**: **Drilling lengths** (default: 0)
- **RB**: **Return plane** (default: back to start position)

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **Drilling**
- Affected parameters: **F**, **S**

Unit G845 Predrill, pocket mill, fig. on lateral in C

The unit determines the hole position and machines the hole. The subsequent milling cycle obtains the hole position from the reference stored in **NF**.

Unit name: **DRILL_MAN_TAS_C** / cycles: **G845**; **G71**

Overvw. form:

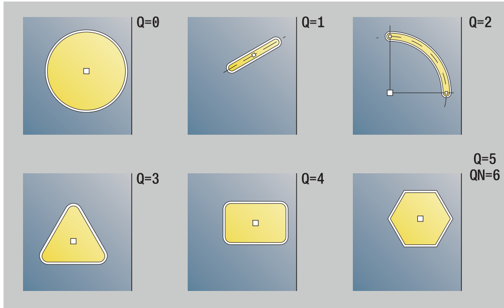
- **AP**: **Predrilling position**
  - **1: Determine predrilling pos**
  - **2: predrill. pos. fig. center**

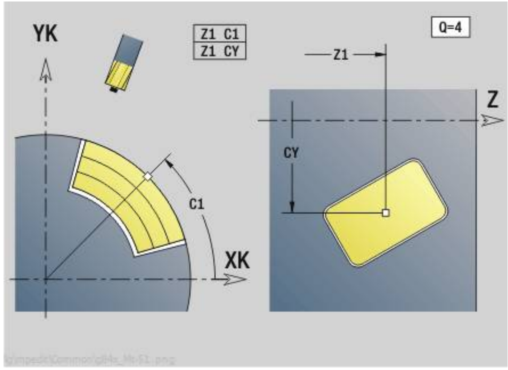
Further information: "G845 – calculating hole positions", Page 473

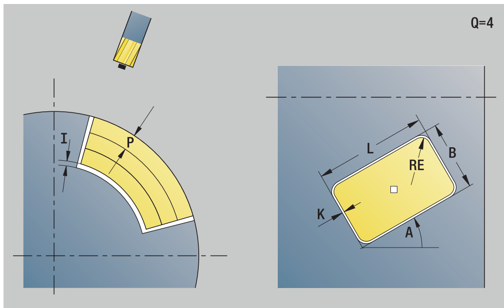
Further information: "Simple drilling G71", Page 423

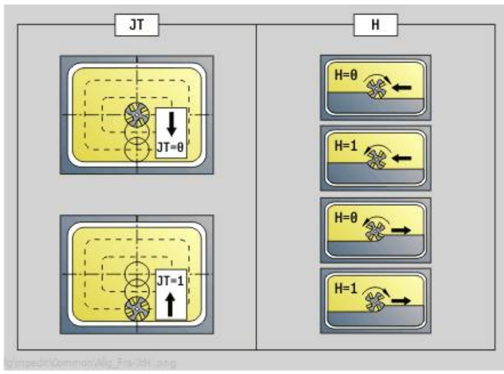
Figure form:

- **Q**: **Type of figure**
  - **0: Complete circle**
  - **1: Linear slot**
  - **2: Circular slot**
  - **3: Triangle**
  - **4: Rectangle / square**
  - **5: Polygon**
- **QN**: **No. of polygon corners** (only with **Q** = **5: Polygon**)
- **Z1**: **Figure center**
- **C1**: **Angle of figure center** (default: **Spindle angle C**)
- **CY**: **Fig. center, unr.lat. surf.**
- **X1**: **Upper edge of milling**
- **P2**: **Depth of figure**
- **L**: **+edge lngth/-width a. flats**
  - **L** > 0: **Edge length**
  - **L** < 0: **Width across** (inside diameter) for polygon
- **B**: **Width of rectangle**
- **RE**: **Rounding radius** (default: 0)
- **A**: **Angle to Z axis** (default: 0°)
- **Q2**: **Rot. direction of slot** (only with **Q** = **2: Circular slot**)
  - **cw**: In clockwise direction
  - **ccw**: In counterclockwise direction
- **W**: **Angle of slot end point** (only with **Q** = **2: Circular slot**)

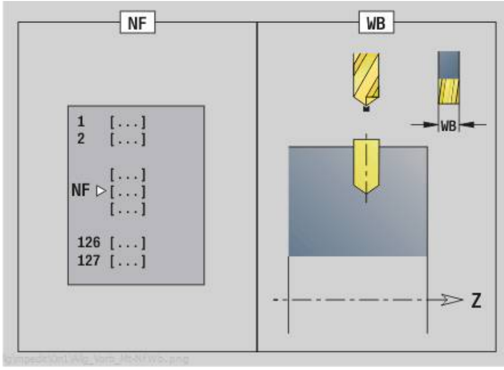


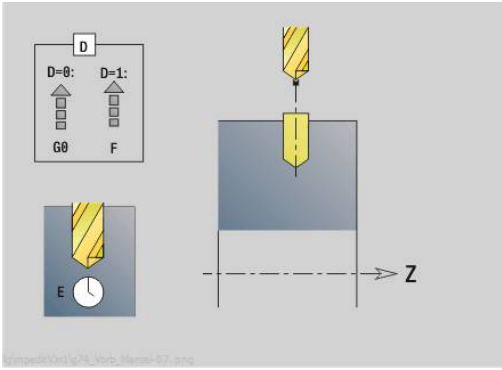






> Program only the parameters relevant to the selected figure type.





Cycle form:

- **JT**: **Machining direction**
  - **0: From the inside out**
  - **1: From the outside in**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
- **WB**: **Milling diameter**
- **NF**: **Position mark** – reference at which the cycle stores the predrill hole positions (range: 1 to 127)
- **E**: **Period of dwell** at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **V**: **Feed reduction**
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **AB**: **Drilling lengths** (default: 0)
- **RB**: **Return plane** (default: back to start position)

Further forms:

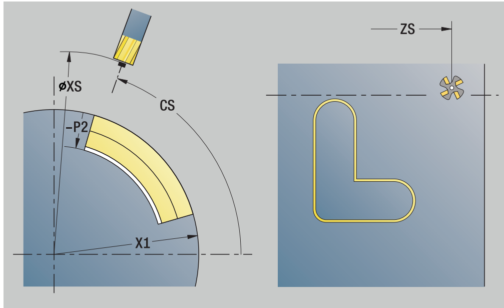
Further information: "smart.Turn unit", Page 98

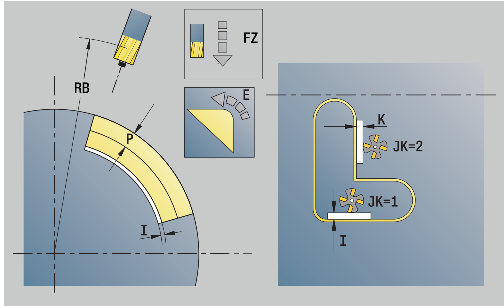
Access to the technology database:

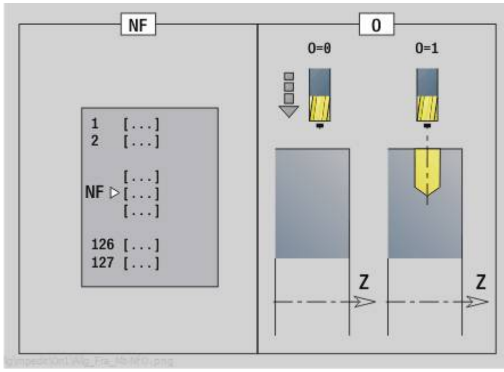
- Machining operation: **Drilling**
- Affected parameters: **F**, **S**

Unit G840 Predrill, contour mill, ICP on lateral in C

The unit determines the hole position and machines the hole. The subsequent milling cycle obtains the hole position from the reference stored in **NF**. If the milling contour consists of multiple sections, the unit machines a hole for each section.







Unit name: **DRILL_MAN_840_C** / cycles: **G840 A1**; **G71**

Further information: "G840 – calculating hole positions", Page 464

Further information: "Simple drilling G71", Page 423

Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
- **X1**: **Upper edge of milling** (diameter value; default: **Starting point X**)
- **P2**: **Depth of contour**

Cycle form:

- **JK**: **Cutter position**
  - **0: On the contour**
  - **1: Within/left of contour**
  - **2: Outside/right of contour**
  - **3: Depending on H and MD**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **R**: **Apprch angle** (default: 0)
- **WB**: **Milling diameter**
- **NF**: **Position mark** – reference at which the cycle stores the predrill hole positions (range: 1 to 127)
- **E**: **Period of dwell** at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **V**: **Feed reduction**
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **AB**: **Drilling lengths** (default: 0)
- **RB**: **Return plane** (default: back to start position)

Further forms:

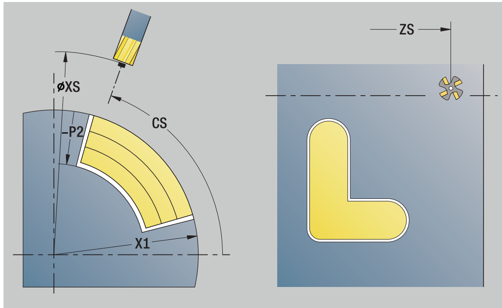
Further information: "smart.Turn unit", Page 98

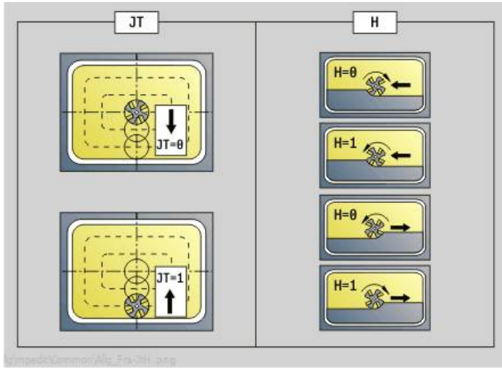
Access to the technology database:

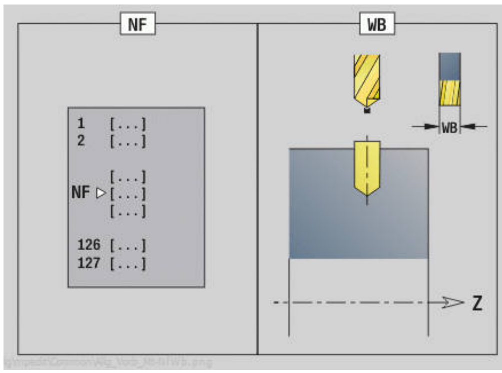
- Machining operation: **Drilling**
- Affected parameters: **F**, **S**

Unit G845
Predrill, pocket mill, ICP on lateral in C

The unit determines the hole position and machines the hole. The subsequent milling cycle obtains the hole position from the reference stored in **NF**. If the pocket consists of multiple sections, the unit machines a hole for each section.







Unit name: **DRILL_MAN_845_C** / cycles: **G845**; **G71**

Overvw. form:

- **AP**: **Predrilling position**
  - **1: Determine predrilling pos**
  - **2: predrill. pos. fig. center**

Further information: "G845 – calculating hole positions", Page 473

Further information: "Simple drilling G71", Page 423

Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
- **X1**: **Upper edge of milling** (diameter value; default: **Starting point X**)
- **P2**: **Depth of contour**

Cycle form:

- **JT**: **Machining direction**
  - **0: From the inside out**
  - **1: From the outside in**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
- **WB**: **Milling diameter**
- **NF**: **Position mark** – reference at which the cycle stores the predrill hole positions (range: 1 to 127)
- **E**: **Period of dwell** at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **V**: **Feed reduction**
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **AB**: **Drilling lengths** (default: 0)
- **RB**: **Return plane** (default: back to start position)

Further forms:

Further information: "smart.Turn unit", Page 98

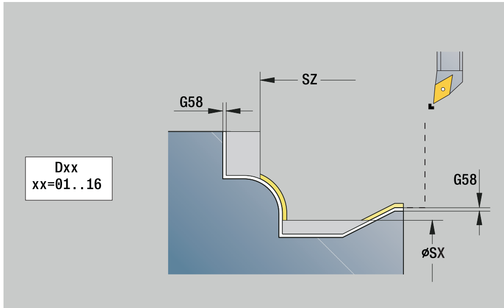
Access to the technology database:

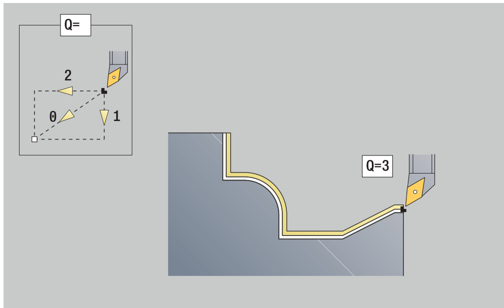
- Machining operation: **Drilling**
- Affected parameters: **F**, **S**

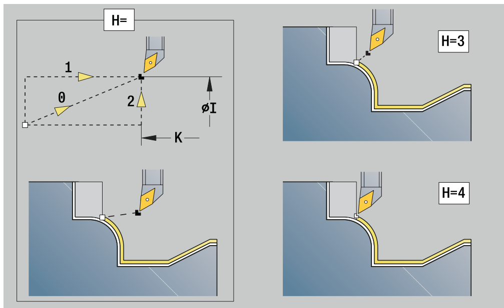
4.7 Units – Finishing

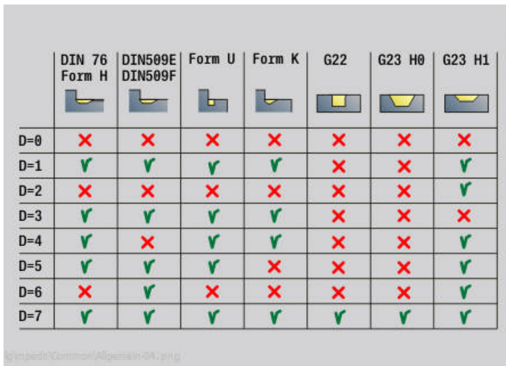
Unit G890 Contouring in ICP

The unit finishes the contour described by **ICP** from **NS** to **NE** in one pass.









	DIN 76 Form H	DIN509E DIN509F	Form U	Form K	G22	G23 H0	G23 H1
D=0	✗	✗	✗	✗	✗	✗	✗
D=1	✓	✓	✓	✓	✗	✗	✓
D=2	✗	✗	✗	✗	✗	✗	✓
D=3	✓	✓	✓	✓	✗	✗	✗
D=4	✓	✗	✓	✓	✗	✗	✓
D=5	✓	✓	✓	✗	✗	✗	✓
D=6	✗	✓	✗	✗	✗	✗	✓
D=7	✓	✓	✓	✓	✓	✓	✓

> In Machine Parameter 602322, you define whether the control is to check the usable length of the cutting edge during finishing. For recessing and button tools, the length of the cutting edge is not checked.

Unit name: **G890_ICP** / cycle: **G890**

Further information: "Contour finishing G890", Page 374

Contour form:

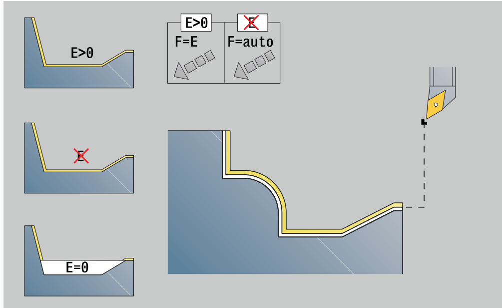
- **B**: **TRC/MRC switch on** – type of tool radius compensation
  - **0: Automatic**
  - **1: Tool left (G41)**
  - **2: Tool right(G42)**
  - **3: Automatic w/o tool compens.**
  - **4: W/o tool compens. left (G41)**
  - **5: W/o tool compens. left (G42)**
- **HR**: **Main machining direction**
  - **0: auto**
  - **1: +Z**
  - **2: +X**
  - **3: -Z**
  - **4: -X**
- **SX**, **SZ**: **Cutting limit in X** and **Z** (default: no cutting limit; diameter value = **SX**)

Further parameters of the **Contour** form:

Further information: "Contour form", Page 100

Cycle form:

- **Q**: **Type of approach** (default: 0)
  - **0: Automatic** – the control checks:
    - Diagonal approach
    - First X, then Z direction
    - Equidistant around the workpiece blank, taking the safety clearance into account
    - Omission of the first contour elements if the starting position is inaccessible
  - **1: First X, then Z**
  - **2: First Z, then X**
  - **3: No approach** – tool is located near the starting point of the contour area
  - **4: Finish rem. mat.**



- **H**: **Type of retraction** – tool backs off at an angle below 45° opposite the machining direction and moves to the position **I**, **K** (default: 3)
  - **0: Same time, to I+K**
  - **1: First X then Z, to I+K**
  - **2: First Z then X, to I+K**
  - **3: Retract by safety clear.**
  - **4: No retraction motion** (tool remains on the end coordinate)
  - **5: Diagonal to start pos.**
  - **6: X then Z to start pos.**
  - **7: Z then X to start pos.**
  - **8: With G1 to I and K**
- **I**, **K**: **Cycle end position X** and **Z** – position approached at end of cycle (**I** = diameter value)
- **D**: **Omit elements** (see figure)
- **E**: **Plunging behavior**
  - **E** = 0: Descending contours are not machined
  - **E** > 0: Plunging feed rate for descending contour elements. Descending contour elements are machined
  - No input: The plunging feed rate is reduced during machining of descending contour elements by up to 50 %. Descending contour elements are machined
- **O**: **Feed reduc. off** for circular elements (default: 0)
  - **0: No** (feed rate reduction is active)
  - **1: Yes** (feed rate reduction is not active)
- **DXX**: **Add. correction number** (range: 1 to 16)

  Further information: User's Manual
- **G58**: **Contour-parallel oversize**
- **DI**, **DK**: **Oversize X** and **Z** paraxial

Further forms:

Further information: "smart.Turn unit", Page 98

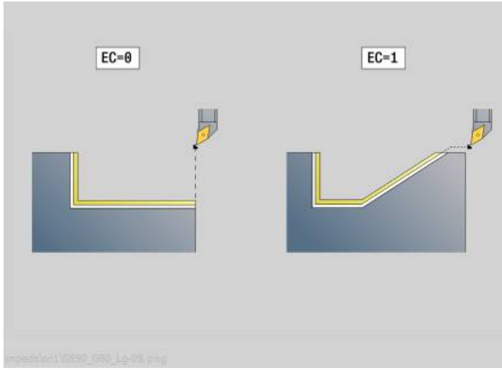
> If feed rate reduction is active, at least four spindle revolutions are used to machine every **small** contour element.
>
> With the address **DXX**, you can activate an additive compensation for the entire cycle run. The additive compensation is switched off again at the end of the cycle. To edit additive compensation values, change to **Program run** submode.

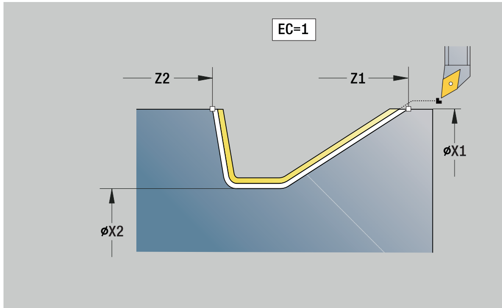
Access to the technology database:

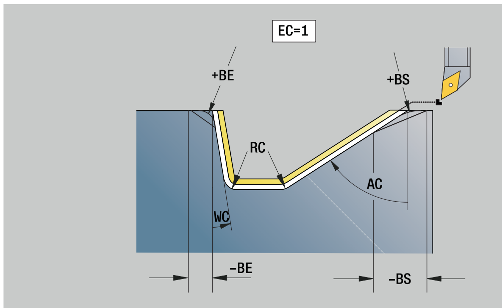
- Machining operation: **Finishing**
- Affected parameters: **F**, **S**

Unit G890 Contouring, direct longitdnl.

The unit finishes the contour defined by the parameters in one pass. In **EC** you define whether you want to machine a normal or a plunging contour.







> In Machine Parameter 602322, you define whether the control is to check the usable length of the cutting edge during finishing. For recessing and button tools, the length of the cutting edge is not checked.

Unit name: **G890_G80_L** / cycle: **G890**

Further information: "Contour finishing G890", Page 374

Contour form:

- **EC**: **Type of contour**
  - **0: Normal contour**
  - **1: Plunging contour**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **RC**: **Rounding** – radius of contour corner
- **AC**: **Start angle** – angle of the first contour element (range: 0°< **AC** < 90°)
- **WC**: **Final angle** – angle of the last contour element (range: 0° < **WC** < 90°)
- **BS**: **-Chamfer/+radius at start**
  - **BS** > 0: Radius of rounding arc
  - **BS** < 0: Width of chamfer
- **BE**: **-Chamfer/+radius at end**
  - **BE** > 0: Radius of rounding arc
  - **BE** < 0: Width of chamfer

Cycle form:

- **E**: **Plunging behavior**
  - **E** = 0: Descending contours are not machined
  - **E** > 0: Plunging feed rate for descending contour elements. Descending contour elements are machined
  - No input: The plunging feed rate is reduced during machining of descending contour elements by up to 50 %. Descending contour elements are machined
- **B**: **TRC/MRC switch on** – type of tool radius compensation
  - **0: Automatic**
  - **1: Tool left (G41)**
  - **2: Tool right(G42)**
  - **3: Automatic w/o tool compens.**
  - **4: W/o tool compens. left (G41)**
  - **5: W/o tool compens. left (G42)**
- **DXX**: **Add. correction number** (range: 1 to 16)

  Further information: User's Manual
- **G58**: **Contour-parallel oversize**

Further forms:

Further information: "smart.Turn unit", Page 98

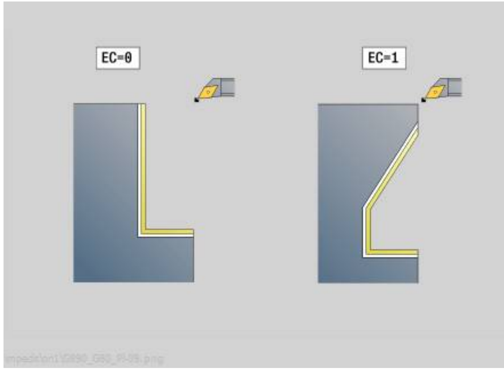
With the address **DXX**, you can activate an additive compensation for the entire cycle run. The additive compensation is switched off again at the end of the cycle. To edit additive compensation values, change to **Program run** submode.

Access to the technology database:

- Machining operation: **Finishing**
- Affected parameters: **F**, **S**, **E**

Unit G890 Contouring, direct transverse

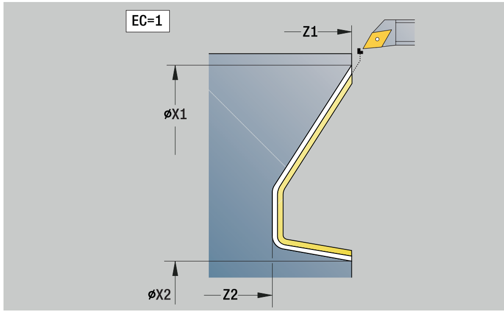
The unit finishes the contour defined by the parameters in one pass. In **EC** you define whether you want to machine a normal or a plunging contour.



> In Machine Parameter 602322, you define whether the control is to check the usable length of the cutting edge during finishing. For recessing and button tools, the length of the cutting edge is not checked.

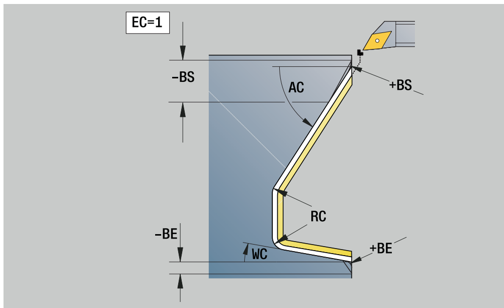
Unit name: **G890_G80_P** / cycle: **G890**

Further information: "Contour finishing G890", Page 374



Contour form:

- **EC**: **Type of contour**
  - **0: Normal contour**
  - **1: Plunging contour**
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**
- **RC**: **Rounding** – radius of contour corner
- **AC**: **Start angle** – angle of the first contour element (range: 0°< **AC** < 90°)
- **WC**: **Final angle** – angle of the last contour element (range: 0° < **WC** < 90°)
- **BS**: **-Chamfer/+radius at start**
  - **BS** > 0: Radius of rounding arc
  - **BS** < 0: Width of chamfer
- **BE**: **-Chamfer/+radius at end**
  - **BE** > 0: Radius of rounding arc
  - **BE** < 0: Width of chamfer



Cycle form:

- **E**: **Plunging behavior**
  - **E** = 0: Descending contours are not machined
  - **E** > 0: Plunging feed rate for descending contour elements. Descending contour elements are machined
  - No input: The plunging feed rate is reduced during machining of descending contour elements by up to 50 %. Descending contour elements are machined
- **B**: **TRC/MRC switch on** – type of tool radius compensation
  - **0: Automatic**
  - **1: Tool left (G41)**
  - **2: Tool right(G42)**
  - **3: Automatic w/o tool compens.**
  - **4: W/o tool compens. left (G41)**
  - **5: W/o tool compens. left (G42)**
- **DXX**: **Add. correction number** (range: 1 to 16)

  Further information: User's Manual
- **G58**: **Contour-parallel oversize**

Further forms:

Further information: "smart.Turn unit", Page 98

> With the address **DXX**, you can activate an additive compensation for the entire cycle run. The additive compensation is switched off again at the end of the cycle. To edit additive compensation values, change to **Program run** submode.

Access to the technology database:

- Machining operation: **Finishing**
- Affected parameters: **F**, **S**, **E**

Unit G890 Relief, type E,F,DIN76 – Recess

The unit machines the undercut defined by **KG**, and then the plane surface. The cylinder chamfer is executed when you enter at least one of the parameters **Cylinder 1st cut length** or **Cut-in radius**.

Unit name: **G85x_DIN_E_F_G** / cycle: **G85**

Further information: "Undercut cycle G85", Page 414

Overvw. form:

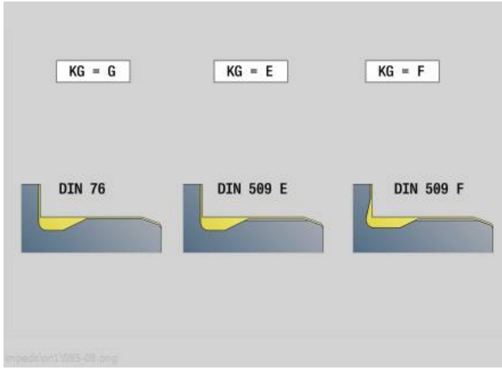
- **APP**: **Type of approach**
- **KG**: **Type of relief turn**
  - **E: DIN 509 E**; cycle **G851**
    Further information: "Undercut DIN 509 E with cylinder machining G851", Page 415
  - **F: DIN 509 F**; cycle **G852**
    Further information: "Undercut DIN 509 F with cylinder machining G852", Page 416
  - **G: DIN 76** (thread undercut); cycle **G853**
    Further information: "Undercut DIN 76 with cylinder machining G853", Page 417
- **X1**, **Z1**: **Start point contour**
- **X2**, **Z2**: **End point contour**

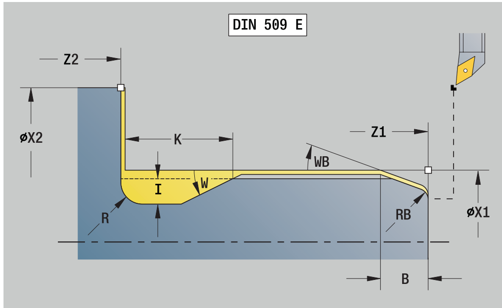
Undercut **Type E**:

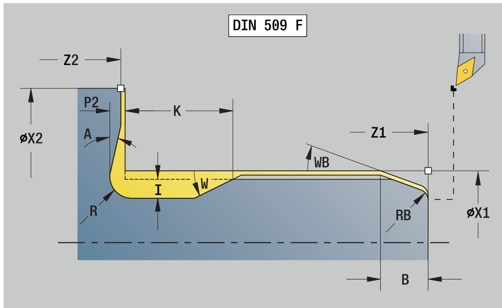
- **I**: **Undercut depth** (default: value from standard table)
- **K**: **Undercut length** (default: standard table)
- **W**: **Undercut angle** (default: value from standard table)
- **R**: **Undercut radius** (default: standard table)
- **H**: **Type of departure**
  - **0: To starting point**
  - **1: Plane surface end**

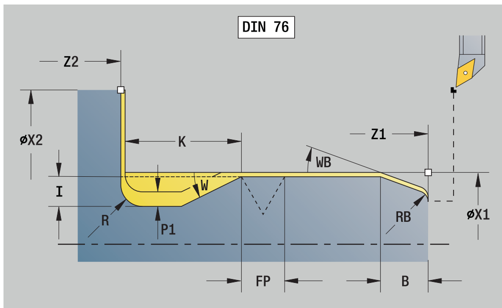
Undercut **Type F**:

- **I**: **Undercut depth** (default: value from standard table)
- **K**: **Undercut length** (default: standard table)
- **W**: **Undercut angle** (default: value from standard table)
- **R**: **Undercut radius** (default: standard table)
- **P2**: **Surf. depth** (default: value from standard table)
- **A**: **Face angle** (default: standard table)
- **H**: **Type of departure**
  - **0: To starting point**
  - **1: Plane surface end**









Undercut **Type G**:

- **FP**: **Thread pitch** (default: value from standard table)
- **I**: **Undercut depth** (default: value from standard table)
- **K**: **Undercut length** (default: standard table)
- **W**: **Undercut angle** (default: value from standard table)
- **R**: **Undercut radius** (default: standard table)
- **P1**: **Undercut oversize**
  - No input: Machining in one cut
  - **P1** > 0: Division into pre-turning and finish-turning. **P1** is longitudinal oversize; the transverse oversize is preset to 0.1 mm
- **H**: **Type of departure**
  - **0: To starting point**
  - **1: Plane surface end**

Additional parameters for cylinder first cut:

- **B**: **Cylinder 1st cut length** (default: no start chamfer)
- **WB**: **Cut-in angle** (default: 45°)
- **RB**: **Cut-in radius** (no input: no element; positive value: cut-in radius; negative value: chamfer)
- **E**: **Reduced feed** for the plunge cut and the thread chamfer (default: **Feed per revolution F**)
- **U**: **Grind. overs.** for the area of the cylinder (default: 0)

Further forms:

Further information: "smart.Turn unit", Page 98

> - Undercuts can only be executed in orthogonal, paraxial contour corners along the longitudinal axis
> - Parameters that are not programmed are automatically calculated by the control from the standard table

Access to the technology database:

- Machining operation: **Finishing**
- Affected parameters: **F**, **S**, **E**

Unit G809 Measuring cut

The unit performs a cylindrical measuring cut with the length defined in the cycle, moves to the breakpoint for measuring and stops the program. After the program was stopped, you can manually measure the workpiece.

Unit name: **MEASURE_G809** / cycle: **G809**

Further information: "Measuring path G809", Page 389

Overview form:

- **EC**: **Machining location**
  - **1: Outside**
  - **-1: Inside**
- **XA**, **ZA**: **Start point** of contour
- **R**: **Length of measuring cut**
- **P**: **Measuring cut oversize**

Contour form:

- **O**: **Approach angle**
  If an approach angle is entered, the cycle positions the tool over the starting point taking into account the safety clearance, and from there plunges at the specified angle to the diameter to be measured.
- **ZR**: **Start point blank** – collision-free approach for inside machining

Cycle form:

- **QC**: **Machining direction**
  - **0: -Z**
  - **1: +Z**
- **V**: **Measuring cut counter** – number of workpieces after which a measurement is performed
- **D**: **Additive correction** (numbers: 1 to 16)
- **WE**: **Type of approach**
  - **0: Simultaneously**
  - **1: First X, then Z**
  - **2: First Z, then X**
- **I**, **K**: **Breakpoint Xi for measuring** and **Zi**
- **AX**: **Departure position X**

Further forms:

Further information: "smart.Turn unit", Page 98

Unit G891 Simultaneous finishing (option 54)

The unit finishes the contour described by **ICP** from **NS** to **NE** simultaneously in three axes in one pass.

NOTICE

Danger of collision!

Collision monitoring is performed only in the two-dimensional machining plane X-Z. The cycle does not check whether an area in the Y coordinate of the cutting edge, tool holder, or tilting body will lead to a collision.

- Verify the NC program in **Single Block**
- Limit the machining area

In machine parameter **checkCuttingLength** (no. 602322), you can define whether the control is to check the usable length of the cutting edge during finishing. For button tools, the default setting does not check the length of the cutting edge.

Unit name: **G891_ICP** / cycle: **G891**

Further information: "Simultaneous finishing G891 (option 54)", Page 384

Contour form:

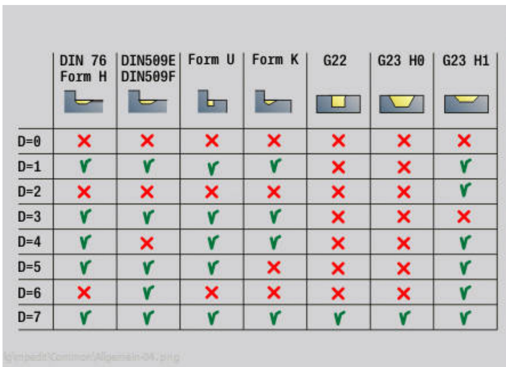
- **D**: **Omit elements** (see figure)

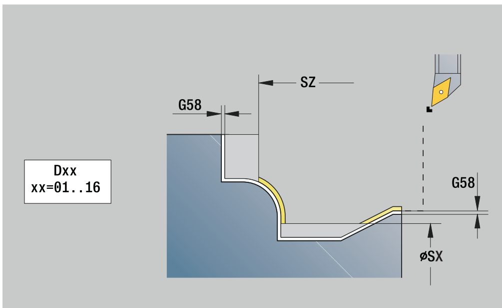
  Codes for omitting recesses and undercuts

G call	Function	D code
G22	Recess for sealing ring	512
G22	Recess for circlip	1,024
G23 H0	General recess	256
G23 H1	Relief turn	2,048
G25 H4	Undercut type U	32,768
G25 H5	Undercut type E	65,536
G25 H6	Undercut type F	131,072
G25 H7	Undercut type G	262,144
G25 H8	Undercut type H	524,288
G25 H9	Undercut type K	1,048,576

  In order to omit multiple elements, you add the D codes from the table or use the D values from the graphic.

- **B**: **TRC/MRC switch on** – type of tool radius compensation
  - **0: Automatic**
  - **1: Tool left (G41)**
  - **2: Tool right(G42)**



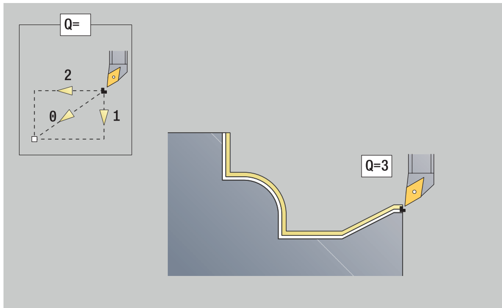


- **O**: **Feed reduc. off** for circular elements (default: 0)
  - **0: No**
  - **1: Yes**
- **SX**, **SZ**: **Cutting limit in X** and **Z** (default: no cutting limit; diameter value = **SX**)
- **A**: **Start angle** (reference: Z axis; default: parallel to Z axis)
- **W**: **Depart.angle** (reference: Z axis; default: parallel to Z axis)

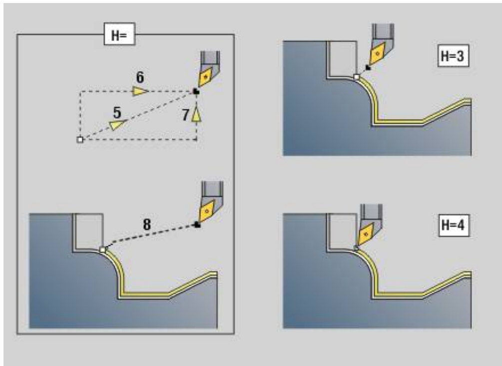
Further parameters of the **Contour** form:

Further information: "Contour form", Page 100

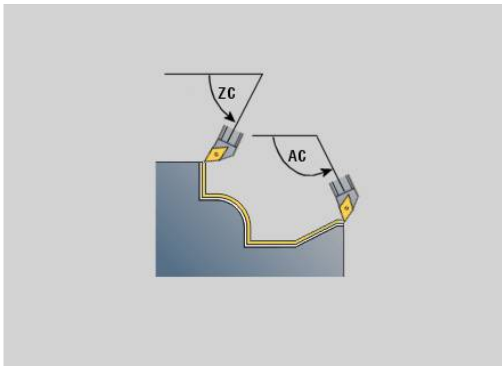
Cycle form:



- **Q**: **Type of approach** (default: 0)
  - **0: automatic (with B)** – the control checks:
    - Diagonal approach
    - First X, then Z direction
    - Equidistant around the workpiece blank, taking the safety clearance into account
    - Omission of the first contour elements if the starting position is inaccessible
  - **1: First X, then Z**
  - **2: First Z, then X**
  - **3: No approach** – tool is located near the starting point of the contour area



- **H** : **Kind of liber.**
  - **3: Retract by safety clear.**
  - **4: No retraction motion** (tool remains on the end coordinate)
  - **5: Diagonal to start pos.**
  - **6: X then Z to start pos.**
  - **7: Z then X to start pos.**
  - **8: with B-axis motion to start pos.**

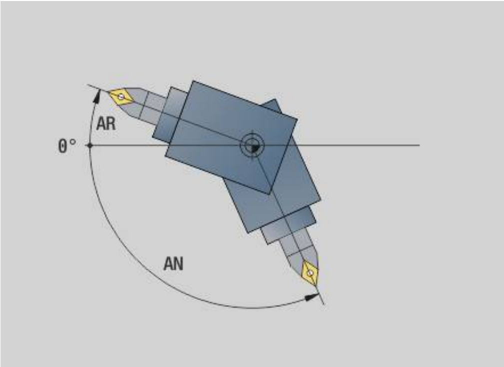


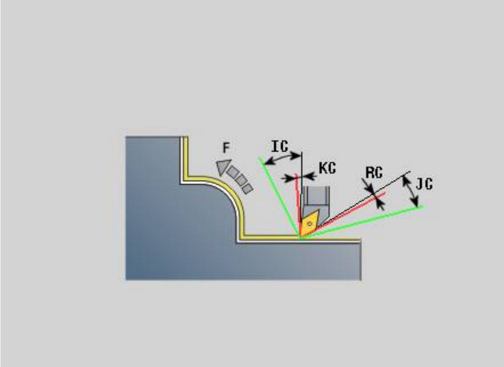
  - **AC**: **B angle at starting point** – inclined tilt angle at the beginning of the contour (range: 0° < **AC** < 360°)
  - **ZC**: **B angle at end point** – inclined tilt angle at the end of the contour (range: 0° < **ZC** < 360°)
  - **AR**: **Minimum angle of incidence** – minimum permissible angle of the tilting axis (range: –359.999° < **AR** < 359.999°)
  - **AN**: **Maximum angle of incidence** – maximum permissible angle of the tilting axis (range: –359.999° < **AN** < 359.999°)
  - **IC**: **Primary clearance angle - soft**: Desired clearance area in front of the cutting edge
  - **JC** : **Secondary clearance angle - soft**: Desired clearance area behind the cutting edge
  - **KC**: **Primary clearance angle - hard**: Safe clearance area in front of the cutting edge

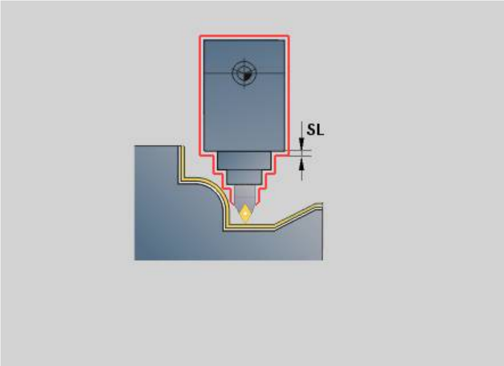
- **RC**: **Primary clearance angle - hard**: Safe clearance area behind the cutting edge

> If you have defined hard clearance angles, they must be complied with during machining (do not go below the defined values). If the hard clearance angles defined for machining a contour cannot be complied with, the control will output an error message.
>
> In addition to the hard clearance angles, you can use the soft clearance angles to define a desired angle range for machining. The control takes into account the soft clearance angles during path calculation and preferably performs machining within the defined angle range. The soft clearance angles do not have to be complied with during machining.

- **SL**: **Oversize of tool holder** – oversize for calculating collisions between the workpiece and the tool holder
- **E**: **Fmax in compensating motion** – speed limit during the compensation movement of the linear axes



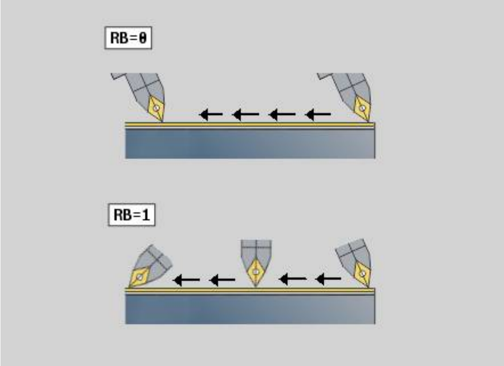




Cycle 2 form:

- **U** : **Use of soft clearance angles** – defines how the soft clearance angles **IC** and **JC** are used
  - **0: very hard**
  - **1: hard**
  - **2: medium**
  - **3: soft**
  - **4: very soft**
- **RB** : **Roll over** – uniform wear of cutting edge by controlling the inclination angle
  - **0: No**
  - **1: Yes**
- **DXX**: **Add. correction number** (range: 1 to 16)

  Further information: User's Manual



- **G58**: **Contour-parallel oversize**
- **DI**, **DK**: **Oversize X** and **Z** paraxial

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **Finishing**
- Affected parameters: **F**, **S**

4.8 Units – Thread

Overview of thread units

Overview of thread units:

- **G32 Thread, direct** cuts a simple internal or external thread in longitudinal direction.
- **G31 Thread, ICP** cuts a single or multi-start internal or external thread in longitudinal or transverse direction. The contour on which the thread is cut is defined with **ICP**
- **G352 API thread** cuts a single or multi-start API thread. The depth of the thread decreases at the thread run-out.
- **G32 Tapered thread** cuts a single or multi-start tapered internal or external thread

Handwheel superimposition (option 11)

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

> Refer to your machine manual.
> This function must be set up by your machine manufacturer.

> Remember that position changes resulting from handwheel superimposition are no longer effective after the cycle end or the **Last cut** function!

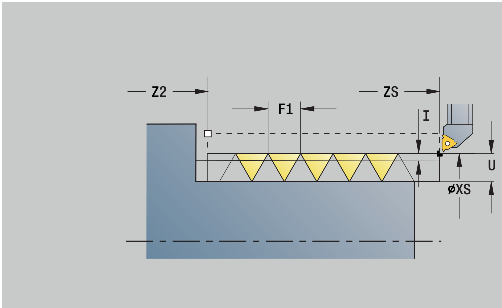
Parameter V: Type of infeed

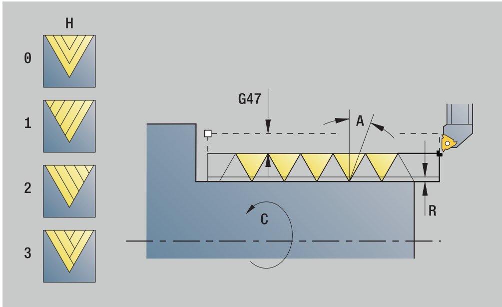
With the **V** parameter you define the type of infeed for thread cutting cycles.

The following infeed types are available:

- **0: Const. mach. X-section** – The control reduces the cutting depth with each infeed so that the chip cross section, and therefore the removal rate, remain constant
- **1: Const. infeed** – with each infeed the control uses the same cutting depth without exceeding the **Max. approach I**
- **2: EPL with distrib. of cuts** – the control calculates the cutting depth for constant infeed from the **Thread pitch F1** and **Shaft speed constant S**. If the multiple of the cutting depth does not correspond to the **Thread depth**, the control uses the **Remaining cut depth (V=4)** for the first infeed. With the "distribution of remaining cuts," the control divides the last cutting depth into four partial cuts. The first cut is half the calculated cutting depth, the second is a quarter and the third and fourth each are an eighth
- **3: EPL w/o distrib. of cuts** – the control calculates the cutting depth for constant infeed from the **Thread pitch F1** and **Shaft speed constant S**. If the multiple of the cutting depth does not correspond to the **Thread depth**, the control uses the **Remaining cut depth (V=4)** for the first infeed. All subsequent infeeds are constant and correspond to the calculated cutting depth
- **4: MANUALplus 4110** – the control executes the first infeed with the **Max. approach I**. To determine the subsequent cutting depths, the control uses the formula **gt** = 2 * **I** * SQRT "current no. of cuts", where **gt** is the absolute depth. Since the cutting depth is reduced with each infeed because the current number of cuts increases by the value 1, the control uses, each time the cutting depth falls below the value given in **Remaining cut depth (V=4) R**, the value defined here as the new constant cutting depth! If the multiple of the cutting depth does not correspond to the **Thread depth**, the control executes the last cut at the final depth
- **5: Constant infeed (4290)** – with each infeed, the control uses the same cutting depth where the cutting depth corresponds to the **Max. approach I**. If the multiple of the cutting depth does not correspond to the **Thread depth**, the control uses the **Remaining cut depth (V=4)** for the first infeed
- **6: Const. w/ distrib. (4290)** – with each infeed, the control uses the same cutting depth where the cutting depth corresponds to the **Max. approach I**. If the multiple of the cutting depth does not correspond to the **Thread depth**, the control uses the **Remaining cut depth (V=4)** for the first infeed. With the "distribution of remaining cuts," the control divides the last cutting depth into four partial cuts. The first cut is half the calculated cutting depth, the second is a quarter and the third and fourth each are an eighth

Unit G32 Thread, direct





The unit cuts a simple internal or external thread in longitudinal direction.

Unit name: **G32_MAN** / cycle: **G32**

Further information: "Single thread cycle G32", Page 403

NOTICE

Danger of collision!

If you modify the angle of infeed or the thread depth, the control shifts the position of the thread in an axial direction. In this case, the tool no longer hits any existing thread grooves and the thread flanks are destroyed. Danger of collision during reworking!

- Be sure to compensate only the tool, not the thread parameters

Thread form:

- **O**: **Thread location:**
  - **0**: Internal thread (infeed in +X)
  - **1**: External thread (infeed in -X)
- **APP**: **Type of approach**
- **XS**: **Starting diameter**
- **ZS**: **Starting position Z**
- **Z2**: **End point thread**
- **F1**: **Thread pitch**
- **U**: **Thread depth**
- **I**: **Max. approach**
- **IC**: **Number of cuts** (only if **I** is not programmed and **Type of infeed V** = 0 or **V** = 1)
- **KE**: **Run-out position:**
  - **0: At end**
  - **1: At beginning**
- **K**: **Thread runout length**

Cycle form:

- **H**: **Type of offset** – offset between the individual infeeds in the cutting direction
  - **0: Without offset**
  - **1: From left**
  - **2: From right**
  - **3: Alternating left/right**
- **V**: **Type of infeed**
  - **0: Const. mach. X-section**
  - **1: Const. infeed**
  - **2: EPL with distrib. of cuts**
  - **3: EPL w/o distrib. of cuts**
  - **4: MANUALplus 4110**
  - **5: Constant infeed (4290)**
  - **6: Const. w/ distrib. (4290)**
- **A**: **Approach ang.** (range: –60° < **A** < 60°; default: 30°)

- **R**: **Remaining cut depth (V=4)**
- **WE**: **Lift off method with K=0** (default: 0)
  - **0: G0 at end**
  - **1: Lift-off in thread**
- **C**: **Start angle**
- **D**: **No.gears**
- **Q**: **Number no-load.**
- **E**: **Variable gr.** (default: 0)
  Increases/decreases the pitch per revolution by **E**.

Further forms:

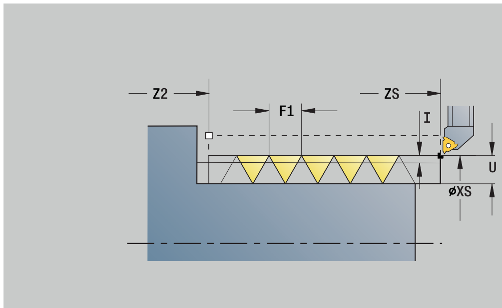
Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **Thread cutting**
- Affected parameters: **F**, **S**

Unit G31 Thread, ICP

The unit cuts a single or multi-start internal or external thread in longitudinal or transverse direction. The contour on which the thread is cut is defined with **ICP**.



Unit name: **G31_ICP** / cycle: **G31**

Further information: "Universal tapping cycle G31", Page 398

Thread form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
- **O1**: **Edit form element:**
  - **0: No machining**
  - **1: At beginning**
  - **2: At end**
  - **3: At beginning and end**
  - **4: Only chamfer/rounding**
- **O**: **Thread location:**
  - **0**: Internal thread (infeed in +X)
  - **1**: External thread (infeed in -X)
- **J1**: **Thread orientation**
  - **From 1st contour element**
  - **0: Longitudinal**
  - **1: Plane**
- **F1**: **Thread pitch**
- **U**: **Thread depth**
- **A**: **Thread angle**
- **D**: **No.gears**
- **K**: **Thread runout length**

Cycle form:

- **H**: **Type of offset** – offset between the individual infeeds in the cutting direction
  - **0: Without offset**
  - **1: From left**
  - **2: From right**
  - **3: Alternating left/right**
- **V**: **Type of infeed**
  - **0: Const. mach. X-section**
  - **1: Const. infeed**
  - **2: EPL with distrib. of cuts**
  - **3: EPL w/o distrib. of cuts**
  - **4: MANUALplus 4110**
  - **5: Constant infeed (4290)**
  - **6: Const. w/ distrib. (4290)**
- **R**: **Remaining cut depth (V=4)**
- **I**: **Max. approach**
- **IC**: **Number of cuts** (only if **I** is not programmed)
- **B**: **Run-in lgth** to obtain the programmed spindle speed and feed rate (default: 2 * **Thread pitch F1**)
- **P**: **Overflow length**
- **C**: **Start angle**
- **Q**: **Number no-load.**

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **Thread cutting**
- Affected parameters: **F**, **S**

Unit G352 API thread

This unit cuts a single or multi-start API thread. The **Thread depth** decreases at the thread run-out.

Unit name: **G352_API** / cycle: **G352**

Further information: "Tapered API thread G352", Page 408

Thread form:

- **O**: **Thread location:**
  - **0**: Internal thread (infeed in +X)
  - **1**: External thread (infeed in -X)
- **X1**, **Z1**: **Start point thread**
- **X2**, **Z2**: **End point thread**
- **W**: **Taper angle** (range: –45° < **W** < 45°)
- **WE**: **Run-out angle** (reference: Z axis; 0° < **WE** < 90°; default: 12°)
- **F1**: **Thread pitch**
- **U**: **Thread depth**

Cycle form:

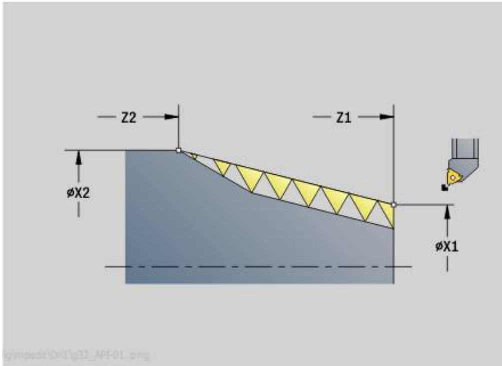
- **I**: **Max. approach**
- **H**: **Type of offset** – offset between the individual infeeds in the cutting direction
  - **0: Without offset**
  - **1: From left**
  - **2: From right**
  - **3: Alternating left/right**
- **V**: **Type of infeed**
  - **0: Const. mach. X-section**
  - **1: Const. infeed**
  - **2: EPL with distrib. of cuts**
  - **3: EPL w/o distrib. of cuts**
  - **4: MANUALplus 4110**
  - **5: Constant infeed (4290)**
  - **6: Const. w/ distrib. (4290)**
- **A**: **Approach ang.** (range: –60° < **A** < 60°; default: 30°)
- **R**: **Remaining cut depth (V=4)**
- **C**: **Start angle**
- **D**: **No.gears**
- **Q**: **Number no-load.**

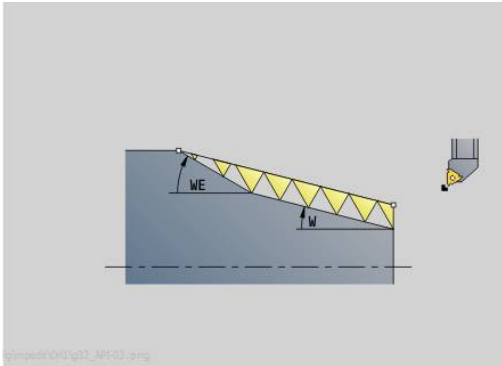
Further forms:

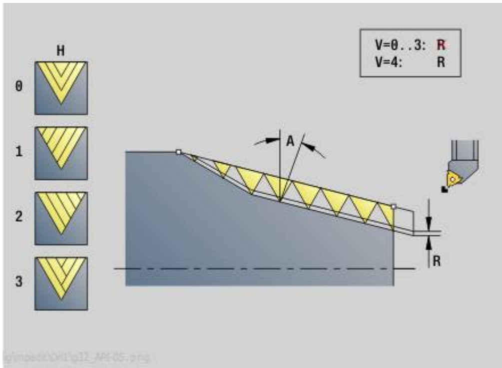
Further information: "smart.Turn unit", Page 98

Access to the technology database:

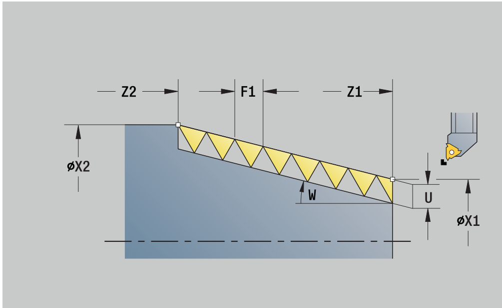
- Machining operation: **Thread cutting**
- Affected parameters: **F**, **S**

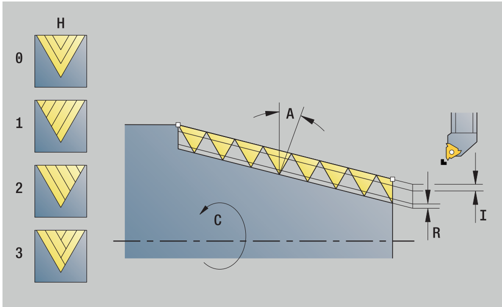






Unit G32 Tapered thread





The unit cuts a single or multi-start tapered internal or external thread.

Unit name: **G32_KEG** / cycle: **G32**

Further information: "Single thread cycle G32", Page 403

> **NOTICE**
>
> **Danger of collision!**
>
> If you modify the angle of infeed or the thread depth, the control shifts the position of the thread in an axial direction. In this case, the tool no longer hits any existing thread grooves and the thread flanks are destroyed. Danger of collision during reworking!
>
> - Be sure to compensate only the tool, not the thread parameters

Thread form:

- **O**: **Thread location:**
  - **0**: Internal thread (infeed in +X)
  - **1**: External thread (infeed in -X)
- **X1**, **Z1**: **Start point thread**
- **X2**, **Z2**: **End point thread**
- **W**: **Taper angle** (range: –45° < **W** < 45°)
- **F1**: **Thread pitch**
- **U**: **Thread depth**
- **KE**: **Run-out position:**
  - **0: At end**
  - **1: At beginning**
- **K**: **Thread runout length**

Cycle form:

- **I**: **Max. approach**
- **IC**: **Number of cuts** (only if **I** is not programmed)
- **H**: **Type of offset** – offset between the individual infeeds in the cutting direction
  - **0: Without offset**
  - **1: From left**
  - **2: From right**
  - **3: Alternating left/right**
- **V**: **Type of infeed**
  - **0: Const. mach. X-section**
  - **1: Const. infeed**
  - **2: EPL with distrib. of cuts**
  - **3: EPL w/o distrib. of cuts**
  - **4: MANUALplus 4110**
  - **5: Constant infeed (4290)**
  - **6: Const. w/ distrib. (4290)**
- **A**: **Approach ang.** (range: –60° < **A** < 60°; default: 30°)
- **R**: **Remaining cut depth (V=4)**

- **WE**: **Lift off method with K=0** (default: 0)
  - **0: G0 at end**
  - **1: Lift-off in thread**
- **C**: **Start angle**
- **D**: **No.gears**
- **Q**: **Number no-load.**
- **E**: **Variable gr.** (default: 0)
  Increases/decreases the pitch per revolution by **E**.

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **Thread cutting**
- Affected parameters: **F**, **S**

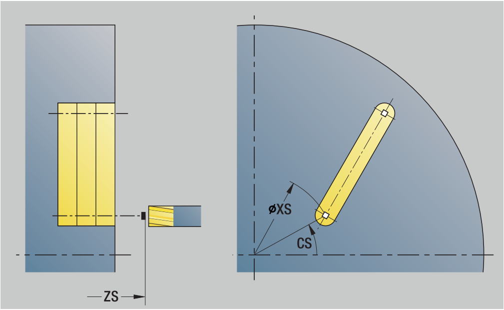
4.9 Units – Millg / C axis, face, C axis, ICP face (option 55)

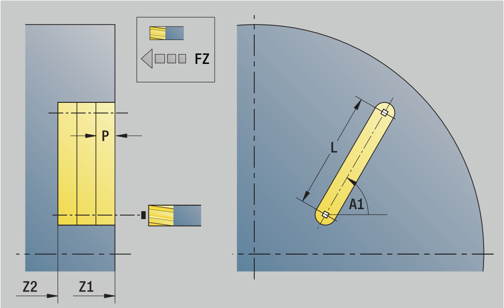
Unit G791 Linear slot, front face

The unit mills a slot from the starting position to the end point on the face of the workpiece. The slot width equals the diameter of the milling cutter.

Unit name: **G791_Nut_Stirn_C** / cycle: **G791**

Further information: "Linear slot, front face G791", Page 453





Cycle form:

- **Z1**: **Upper edge of milling**
- **Z2**: **Milling floor**
- **L**: **Slot length**
- **A1**: **Angle to X axis** (default: 0°)
- **X1**, **C1**: **Polar slot target point**
- **XK**, **YK**: **Cartes. slot target pnt**
- **P**: **Maximum infeed**
- **FZ**: **Approach feed** (default: active feed rate)

Further forms:

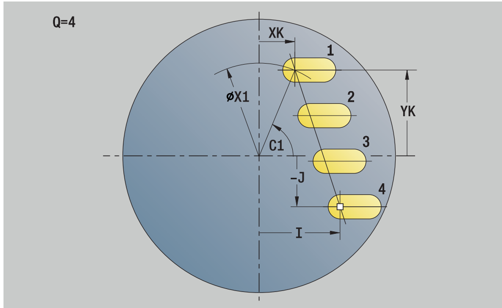
Further information: "smart.Turn unit", Page 98

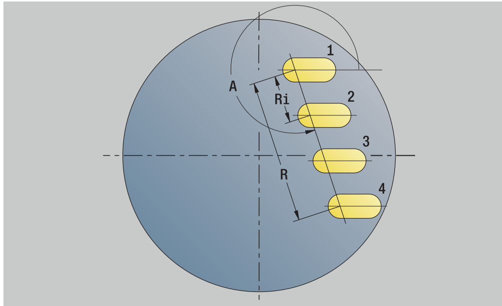
Access to the technology database:

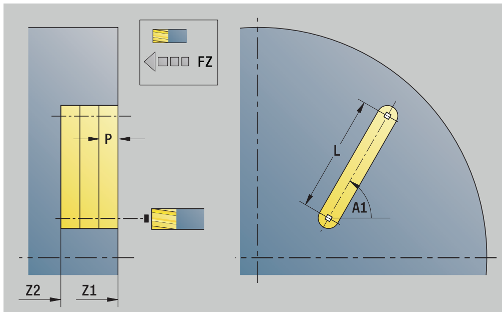
- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G791 Slot in linear pattern, front face

The unit machines a linear slot pattern in which the individual features are arranged at a regular spacing on the face of the workpiece. The starting points of the slots correspond to the pattern positions. You define the length and the position of the slots in the unit. The slot width equals the diameter of the milling cutter.







Unit name: **G791_Lin_Stirn_C** / cycle: **G791**

Further information: "Linear slot, front face G791", Page 453

Pattern form:

- **Q**: **Number of slots**
- **X1**, **C1**: **Polar starting point**
- **XK**, **YK**: **Cartesian starting pnt**
- **I**, **J**: **End point (XK)** and **(YK)**
- **Ii**, **Ji**: **Distance (XKi)** and **(YKi)**
- **R**: **Dist. to first/last contour**
- **Ri**: **Length** – **Incremental distance**
- **A**: **Pattern ang.** (reference: XK axis)

Cycle form:

- **Z1**: **Upper edge of milling**
- **Z2**: **Milling floor**
- **L**: **Slot length**
- **A1**: **Angle to X axis** (default: 0°)
- **P**: **Maximum infeed**
- **FZ**: **Approach feed** (default: active feed rate)

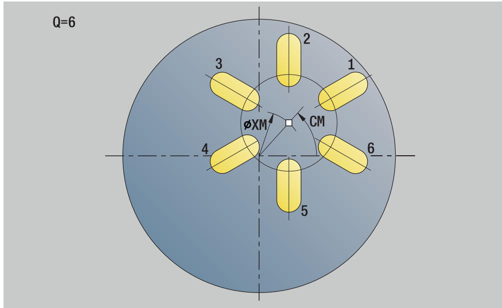
Further forms:

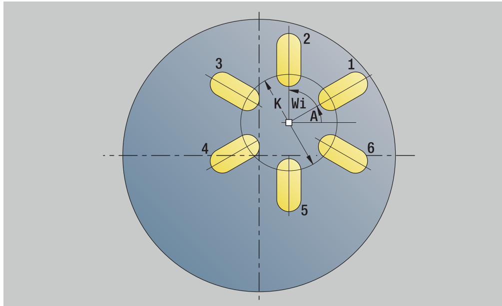
Further information: "smart.Turn unit", Page 98

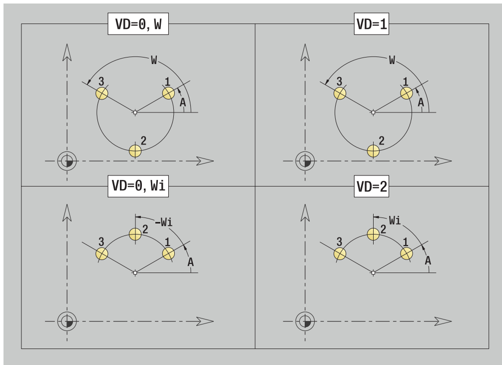
Access to the technology database:

- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G791 Slot in circ. pattern, front face







The unit machines a circular slot pattern in which the individual features are arranged at a regular spacing on the face of the workpiece. The starting points of the slots correspond to the pattern positions. You define the length and the position of the slots in the unit. The slot width equals the diameter of the milling cutter.

Unit name: **G791_Cir_Stirn_C** / cycle: **G791**

Further information: "Linear slot, front face G791", Page 453

Pattern form:

- **Q**: **Number of slots**
- **XM**, **CM**: **Polar center point**
- **XK**, **YK**: **Cartesian center point**
- **A**: **Start angle**
- **Wi**: **End angle – Angle increment**
- **K**: **Pattern diameter**
- **W**: **Final angle**
- **V**: **Rotation dir.** (default: 0)
  - **V** = 0, without **W**: Figures are arranged on a full circle
  - **V** = 0, with **W**: Figures are arranged on the longer circular arc
  - **V** = 0, with **Wi**: The algebraic sign of **Wi** defines the direction (**Wi** < 0: clockwise)
  - **V** = 1, with **W**: Clockwise
  - **V** = 1, with **Wi**: Clockwise (algebraic sign of **Wi** has no effect)
  - **V** = 2, with **W**: Counterclockwise
  - **V** = 2, with **Wi**: Counterclockwise (algebraic sign of **Wi** has no effect)

Cycle form:

- **Z1**: **Upper edge of milling**
- **Z2**: **Milling floor**
- **L**: **Slot length**
- **A1**: **Angle to X axis** (default: 0°)
- **P**: **Maximum infeed**
- **FZ**: **Approach feed** (default: active feed rate)

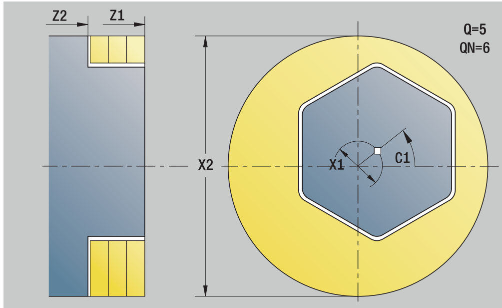
Further forms:

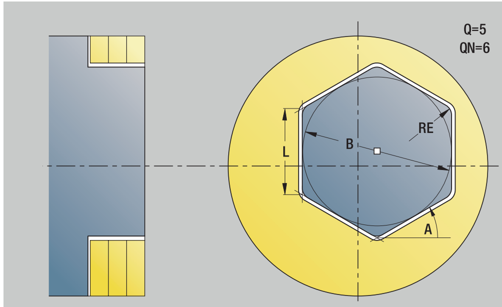
Further information: "smart.Turn unit", Page 98

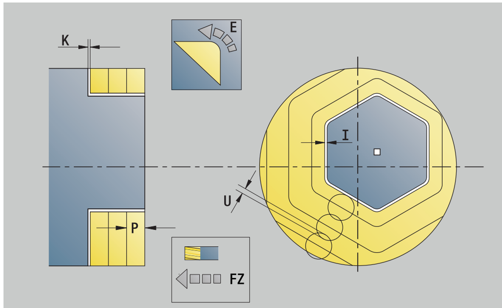
Access to the technology database:

- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G797 Face milling C







Depending on **Q**, the unit mills surfaces or the defined figure. The unit cuts the material around the figures.

Unit name: **G797_Stirnfr_C** / cycle: **G797**

Further information: "Area milling on front face G797", Page 460

Figure form:

- **Q**: **Type of figure**
  - **0: Complete circle**
  - **1: Single surface**
  - **2: Width across flats**
  - **3: Triangle**
  - **4: Rectangle / square**
  - **5: Polygon**
- **QN**: **No. of polygon corners** (only with **Q** = **5: Polygon**)
- **X1**: **Dia. of figure center**
- **C1**: **Angle of figure center** (default: **Spindle angle C**)
- **Z1**: **Upper edge of milling**
- **Z2**: **Milling floor**
- **X2**: **Limit diameter**
- **L**: **Edge length**
- **B**: **Width/Width across flats**
- **RE**: **Rounding radius** (default: 0)
- **A**: **Angle to X axis** (default: 0°)

Cycle form:

- **QK**: **Machining operation**
  - Roughing
  - Finishing
- **J**: **Mill direction**
  - **0: Unidirectional**
  - **1: Bidirectional**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Maximum infeed**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **FZ**: **Approach feed** (default: active feed rate)
- **E**: **Reduced feed**
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter

Further forms:

Further information: "smart.Turn unit", Page 98

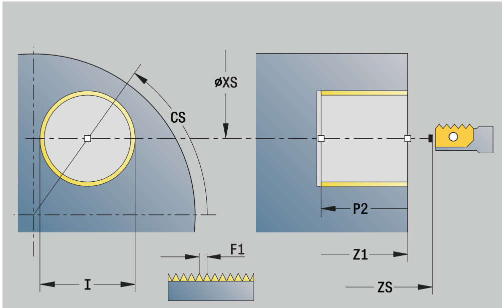
Access to the technology database:

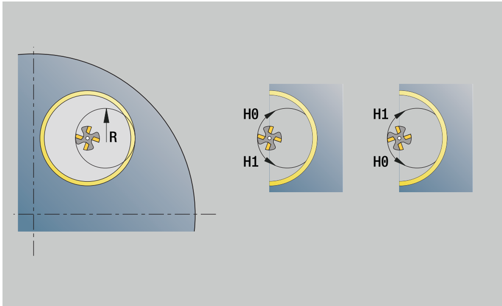
- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G799 Thread milling, front face C

The unit mills a thread in existing holes.

Place the tool at the center of the hole before calling **G799**. The cycle positions the tool to the **End point thread** inside the hole. Then, the tool approaches at **Apprch angle R** and mills the thread. With each rotation, the tool moves by the **Thread pitch F1**. Then, the control retracts the tool and returns it to the **Start point**. With parameter **V**, you can program whether the thread is to be milled in one rotation or, with single-point tools, in several rotations.





Unit name: **G799_Gewindefr_C** / cycle: **G799**

Further information: "Thread milling axial G799", Page 439

Position form:

- **Z1**: **Start point drill**
- **P2**: **Thread depth**
- **I**: **Thread diameter**
- **F1**: **Thread pitch**

Cycle form:

- **J**: **Direction of thread:**
  - **0: Right-hand thread**
  - **1: Left-hand thread**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **V**: **Milling method**
  - **0: One revolution** – the thread is milled in a 360-degree helix
  - **1: Two or more revolutions** – the thread is milled in several helix paths (single-point tool)
- **R**: **Approach radius**
- **FK**: **Finished part contour** – name of the contour to be machined
- **NS**: **Block number of contour** – reference to the contour description

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

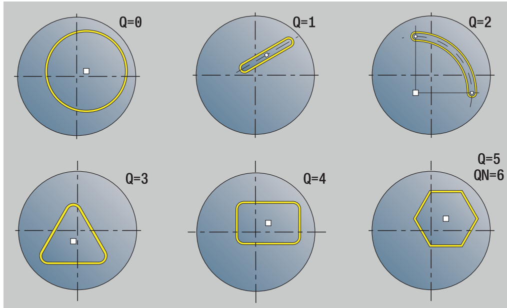
- Machining operation: Finish-milling
- Affected parameters: **F**, **S**

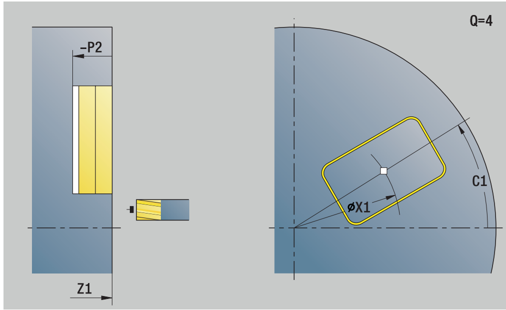
Unit G840 Contour mllng, figures front face C

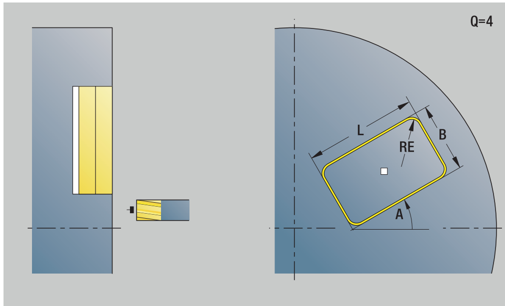
The unit mills the contour defined by **Q** on the face of the workpiece.

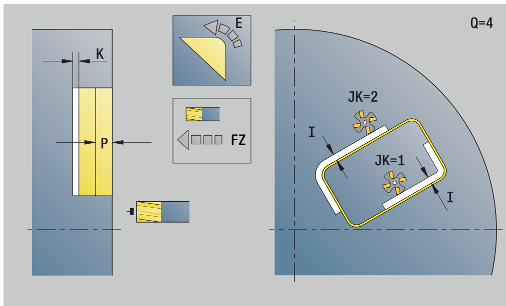
Unit name: **G840_Fig_Stirn_C** / cycle: **G840**

Further information: "G840 – milling", Page 466









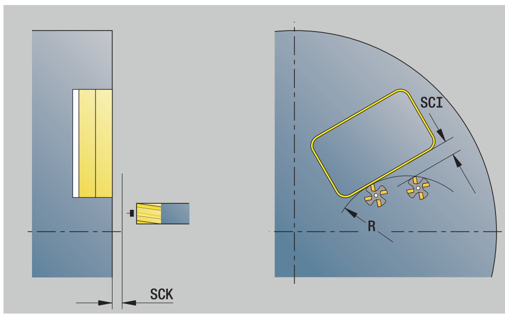


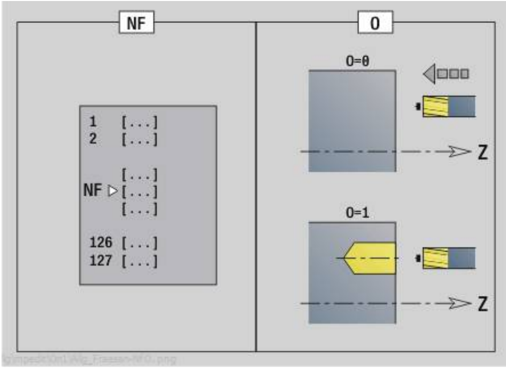
Figure form:

- **Q**: **Type of figure**
  - **0: Complete circle**
  - **1: Linear slot**
  - **2: Circular slot**
  - **3: Triangle**
  - **4: Rectangle / square**
  - **5: Polygon**
- **QN**: **No. of polygon corners** (only with **Q** = **5: Polygon**)
- **X1**: **Dia. of figure center**
- **C1**: **Angle of figure center** (default: **Spindle angle C**)
- **Z1**: **Upper edge of milling**
- **P2**: **Depth of figure**
- **L**: **+edge lngth/-width a. flats**
  - **L** > 0: **Edge length**
  - **L** < 0: **Width across** (inside diameter) for polygon
- **B**: **Width of rectangle**
- **RE**: **Rounding radius** (default: 0)
- **A**: **Angle to X axis** (default: 0°)
- **Q2**: **Rot. direction of slot** (only with **Q** = **2: Circular slot**)
  - **cw**: In clockwise direction
  - **ccw**: In counterclockwise direction
- **W**: **Angle of slot end point** (only with **Q** = **2: Circular slot**)

> Program only the parameters relevant to the selected figure type.

Cycle form:

- **JK**: **Cutter position**
  - **0: On the contour**
  - **1: Within the contour**
  - **2: Outside the contour**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Maximum infeed**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **FZ**: **Approach feed** (default: active feed rate)
- **E**: **Reduced feed**
- **R**: **Approach radius**



- **O**: **Plunging behavior** (default: 0)
  - **0: Straight** – the cycle moves the tool to the starting point; the tool plunges at feed rate and mills the contour
  - **1: In predrilling** – the control positions the tool above the predrill hole position; the tool plunges and mills the contour
- **NF**: **Position mark** (only with **O** = 1)

Global form:

- **RB**: **Return plane**

Further parameters:

Further information: "Global form", Page 103

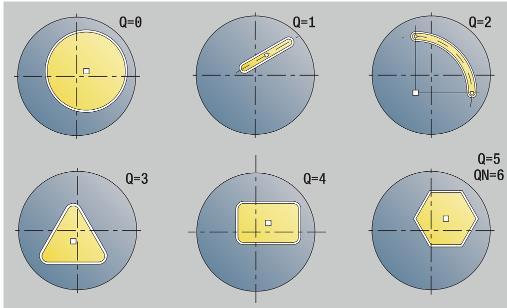
Further forms:

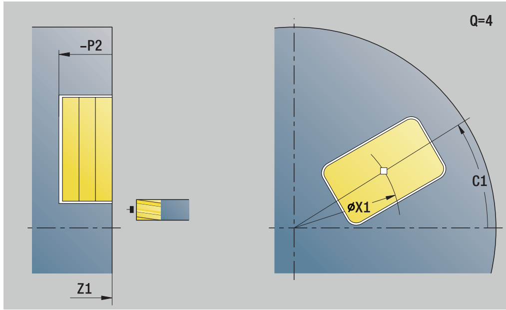
Further information: "smart.Turn unit", Page 98

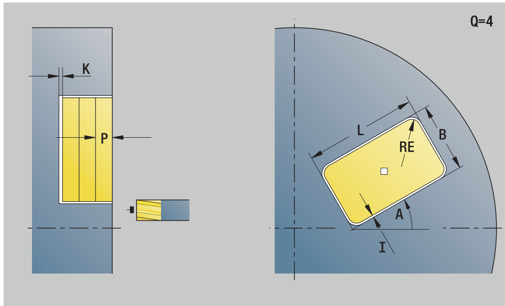
Access to the technology database:

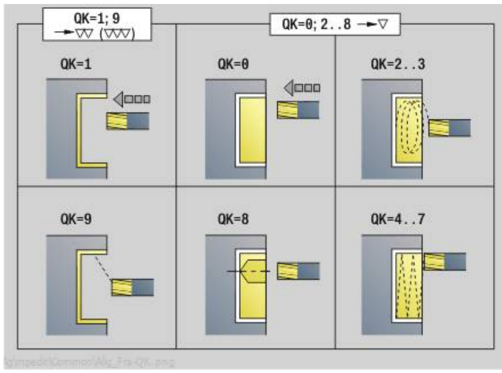
- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G84X Pocket mllng, figures front face C









The unit mills the pocket defined by **Q**. In **QK**, select the Machining operation (roughing/finishing) and the plunging strategy.

Unit name: **G84x_Fig_Stirn_C** / cycles: **G845**; **G846**

Further information: "G845 – milling", Page 474

Further information: "Pocket milling - finishing G846", Page 478

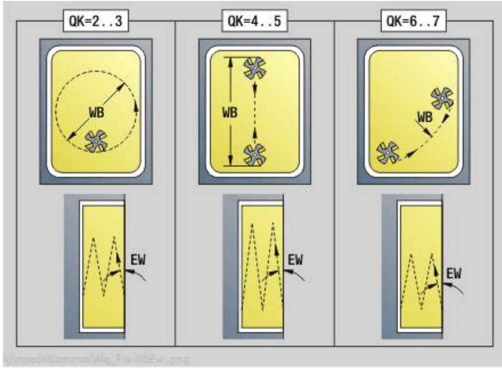
Figure form:

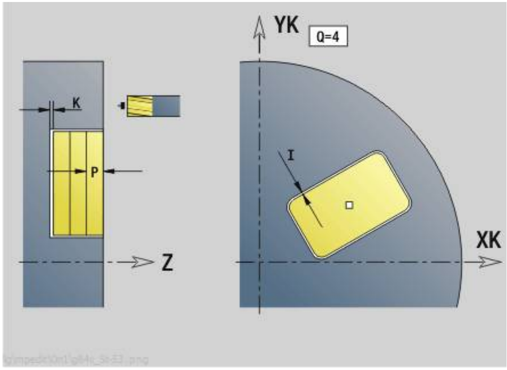
- **Q**: **Type of figure**
  - **0: Complete circle**
  - **1: Linear slot**
  - **2: Circular slot**
  - **3: Triangle**
  - **4: Rectangle / square**
  - **5: Polygon**
- **QN**: **No. of polygon corners** (only with **Q** = **5: Polygon**)
- **X1**: **Dia. of figure center**
- **C1**: **Angle of figure center** (default: **Spindle angle C**)
- **Z1**: **Upper edge of milling**
- **P2**: **Depth of figure**
- **L**: **+edge lngth/-width a. flats**
  - **L** > 0: **Edge length**
  - **L** < 0: **Width across** (inside diameter) for polygon
- **B**: **Width of rectangle**
- **RE**: **Rounding radius** (default: 0)
- **A**: **Angle to X axis** (default: 0°)
- **Q2**: **Rot. direction of slot** (only with **Q** = **2: Circular slot**)
  - **cw**: In clockwise direction
  - **ccw**: In counterclockwise direction
- **W**: **Angle of slot end point** (only with **Q** = **2: Circular slot**)

> **i** Program only the parameters relevant to the selected figure type.

Cycle form:

- **QK**: **Machining operation** and plunging strategy
  - **0: Roughing**
  - **1: Finishing**
  - **2: Helical roughing, manual**
  - **3: Helical roughing, automatic**
  - **4: Recipr. linear roughing, manual**
  - **5: Recipr. linear roughing, auto**
  - **6: Reciproc. circ. roughing, manual**
  - **7: Recipr. circular roughing, auto**
  - **8: Plunge rough at predrill pos.**
  - **9: Finishing with 3-D approach arc**





- **JT**: **Machining direction**
  - **0: From the inside out**
  - **1: From the outside in**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Maximum infeed**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **FZ**: **Approach feed** (default: active feed rate)
- **E**: **Reduced feed**
- **R**: **Approach radius**
- **WB**: **Plunging length**
- **EW**: **Plunging angle**
- **NF**: **Position mark** (only with **QK** = 8)
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter

Global form:

- **RB**: **Return plane**

Further parameters:

Further information: "Global form", Page 103

Further forms:

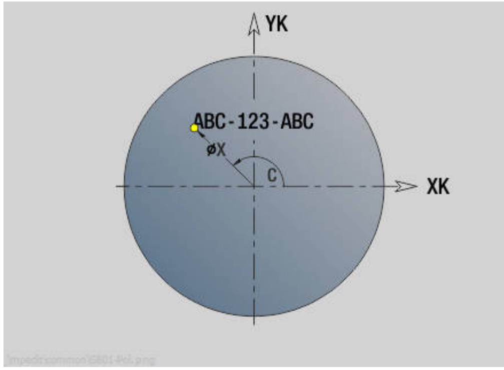
Further information: "smart.Turn unit", Page 98

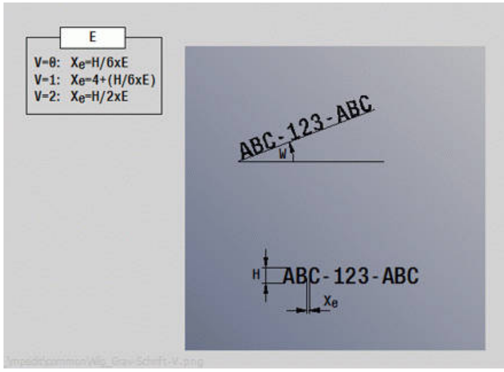
Access to the technology database:

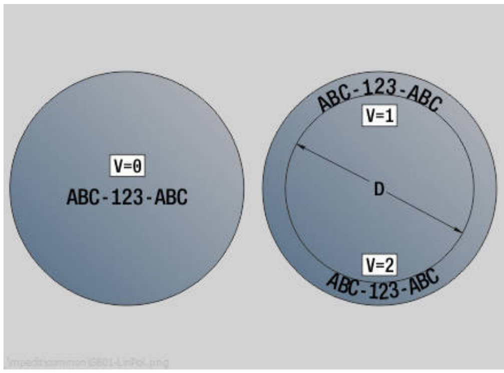
- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G801 Engraving in C axis on face

The unit engraves character strings in linear or angular arrangement on the face of the workpiece. Diacritics and special characters that you cannot enter in **smart.Turn** operating mode can be defined, character by character, in **NF**. If you program **Q** = 1 (**Continue from last text**), tool change and pre-positioning are suppressed. The technological data of the previous engraving cycle apply.







Unit name: **G801_GRA_STIRN_C** / cycle: **G801**

Further information: "Engraving on front face G801", Page 487

Position form:

- **X**, **C**: **Start point** and **Start. angle** (in polar coordinates)
- **XK**, **YK**: **Start point** (in Cartesian coordinates)
- **Z**: **Final point** – Z position, infeed depth during milling
- **RB**: **Return plane**

Cycle form:

- **TXT**: **Text** to be engraved
- **NF**: **Char. no.** – ASCII code of the character to be engraved
- **H**: **Font height**
- **E**: **Distance factor**
  The spacing between the characters is calculated according to the following formula: **H** / 6 * **E**
- **W**: **Inclinat. ang.** of the character string
- **FZ**: **Plunging feed rate factor** (plunging feed rate = current feed rate * **FZ**)
- **V**: **Execution (linear/polar)**
  - **0: Linear**
  - **1: Arched above**
  - **2: Arched below**
- **D**: **Reference diameter**
- **Q**: **Continue from last text**
  - **0** (**No**): Engraving starts at the starting point
  - **1** (**Yes**): Engraving starts at the tool position
- **O**: **Mirror writing**
  - **0** (**No**): Engraving is not mirrored
  - **1** (**Yes**): Engraving is mirrored (mirror writing)
- **NS**: **Block number of contour** – reference to the contour description

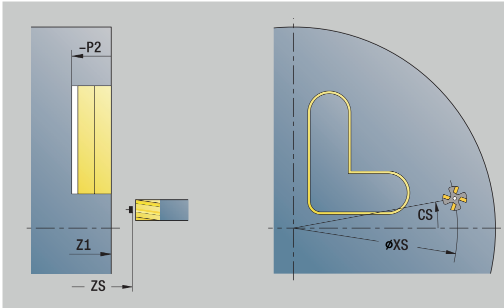
Further forms:

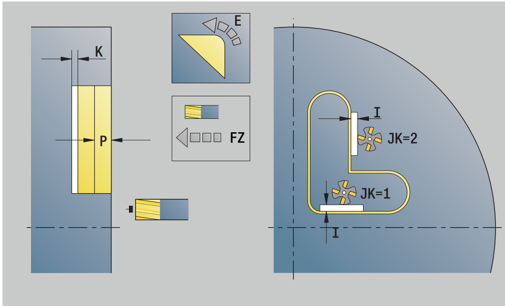
Further information: "smart.Turn unit", Page 98

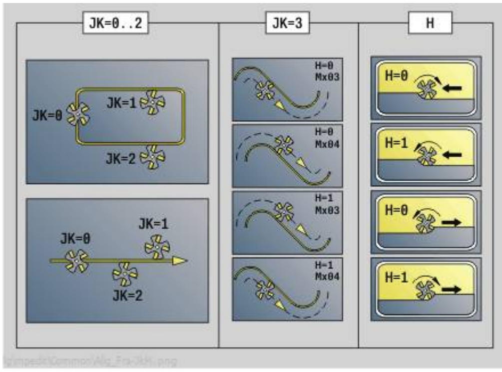
Access to the technology database:

- Machining operation: **Engraving**
- Affected parameters: **F**, **S**

Unit G840 ICP contour milling, front face C







The unit mills the contour defined with **ICP** on the face of the workpiece.

Unit name: **G840_Kon_C_Stirn** / cycle: **G840**

Further information: "G840 – milling", Page 466

Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
- **BF**: **Machine form elements** (default: 0)
  A chamfer/rounding arc is machined
  - **0: No machining**
  - **1: At beginning**
  - **2: At end**
  - **3: At beginning and end**
  - **4: Only chamfer/rounding** is machined—not the basic element (requirement: the contour section consists of a single element)
- **Z1**: **Upper edge of milling**
- **P2**: **Depth of contour**

Cycle form:

- **JK**: **Cutter position**
  - **0: On the contour**
  - **1: Within/left of contour**
  - **2: Outside/right of contour**
  - **3: Depending on H and MD**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Maximum infeed**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **FZ**: **Approach feed** (default: active feed rate)
- **E**: **Reduced feed**
- **R**: **Approach radius**
- **O**: **Plunging behavior** (default: 0)
  - **0: Straight** – the cycle moves the tool to the starting point; the tool plunges at feed rate and mills the contour
  - **1: In predrilling** – the control positions the tool above the predrill hole position; the tool plunges and mills the contour
- **NF**: **Position mark** (only with **O** = 1)
- **RB**: **Return plane**

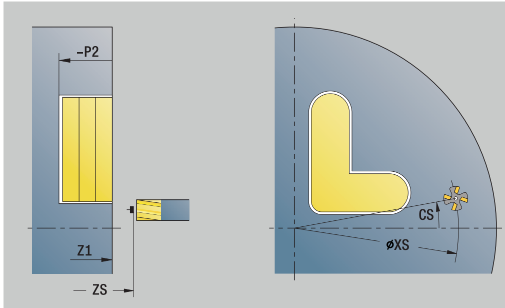
Further forms:

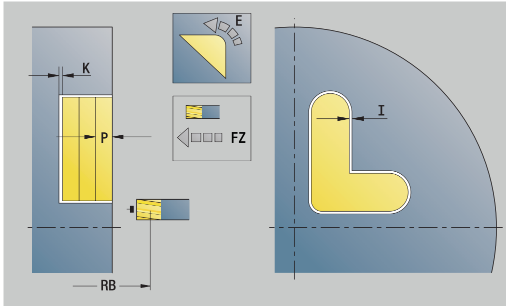
Further information: "smart.Turn unit", Page 98

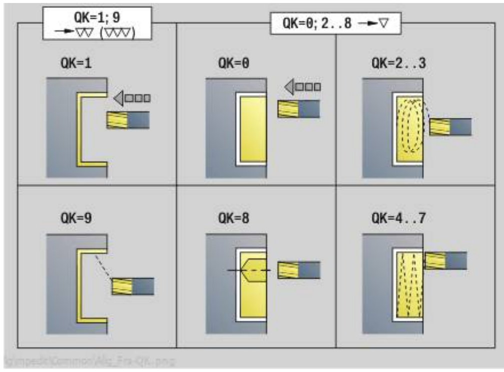
Access to the technology database:

- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G845 ICP pocket milling, front face C







The unit mills the pocket defined by **Q**. In **QK**, select the machining operation (roughing/finishing) and the plunging strategy.

Unit name: **G845_Tas_C_Stirn** / cycles: **G845**; **G846**

Further information: "G845 – milling", Page 474

Further information: "Pocket milling - finishing G846", Page 478

Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
- **Z1**: **Upper edge of milling**
- **P2**: **Depth of contour**
- **NF**: **Position mark** (only with **QK** = 8)
- **FZ**: **Approach feed** (default: active feed rate)
- **E**: **Reduced feed**
- **FP**: **Infeed rate in the plane**

Cycle form:

- **QK**: **Machining operation** and plunging strategy
  - **0: Roughing**
  - **1: Finishing**
  - **2: Helical roughing, manual**
  - **3: Helical roughing, automatic**
  - **4: Recipr. linear roughing, manual**
  - **5: Recipr. linear roughing, auto**
  - **6: Reciproc. circ. roughing, manual**
  - **7: Recipr. circular roughing, auto**
  - **8: Plunge rough at predrill pos.**
  - **9: Finishing with 3-D approach arc**
- **JT**: **Machining direction**
  - **0: From the inside out**
  - **1: From the outside in**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Maximum infeed**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **R**: **Approach radius**
- **WB**: **Plunging length**
- **EW**: **Plunging angle**
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
- **RB**: **Return plane**

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

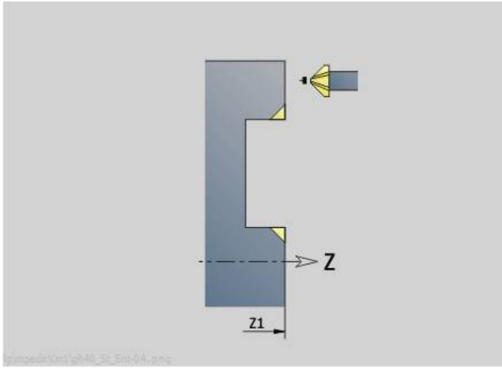
- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

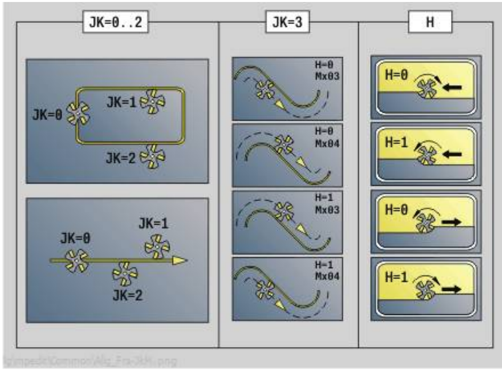
Unit G840 ICP deburring, front face C

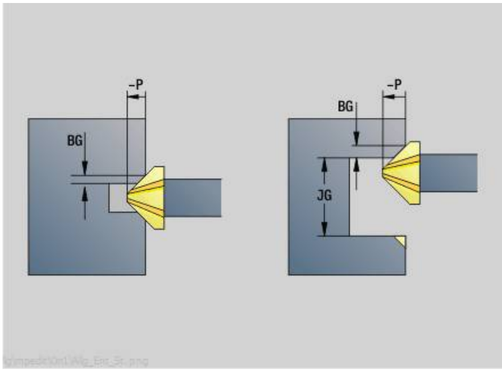
The unit deburrs the contour defined with **ICP** on the face of the workpiece.

Unit name: **G840_ENT_C_STIRN** / cycle: **G840**

Further information: "G840 – deburring", Page 470







Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **BF**: **Machine form elements** (default: 0)

  A chamfer/rounding arc is machined
  - **0: No machining**
  - **1: At beginning**
  - **2: At end**
  - **3: At beginning and end**
  - **4: Only chamfer/rounding** is machined—not the basic element (requirement: the contour section consists of a single element)
- **NE**: **Contour end block no.** – end of contour section
- **Z1**: **Upper edge of milling**

Cycle form:

- **JK**: **Cutter position**
  - **0: On the contour**
  - **1: Within/left of contour**
  - **2: Outside/right of contour**
  - **3: Depending on H and MD**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **BG**: **Cham. width** for deburring
- **JG**: **Preparation diameter**
- **P**: **Plunging depth** (indicated as a negative value)
- **I**: **Contour-parallel oversize**
- **R**: **Approach radius**
- **FZ**: **Approach feed** (default: active feed rate)
- **E**: **Reduced feed**
- **RB**: **Return plane**

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

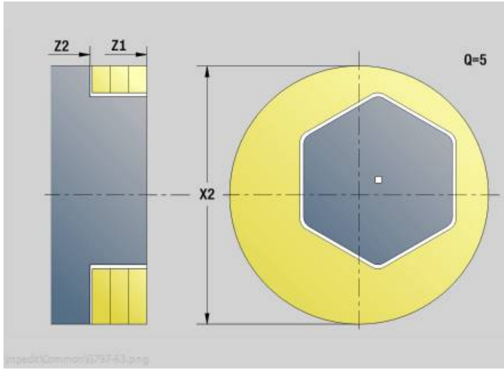
- Machining operation: **Deburring**
- Affected parameters: **F**, **S**

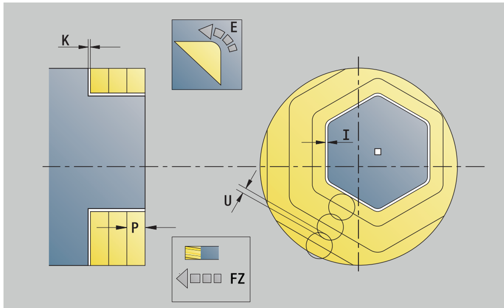
Unit G797 face milling ICP

The unit mills the contour defined with **ICP** on the face of the workpiece.

Unit name: **G797_ICP** / cycle: **G797**

Further information: "Area milling on front face G797", Page 460





Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **Z1**: **Upper edge of milling**
- **Z2**: **Milling floor**
- **X2**: **Limit diameter**

Cycle form:

- **QK**: **Machining operation**
  - Roughing
  - Finishing
- **J**: **Mill direction**
  - **0: Unidirectional**
  - **1: Bidirectional**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Maximum infeed**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **FZ**: **Approach feed** (default: active feed rate)
- **E**: **Reduced feed**
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G847 ICP Troch. contour mllng, front face C

The unit uses trochoidal milling to rough out the open or closed contour on the front face of the workpiece defined with **ICP**.

Unit name: **G847_KON_C_STIRN** / cycle: **G847**

Further information: "Trochoidal contour milling G847 ", Page 480

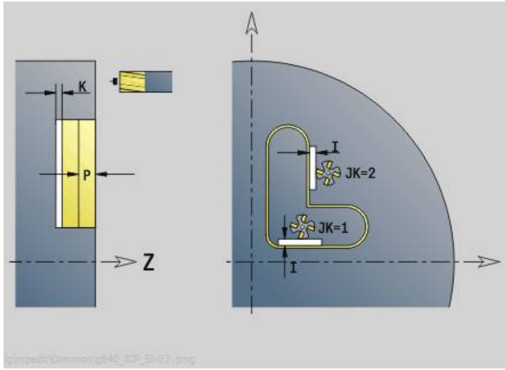
Contour form:

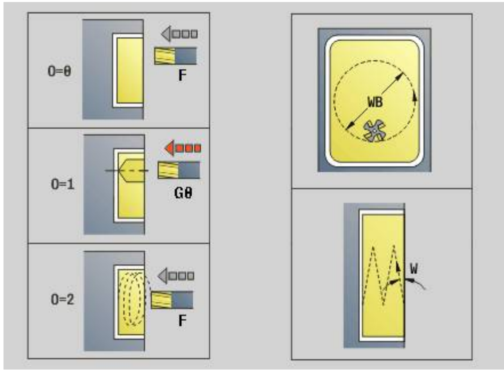
- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
- **BF**: **Machine form elements** (default: 0)

  A chamfer/rounding arc is machined
  - **0: No machining**
  - **1: At beginning**
  - **2: At end**
  - **3: At beginning and end**
  - **4: Only chamfer/rounding** is machined—not the basic element (requirement: the contour section consists of a single element)
- **Z1**: **Upper edge of milling**
- **P2**: **Depth of contour**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **RB**: **Return plane** (default: back to start position)
- **NF**: **Position mark** (only with **O** = 1)

Cycle form:

- **JK**: **Cutter position**
  - **0: On the contour**
  - **1: Within/left of contour**
  - **2: Outside/right of contour**
- **H**: **Mill cutting direction** (default: 1)
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Maximum infeed**
- **BR**: **Trochoid width**
- **R**: **Radius for return**
- **FP**: **Feed rate for return** (default: active feed rate)
- **AL**: **Retraction path for return**





- **O**: **Plunging behavior** (default: 2)
  - **O** = 0 (vertical plunging): The cycle moves the tool to the starting point; the tool plunges at the feed rate for infeed and mills the contour
  - **O** = 1 (vertical plunging, e.g. at the predrilled position):
    - If **NF** is programmed: The control positions the milling cutter above the first predrill hole position; the tool plunges at rapid traverse to safety clearance and mills the first area. If applicable, the cycle positions the tool to the next predrill hole position and mills the next area etc.
    - If **NF** is not programmed: The cycle plunges at the current position at rapid traverse and mills the area. If applicable, position the tool to the next predrill hole position and mill the next area etc.
  - **O** = 2 (plunge in helical motion): The tool plunges at the current position at the angle **W** and mills full circles with the diameter **WB**.
- **FZ**: **Approach feed** (default: active feed rate)
- **EW**: **Plunging angle**
- **WB**: **Diameter of the helix** (default: helix diameter = 1.5 * milling diameter)
- **U**: **Overlap factor** – overlap of milling paths = **U** * milling diameter (default: 0.9)
- **HCC**: **Contour smoothing**
  - **0: No smoothing cut**
  - **1: With smoothing cut**

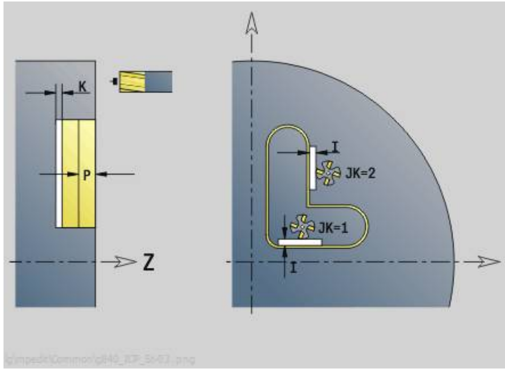
Further forms:

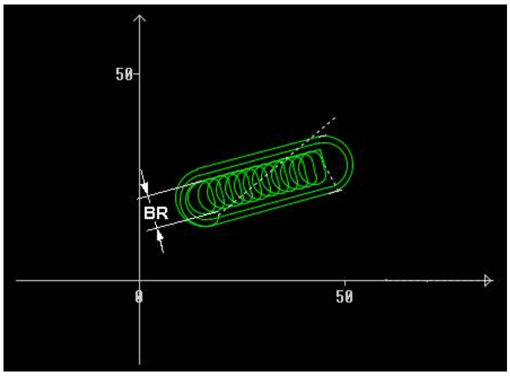
Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G848 ICP Troch. pocket mllng, front face C





The unit uses trochoidal milling to rough out the figure or figure pattern on the front face defined with **ICP**.

Unit name: **G848_TAS_C_STIRN** / cycle: **G848**

Further information: "Trochoidal pocket milling G847 ", Page 482

Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **Z1**: **Upper edge of milling**
- **P2**: **Depth of contour**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **RB**: **Return plane** (default: back to start position)
- **NF**: **Position mark** (only with **O** = 1)

Cycle form:

- **H**: **Mill cutting direction** (default: 1)
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Maximum infeed**
- **BR**: **Trochoid width**
- **R**: **Radius for return**
- **FP**: **Feed rate for return** (default: active feed rate)
- **AL**: **Retraction path for return**
- **O**: **Plunging behavior** (default: 2)
  - **O**=0 (vertical plunging): The cycle moves the tool to the starting point; the tool plunges at the feed rate for infeed and mills the figure
  - **O** = 1 (vertical plunging, e.g. at the predrilled position):
    - If **NF** is programmed: The control positions the milling cutter above the first predrill hole position; the tool plunges at rapid traverse to safety clearance and mills the first area. If applicable, the cycle positions the tool to the next predrill hole position and mills the next area etc.
    - If **NF** is not programmed: The cycle plunges at the current position at rapid traverse and mills the area. If applicable, position the tool to the next predrill hole position and mill the next area etc.
  - **O** = 2 (plunge in helical motion): The tool plunges at the current position at the angle **W** and mills full circles with the diameter **WB**.
- **FZ**: **Approach feed** (default: active feed rate)
- **EW**: **Plunging angle**
- **WB**: **Diameter of the helix** (default: helix diameter = 1.5 * milling diameter)
- **U**: **Overlap factor** – overlap of milling paths = **U** * milling diameter (default: 0.9)

- **J**: **Machining operation**
  - **0: Complete**
  - **1: W/o corner machining**
  - **2: Only corner machining**

> For slots and rectangles, you need to program width **BR** of the trochoidal tool path; for circles and polygons, this is not necessary.

Further forms:

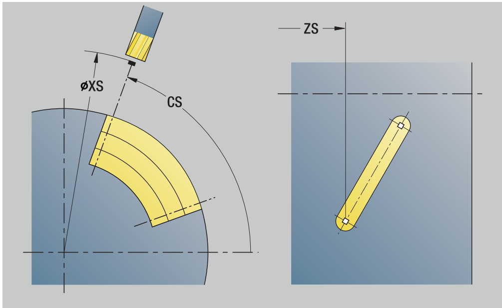
Further information: "smart.Turn unit", Page 98

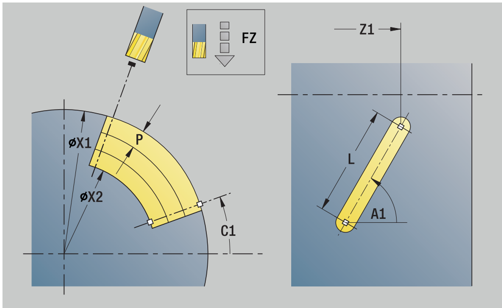
Access to the technology database:

- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

4.10 Units – Milling / C axis, lateral, C axis, ICP latrl. (option 55)

Unit G792 Linear slot, lat. surface





The unit mills a slot from the starting position to the end point on the lateral surface. The slot width equals the diameter of the milling cutter.

Unit name: **G792_Nut_MANT_C** / cycle: **G792**

Further information: "Linear slot, lat. surface G792", Page 454

Cycle form:

- **X1**: **Upper edge of milling**
- **X2**: **Milling floor**
- **L**: **Slot length**
- **A1**: **Angle to Z axis** (default: 0°)
- **Z1**, **C1**: **Polar slot target point**
- **P**: **Maximum infeed**
- **FZ**: **Approach feed** (default: active feed rate)

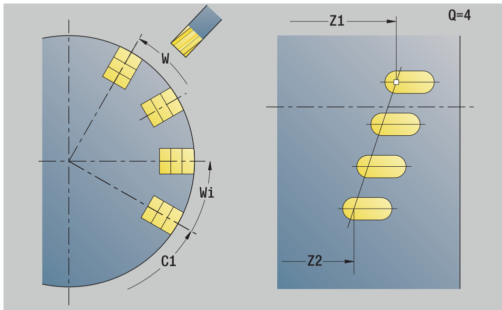
Further forms:

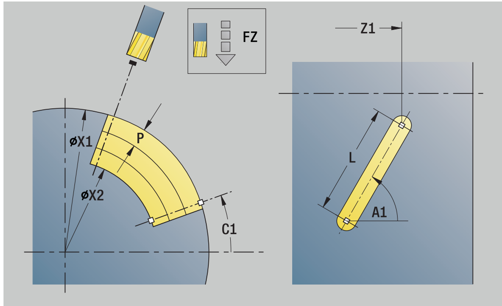
Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G792 Slot in lin. pattern, lat. surface





The unit machines a linear slot pattern in which the individual features are arranged at a regular spacing on the lateral surface. The **Start point** of the slots corresponds to the pattern positions. The **Slot length** and the **positions of the slots** are defined in the unit. The slot width equals the diameter of the milling cutter.

Unit name: **G792_Lin_Mant_C** / cycle: **G792**

Further information: "Linear slot, lat. surface G792", Page 454

Pattern form:

- **Q**: **Number of slots**
- **Z1**: **Start point pattern** – Position of first slot
- **C1**: **Start. angle**
- **Wi**: **End angle** – **Angle increment**
- **W**: **Final angle**
- **Z2**: **End point pattern**

Cycle form:

- **X1**: **Upper edge of milling**
- **X2**: **Milling floor**
- **L**: **Slot length**
- **A1**: **Angle to Z axis** (default: 0°)
- **P**: **Maximum infeed**
- **FZ**: **Approach feed** (default: active feed rate)

Further forms:

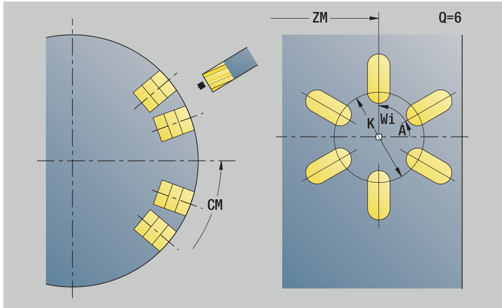
Further information: "smart.Turn unit", Page 98

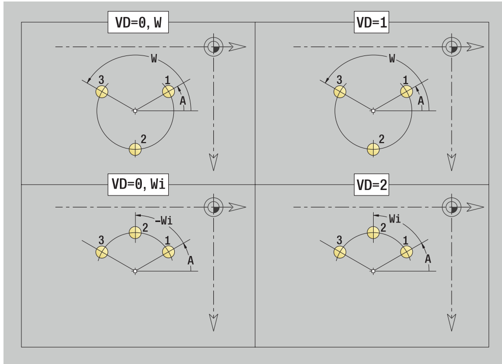
Access to the technology database:

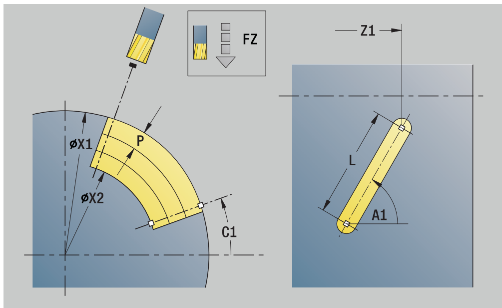
- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G792 Slot in circ. pattern, lat. surface

The unit machines a circular slot pattern in which the individual features are arranged at a regular spacing on the lateral surface. The **Start point** of the slots corresponds to the pattern positions. The **Slot length** and the **positions of the slots** are defined in the unit. The slot width equals the diameter of the milling cutter.







Unit name: **G792_Cir_Mant_C** / cycle: **G792**

Further information: "Linear slot, lat. surface G792", Page 454

Pattern form:

- **Q**: **Number of slots**
- **ZM**: **Center** of pattern
- **CM**: **Angle of pattern center**
- **A**: **Start angle**
- **Wi**: **End angle** – **Angle increment**
- **K**: **Pattern diameter**
- **W**: **Final angle**
- **V**: **Rotation dir.** (default: 0)
  - **V** = 0, without **W**: Figures are arranged on a full circle
  - **V** = 0, with **W**: Figures are arranged on the longer circular arc
  - **V** = 0, with **Wi**: The algebraic sign of **Wi** defines the direction (**Wi** < 0: clockwise)
  - **V** = 1, with **W**: Clockwise
  - **V** = 1, with **Wi**: Clockwise (algebraic sign of **Wi** has no effect)
  - **V** = 2, with **W**: Counterclockwise
  - **V** = 2, with **Wi**: Counterclockwise (algebraic sign of **Wi** has no effect)

Cycle form:

- **X1**: **Upper edge of milling**
- **X2**: **Milling floor**
- **L**: **Slot length**
- **A1**: **Angle to Z axis** (default: 0°)
- **P**: **Maximum infeed**
- **FZ**: **Approach feed** (default: active feed rate)

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

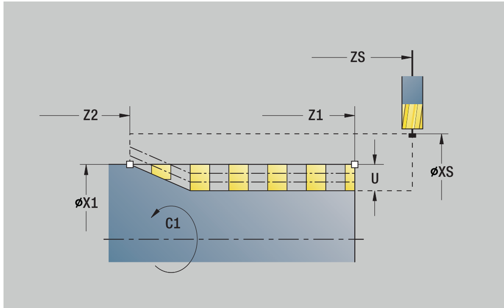
- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

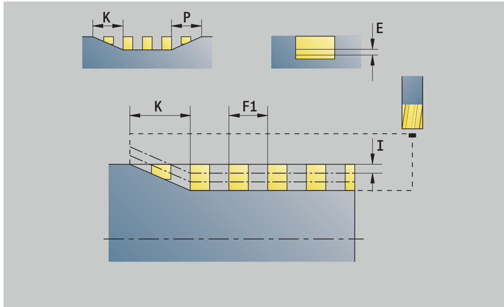
Unit G798 Helical slot milling

The unit mills a helical slot. The slot width equals the diameter of the milling cutter.

Unit name: **G798_WENDELNUT_C** / cycle: **G798**

Further information: "Helical slot milling G798", Page 462





Position form:

- **X1**: **Thread diameter**
- **C1**: **Start. angle**
- **Z1**: **Start point thread**
- **Z2**: **End point thread**
- **U**: **Thread depth**

Cycle form:

- **F1**: **Thread pitch**
- **J**: **Direction of thread:**
  - **0: Right-hand thread**
  - **1: Left-hand thread**
- **D**: **No.gears**
- **P**: **Run-in lgth**
- **K**: **Thread runout length**
- **I**: **Max. approach**
- **E**: **Cutting depth reduction**

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

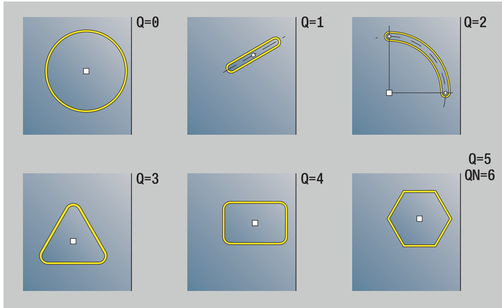
- Machining operation: Finish-milling
- Affected parameters: **F**, **S**

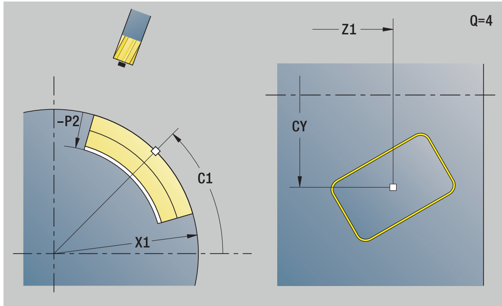
Unit G840 Contour mllng, figures lat. surf. C

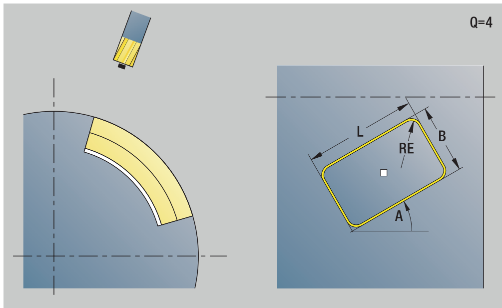
The unit mills the contour defined by **Q** on the lateral surface.

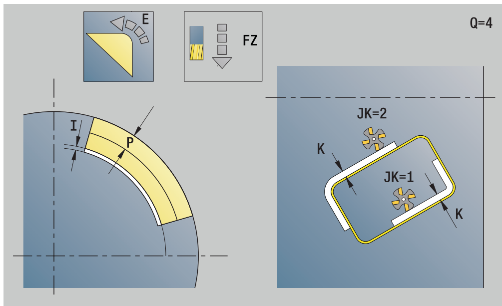
Unit name: **G840_Fig_Mant_C** / cycle: **G840**

Further information: "G840 – milling", Page 466









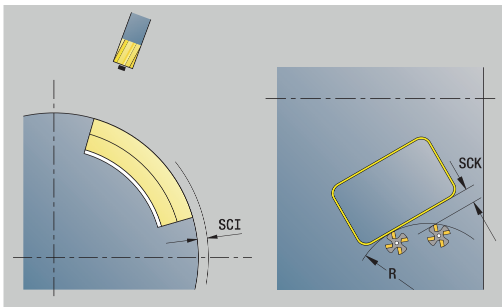


Figure form:

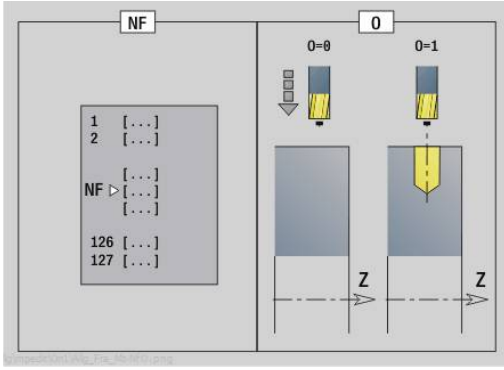
- **Q**: **Type of figure**
  - **0: Complete circle**
  - **1: Linear slot**
  - **2: Circular slot**
  - **3: Triangle**
  - **4: Rectangle / square**
  - **5: Polygon**
- **QN**: **No. of polygon corners** (only with **Q** = **5: Polygon**)
- **Z1**: **Figure center**
- **C1**: **Angle of figure center** (default: **Spindle angle C**)
- **CY**: **Fig. center, unr.lat. surf.**
- **X1**: **Upper edge of milling**
- **P2**: **Depth of figure**
- **L**: **+edge lngth/-width a. flats**
  - **L** > 0: **Edge length**
  - **L** < 0: **Width across** (inside diameter) for polygon
- **B**: **Width of rectangle**
- **RE**: **Rounding radius** (default: 0)
- **A**: **Angle to Z axis** (default: 0°)
- **Q2**: **Rot. direction of slot** (only with **Q** = **2: Circular slot**)
  - **cw**: In clockwise direction
  - **ccw**: In counterclockwise direction
- **W**: **Angle of slot end point** (only with **Q** = **2: Circular slot**)

> **i** Program only the parameters relevant to the selected figure type.

Cycle form:

- **JK**: **Cutter position**
  - **0: On the contour**
  - **1: Within the contour**
  - **2: Outside the contour**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Maximum infeed**
- **I**: **Infeed-direction oversize**
- **K**: **Contour-parallel oversize**
- **FZ**: **Approach feed** (default: active feed rate)
- **E**: **Reduced feed**
- **R**: **Approach radius**

- **O**: **Plunging behavior** (default: 0)
  - **0: Straight** – the cycle moves the tool to the starting point; the tool plunges at feed rate and mills the contour
  - **1: In predrilling** – the control positions the tool above the predrill hole position; the tool plunges and mills the contour
- **NF**: **Position mark** (only with **O** = 1)



Global form:

- **RB**: **Return plane**

Further parameters:

Further information: "Global form", Page 103

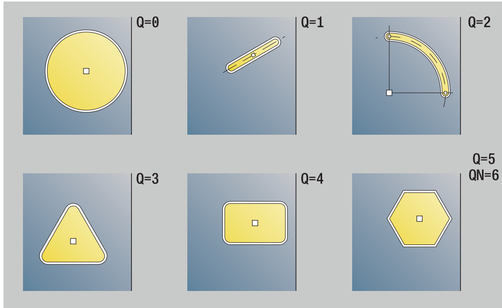
Further forms:

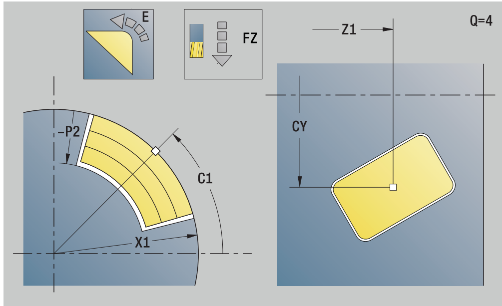
Further information: "smart.Turn unit", Page 98

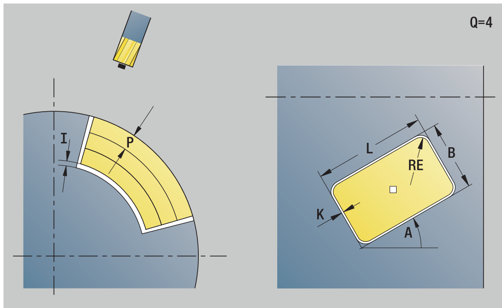
Access to the technology database:

- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G84X Pocket mllng, figures lat. surface C







The unit mills the pocket defined by **Q**. In **QK**, select the machining operation (roughing/finishing) and the plunging strategy.

Unit name: **G84x_Fig_Mant_C** / cycles: **G845**; **G846**

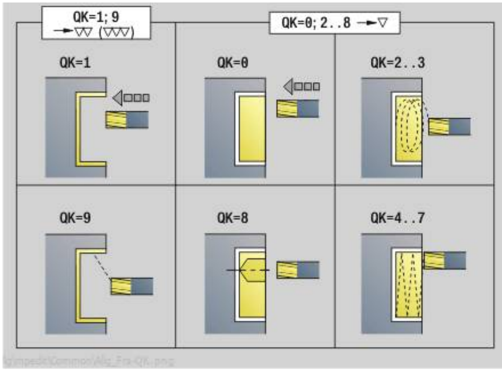
Further information: "G845 – milling", Page 474

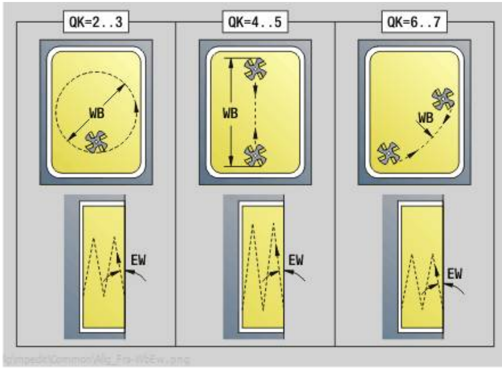
Further information: "Pocket milling - finishing G846", Page 478

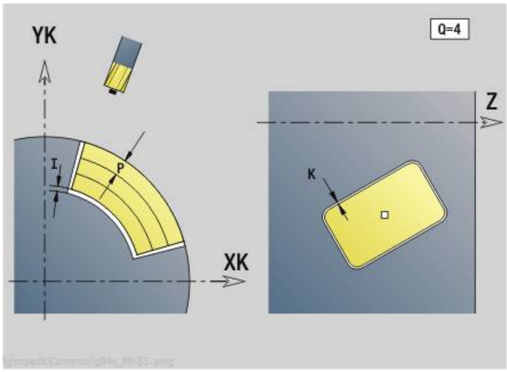
Figure form:

- **Q**: **Type of figure**
  - **0: Complete circle**
  - **1: Linear slot**
  - **2: Circular slot**
  - **3: Triangle**
  - **4: Rectangle / square**
  - **5: Polygon**
- **QN**: **No. of polygon corners** (only with **Q** = **5: Polygon**)
- **Z1**: **Figure center**
- **C1**: **Angle of figure center** (default: **Spindle angle C**)
- **CY**: **Fig. center, unr.lat. surf.**
- **X1**: **Upper edge of milling**
- **P2**: **Depth of figure**
- **L**: **+edge lngth/-width a. flats**
  - **L** > 0: **Edge length**
  - **L** < 0: **Width across** (inside diameter) for polygon
- **B**: **Width of rectangle**
- **RE**: **Rounding radius** (default: 0)
- **A**: **Angle to Z axis** (default: 0°)
- **Q2**: **Rot. direction of slot** (only with **Q** = **2: Circular slot**)
  - **cw**: In clockwise direction
  - **ccw**: In counterclockwise direction
- **W**: **Angle of slot end point** (only with **Q** = **2: Circular slot**)

> Program only the parameters relevant to the selected figure type.







Cycle form:

- **QK**: **Machining operation** and plunging strategy
  - **0: Roughing**
  - **1: Finishing**
  - **2: Helical roughing, manual**
  - **3: Helical roughing, automatic**
  - **4: Recipr. linear roughing, manual**
  - **5: Recipr. linear roughing, auto**
  - **6: Reciproc. circ. roughing, manual**
  - **7: Recipr. circular roughing, auto**
  - **8: Plunge rough at predrill pos.**
  - **9: Finishing with 3-D approach arc**
- **JT**: **Machining direction**
  - **0: From the inside out**
  - **1: From the outside in**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Maximum infeed**
- **I**: **Infeed-direction oversize**
- **K**: **Contour-parallel oversize**
- **FZ**: **Approach feed** (default: active feed rate)
- **E**: **Reduced feed**
- **R**: **Approach radius**
- **WB**: **Plunging length**
- **EW**: **Plunging angle**
- **NF**: **Position mark** (only with **QK** = 8)
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter

Global form:

- **RB**: **Return plane**

Further parameters:

Further information: "Global form", Page 103

Further forms:

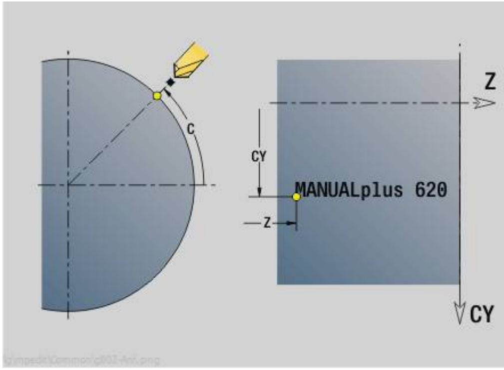
Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G802 Engraving in C on latrl. surface

The unit engraves character strings aligned linearly on the lateral surface. Diacritics and special characters that you cannot enter in **smart.Turn** operating mode can be defined, character by character, in **NF**. If you program **Q** = 1 (**Continue from last text**), then the tool change and pre-positioning are suppressed. The technological data of the previous engraving cycle apply.

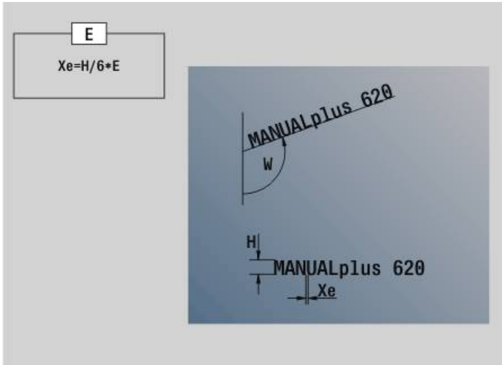


Unit name: **G802_GRA_MANT_C** / cycle: **G802**

Further information: "Engraving on lateral surface G802", Page 488

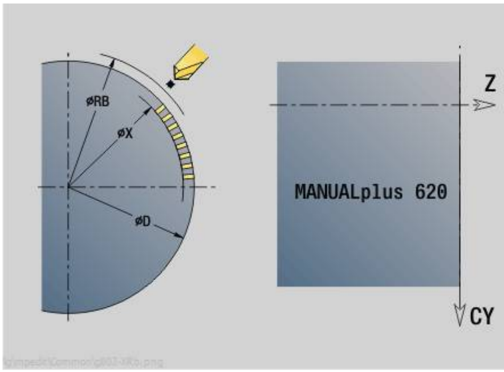
Character set:

Further information: "Character sets", Page 484



Position form:

- **Z**: **Start point**
- **C**: **Start. angle**
- **CY**: **Start point** of first character
- **X**: **Final point** – X position, infeed depth during milling (diameter value)
- **RB**: **Return plane**



Cycle form:

- **TXT**: **Text** to be engraved
- **NF**: **Char. no.** – ASCII code of the character to be engraved
- **H**: **Font height**
- **E**: **Distance factor**
  The spacing between the characters is calculated according to the following formula: **H** / 6 * **E**
- **W**: **Inclinat. ang.** of the character string
- **FZ**: **Plunging feed rate factor** (plunging feed rate = current feed rate * **FZ**)
- **D**: **Reference diameter**
- **Q**: **Continue from last text**
  - **0** (**No**): Engraving starts at the starting point
  - **1** (**Yes**): Engraving starts at the tool position
- **O**: **Mirror writing**
  - **0** (**No**): Engraving is not mirrored
  - **1** (**Yes**): Engraving is mirrored (mirror writing)
- **NS**: **Block number of contour** – reference to the contour description

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

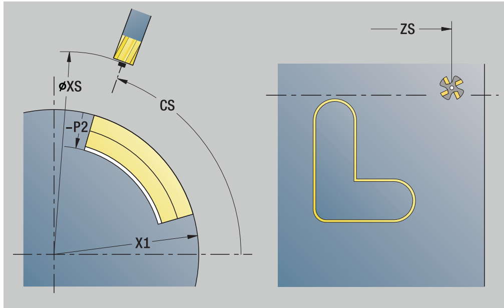
- Machining operation: **Engraving**
- Affected parameters: **F**, **S**

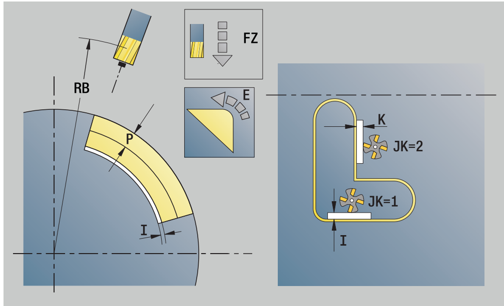
Unit G840 ICP contour milling, lat. surface C

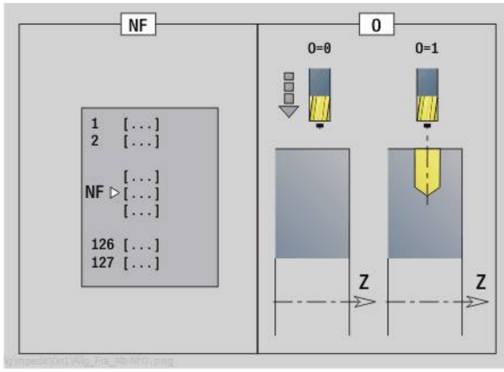
The unit mills the contour defined with **ICP** on the lateral surface.

Unit name: **G840_Kon_C_Mant** / cycle: **G840**

Further information: "G840 – milling", Page 466







Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
- **BF**: **Machine form elements** (default: 0)
  A chamfer/rounding arc is machined
  - **0: No machining**
  - **1: At beginning**
  - **2: At end**
  - **3: At beginning and end**
  - **4: Only chamfer/rounding** is machined—not the basic element (requirement: the contour section consists of a single element)
- **X1**: **Upper edge of milling**
- **P2**: **Depth of contour**

Cycle form:

- **JK**: **Cutter position**
  - **0: On the contour**
  - **1: Within/left of contour**
  - **2: Outside/right of contour**
  - **3: Depending on H and MD**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Maximum infeed**
- **I**: **Infeed-direction oversize**
- **K**: **Contour-parallel oversize**
- **FZ**: **Approach feed** (default: active feed rate)
- **E**: **Reduced feed**
- **R**: **Approach radius**
- **O**: **Plunging behavior** (default: 0)
  - **0: Straight** – the cycle moves the tool to the starting point; the tool plunges at feed rate and mills the contour
  - **1: In predrilling** – the control positions the tool above the predrill hole position; the tool plunges and mills the contour
- **NF**: **Position mark** (only with **O** = 1)
- **RB**: **Return plane**

Further forms:

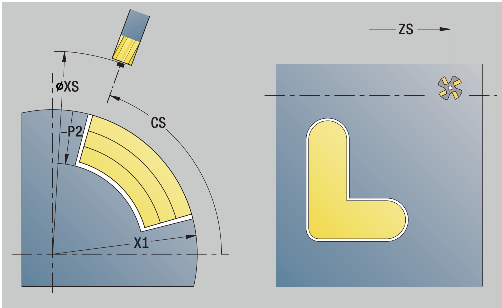
Further information: "smart.Turn unit", Page 98

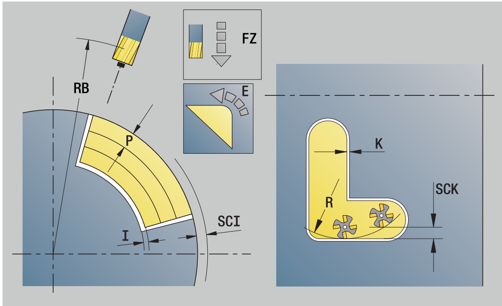
Access to the technology database:

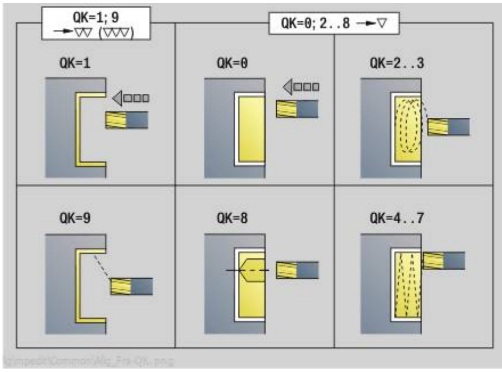
- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G845 ICP pocket milling, lat. surface C

The unit mills the pocket defined by **Q**. In **QK**, select the machining operation (roughing/finishing) and the plunging strategy.







Unit name: **G845_Tas_C_Mant** / cycles: **G845**; **G846**

Further information: "G845 – milling", Page 474

Further information: "Pocket milling - finishing G846", Page 478

Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **X1**: **Upper edge of milling**
- **P2**: **Depth of contour**
- **NF**: **Position mark** (only with **QK** = 8)
- **FZ**: **Approach feed** (default: active feed rate)
- **E**: **Reduced feed**
- **FP**: **Infeed rate in the plane**

Cycle form:

- **QK**: **Machining operation** and plunging strategy
  - **0: Roughing**
  - **1: Finishing**
  - **2: Helical roughing, manual**
  - **3: Helical roughing, automatic**
  - **4: Recipr. linear roughing, manual**
  - **5: Recipr. linear roughing, auto**
  - **6: Reciproc. circ. roughing, manual**
  - **7: Recipr. circular roughing, auto**
  - **8: Plunge rough at predrill pos.**
  - **9: Finishing with 3-D approach arc**
- **JT**: **Machining direction**
  - **0: From the inside out**
  - **1: From the outside in**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Maximum infeed**
- **I**: **Infeed-direction oversize**
- **K**: **Contour-parallel oversize**
- **R**: **Approach radius**
- **WB**: **Plunging length**
- **EW**: **Plunging angle**
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
- **RB**: **Return plane**

Further forms:

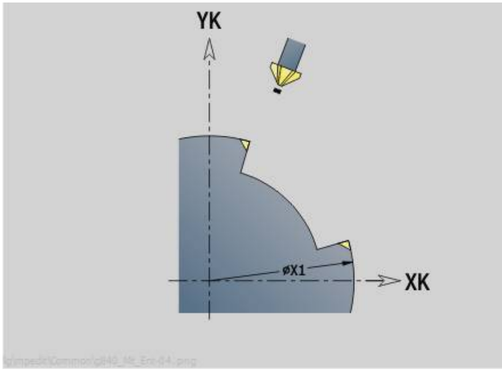
Further information: "smart.Turn unit", Page 98

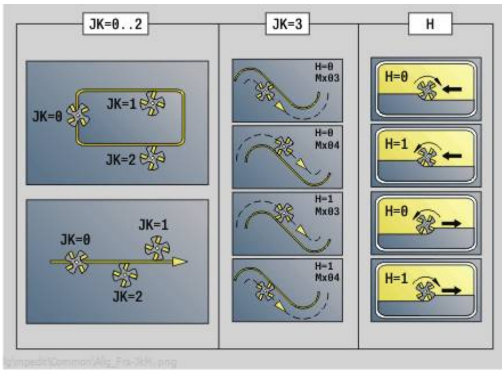
Access to the technology database:

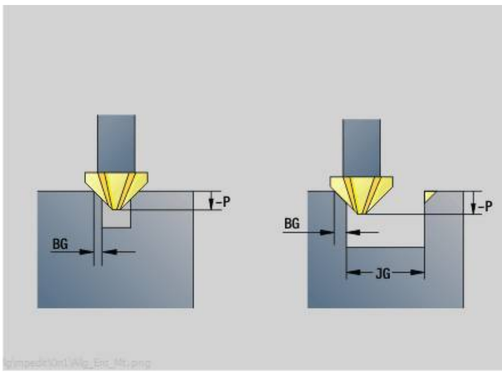
- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G840 ICP deburring, lat. surface C

The unit deburrs the contour defined with **ICP** on the lateral surface.







Unit name: **G840_ENT_C_MANT** / cycle: **G840**

Further information: "G840 – deburring", Page 470

Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
- **BF**: **Machine form elements** (default: 0)

  A chamfer/rounding arc is machined

  - **0: No machining**
  - **1: At beginning**
  - **2: At end**
  - **3: At beginning and end**
  - **4: Only chamfer/rounding** is machined—not the basic element (requirement: the contour section consists of a single element)
- **X1**: **Upper edge of milling**

Cycle form:

- **JK**: **Cutter position**
  - **0: On the contour**
  - **1: Within/left of contour**
  - **2: Outside/right of contour**
  - **3: Depending on H and MD**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **BG**: **Cham. width** for deburring
- **JG**: **Preparation diameter**
- **P**: **Plunging depth** (indicated as a negative value)
- **K**: **Contour-parallel oversize**
- **R**: **Approach radius**
- **FZ**: **Approach feed** (default: active feed rate)
- **E**: **Reduced feed**
- **RB**: **Return plane**

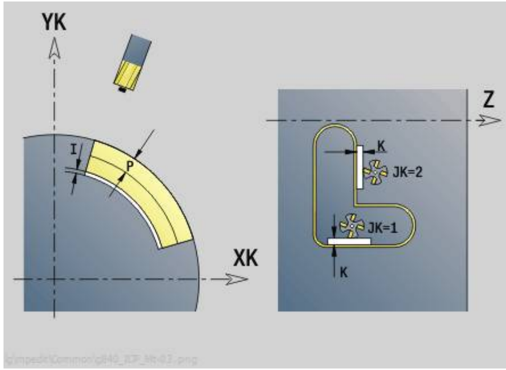
Further forms:

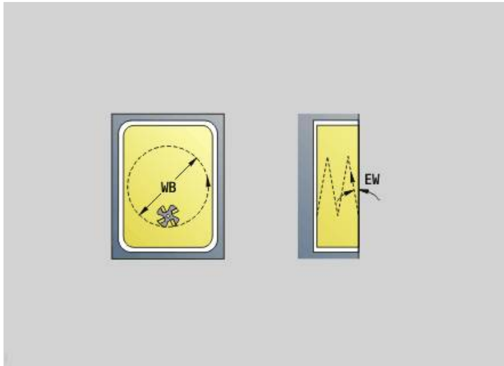
Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **Deburring**
- Affected parameters: **F**, **S**

Unit G847 ICP Troch. contour mllng, lat. surf. C





The unit uses trochoidal milling to rough out the open or closed contour on the lateral surface of the workpiece defined with **ICP**.

Unit name: **G847_KON_C_MANT** / cycle: **G847**

Further information: "Trochoidal contour milling G847 ", Page 480

Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
- **BF**: **Machine form elements** (default: 0)

  A chamfer/rounding arc is machined
  - **0: No machining**
  - **1: At beginning**
  - **2: At end**
  - **3: At beginning and end**
  - **4: Only chamfer/rounding** is machined—not the basic element (requirement: the contour section consists of a single element)
- **X1**: **Upper edge of milling** (diameter value; default: **Starting point X**)
- **P2**: **Depth of contour**
- **I**: **Infeed-direction oversize**
- **K**: **Contour-parallel oversize**
- **RB**: **Return plane** (default: back to start position)
- **NF**: **Position mark** (only with **O** = 1)

Cycle form:

- **JK**: **Cutter position**
  - **0: On the contour**
  - **1: Within/left of contour**
  - **2: Outside/right of contour**
- **H**: **Mill cutting direction** (default: 1)
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Maximum infeed**
- **BR**: **Trochoid width**
- **R**: **Radius for return**
- **FP**: **Feed rate for return** (default: active feed rate)
- **AL**: **Retraction path for return**

- **O**: **Plunging behavior** (default: 2)
  - **O** = 0 (vertical plunging): The cycle moves the tool to the starting point; the tool plunges at the feed rate for infeed and mills the contour
  - **O** = 1 (vertical plunging, e.g. at the predrilled position):
    - If **NF** is programmed: The control positions the milling cutter above the first predrill hole position; the tool plunges at rapid traverse to safety clearance and mills the first area. If applicable, the cycle positions the tool to the next predrill hole position and mills the next area etc.
    - If **NF** is not programmed: The cycle plunges at the current position at rapid traverse and mills the area. If applicable, position the tool to the next predrill hole position and mill the next area etc.
  - **O** = 2 (plunge in helical motion): The tool plunges at the current position at the angle **W** and mills full circles with the diameter **WB**.
- **FZ**: **Approach feed** (default: active feed rate)
- **EW**: **Plunging angle**
- **WB**: **Diameter of the helix** (default: helix diameter = 1.5 * milling diameter)
- **U**: **Overlap factor** – overlap of milling paths = **U** * milling diameter (default: 0.9)
- **HCC**: **Contour smoothing**
  - **0: No smoothing cut**
  - **1: With smoothing cut**

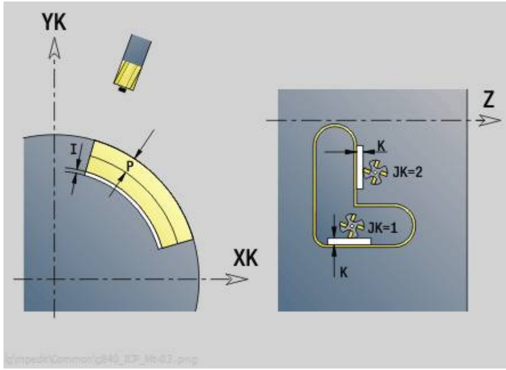
Further forms:

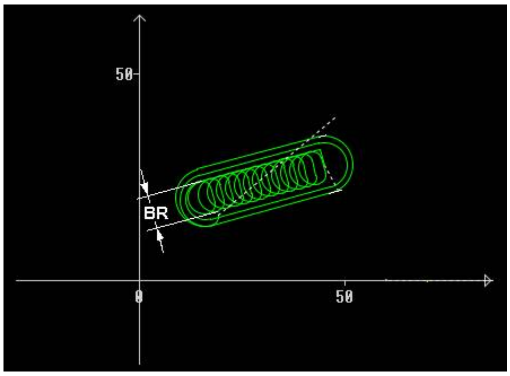
Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G848 ICP Troch. pocket mllng, lat. surf. C





The unit uses trochoidal milling to rough out the figure or figure pattern defined on the lateral surface with **ICP**.

Unit name: **G848_TAS_C_MANT** / cycle: **G848**

Further information: "Trochoidal pocket milling G847 ", Page 482

Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **X1**: **Upper edge of milling**
- **P2**: **Depth of contour**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **RB**: **Return plane** (default: back to start position)
- **NF**: **Position mark** (only with **O** = 1)

Cycle form:

- **H**: **Mill cutting direction** (default: 1)
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Maximum infeed**
- **BR**: **Trochoid width**
- **R**: **Radius for return**
- **FP**: **Feed rate for return** (default: active feed rate)
- **AL**: **Retraction path for return**
- **O**: **Plunging behavior** (default: 2)
  - **O**=0 (vertical plunging): The cycle moves the tool to the starting point; the tool plunges at the feed rate for infeed and mills the figure
  - **O** = 1 (vertical plunging, e.g. at the predrilled position):
    - If **NF** is programmed: The control positions the milling cutter above the first predrill hole position; the tool plunges at rapid traverse to safety clearance and mills the first area. If applicable, the cycle positions the tool to the next predrill hole position and mills the next area etc.
    - If **NF** is not programmed: The cycle plunges at the current position at rapid traverse and mills the area. If applicable, position the tool to the next predrill hole position and mill the next area etc.
  - **O** = 2 (plunge in helical motion): The tool plunges at the current position at the angle **W** and mills full circles with the diameter **WB**.
- **FZ**: **Approach feed** (default: active feed rate)
- **EW**: **Plunging angle**
- **WB**: **Diameter of the helix** (default: helix diameter = 1.5 * milling diameter)
- **U**: **Overlap factor** – overlap of milling paths = **U** * milling diameter (default: 0.9)

- **J**: **Machining operation**
  - **0: Complete**
  - **1: W/o corner machining**
  - **2: Only corner machining**

> For slots and rectangles, you need to program width **BR** of the trochoidal tool path; for circles and polygons, this is not necessary.

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

4.11 Units – Spec (special operations)

Unit Program beginning (START)

Refer to your machine manual.

The machine tool builder can provide you with a machine-dependent start unit containing the definitions of various transfer parameters that automatically take into account a bar feeder, for example.

In the start unit, you define default values that are used by the control in the following units. The start unit is called once at the beginning of the machining section. You can also directly specify **Max. shaft speed**, **Datum shift**, and the **Tool change point** for this NC program.

Unit name: **Start** / Called cycle: None

Limits form:

- **S0**: **Maximum speed** for main spindle
- **S1**: **Maximum speed** for driven tool
- **Z**: **Datum shift G59**

TC point form (tool change point):

- **WT1**: **Tool change point**
  - **No axis** (do not approach the tool change point)
  - **0: Simultaneously**
  - **1: First X, then Z**
  - **2: First Z, then X**
  - **3: Only X**
  - **4: Only Z**
  - **5: Only Y**
  - **6: Simultaneous w/ Y**
- **WX1**: **Tool change point** in X (reference: distance between slide position and machine datum as radius value)
- **WY1**: **Tool change point** in Y (reference: distance between slide position and machine datum)
- **WZ1**: **Tool change point** in Z (reference: distance between slide position and machine datum)

Defaults form:

- **GWW**: **Tool change point**
  - **No axis** (do not approach the tool change point)
  - **0: Simultaneously** X and Z axis depart diagonally
  - **1: First X, then Z**
  - **2: First Z, then X**
  - **3: Only X**
  - **4: Only Z**
  - **5: Only Y**
  - **6: Simultaneous w/ Y**

- **CLT**: **Coolant**
  - **0: Without**
  - **1: Coolant 1 on**
  - **2: Coolant 2 on**
- **G60**: Deactivate **Protection zone** for the drilling operation
  - **0: Active**
  - **1: Inactive**

Cycle form:

- **L**: **Subprogram - Name** – name of a subprogram that is called by the start unit

Global form:

- **G47**: **Safety clearance**
- **SCK**: **Safety clearance** in infeed direction during drilling and milling operations
- **SCI**: **Safety clearance** in the machining plane during drilling and milling operations
- **I**, **K**: **O-size X** and **Z**

> - You can load the datum shift and the tool change point by soft key
> - The setting in the **TC point** form applies only within the current program
> - Position of tool change point (**WX1**, **WZ1**, **WY1**):
>   - If the tool change point is defined, you use **G14** to move to this position
>   - If the tool change point is not defined, you use **G14** to move to the position defined in manual mode
> - If you call a subprogram using the start unit, you should set the subprogram with **G65** Chuck selection with fixture **D0**. You should also move the C axes out, for example with **M15** or **M315**

Soft keys in the program beginning form	
Acceptance of datum	Loads the datum defined during setup
Acceptance TC point $1	Loads the tool-change point defined during setup

Unit C axis ON (option 9)

The unit activates the **SPI** (spindle) C axis.

Unit name: **C_Axis_ON** / Called cycle: None

C axis ON form:

- **SPI**: **Workpc spindle no. 0..3** – spindle that is holding the workpiece
- **C**: **Approach position C**

Unit C axis OFF (option 9)

The unit deactivates the **SPI** (spindle) C axis.

Unit name: **C_Axis_OFF** / Called cycle: None

C axis OFF form:

- **SPI**: **Workpc spindle no. 0..3** – spindle that is holding the workpiece

Unit Subprogram call

The unit calls the subprogram defined in **L**.

Unit name: **SUBPROG** / Called cycle: Any subprogram

Contour form:

- **L**: **Subprogram - Name**
- **Q**: **Number of repetitions** (default: 1)
- **LA**-**LF**: **Transmiss.value**
- **LH**: **Transmiss.value**
- **LN**: **Transmiss.value** – reference to a block number as contour reference
  Is updated during block numbering.

Cycle form:

- **LI**-**LK**: **Transmiss.value**
- **LO**: **Transmiss.value**
- **LP**: **Transmiss.value**
- **LR**: **Transmiss.value**
- **LS**: **Transmiss.value**
- **LU**: **Transmiss.value**
- **LW**-**LZ**: **Transmiss.value**

Cycle form:

- **ID1**: **Transmiss.value** – text variable (string)
- **AT1**: **Transmiss.value** – text variable (string)
- **BS**: **Transmiss.value**
- **BE**: **Transmiss.value**
- **WS**: **Transmiss.value**
- **AC**: **Transmiss.value**
- **WC**: **Transmiss.value**
- **RC**: **Transmiss.value**
- **IC**: **Transmiss.value**
- **KC**: **Transmiss.value**
- **JC**: **Transmiss.value**

Accessing the technology database is not possible.

- The tool call is not a mandatory parameter in this unit
- Instead of the text **transfer value,** texts can be displayed that were defined in the subprogram. You can also define help graphics for each line of the subprogram
  Further information: "Subprograms", Page 542

Unit Process logic / Repetition – program section repeat

Use the **Repeat** unit to program a program section repeat. The unit consists of two inseparable parts. Program the unit with the **Start** form immediately before the repeatable part, and the unit with the **End** form immediately after the repeatable part. Be sure to use the same variable number here.

Unit name: **REPEAT** / Called cycle: None

Start form:

- **AE**: **Repetition**
  - **0: Beginning**
  - **1: End**
- **V**: **Variable number 1-30** – counting variable for the iteration loop
- **NN**: **Number of repetitions**
- **QR**: **Save workpiece blank**
  - **0: No**
  - **1: Yes**
- **K**: **Comment**

End form:

- **AE**: **Repetition**
  - **0: Beginning**
  - **1: End**
- **V**: **Variable number 1-30** – counting variable for the iteration loop
- **Z**: **Additive datum shift**
- **C**: **C-axis incremental shift**
- **Q**: **No. axis C**
- **K**: **Comment**

Unit Program end (END)

In every smart.Turn program, the end unit should be called once at the end of the machining section.

Unit name: **END** / Called cycle: None

Program end form:

- **ME**: **Type of return jump:**
  - **30: Without restart M30**
  - **99: With restart M99**
- **NS**: **Block no. for return jump**
- **G14**: **Tool change point**
  - **No axis**
  - **0: Simultaneously**
  - **1: First X, then Z**
  - **2: First Z, then X**
  - **3: Only X**
  - **4: Only Z**
  - **5: Only Y** (machine-dependent)
  - **6: Simultaneous w/ Y** (machine-dependent)
- **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

Unit Tilt plane

The unit executes the following transformations and rotations:

- Shifts the coordinate system to the position **I**, **K**
- Rotates the coordinate system by the **Angle B**; reference: **I**, **K**
- Shifts, if programmed, the coordinate system by **U** and **W** in the rotated coordinate system

Unit name: **G16_ROTWORKPLAN** / Called cycle: **G16**

Further information: "Tilt working plane G16", Page 648

Tilt plane form:

- **Q**: **Tilt plane**
  - **0: OFF** (deactivate tilting)
  - **1: ON** (tilt working plane)
- **B**: **Angle** – plane angle (reference: positive Z axis)
- **I**: **Reference pt.** – plane reference in X direction (radius value)
- **K**: **Reference pt.** – plane reference (in Z)
- **U**: **Shift in X** (radius value)
- **W**: **Shift in Z**

Please note:

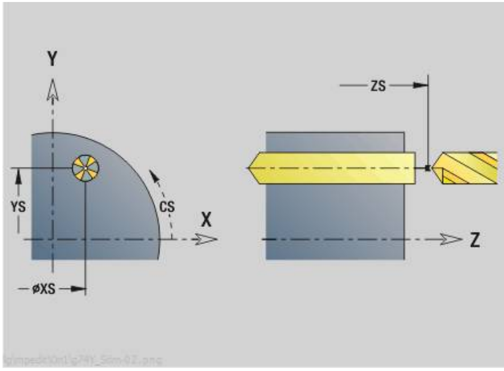
- **Q0** resets the working plane. The datum and coordinate system defined before the unit are then in effect again
- The positive Z axis is the reference axis for the **Angle B**. This also applies in a mirrored coordinate system
- X is the infeed axis in a tilted coordinate system. X coordinates are entered as diameter coordinates.
- Other datum shifts are not permitted as long as tilting is active

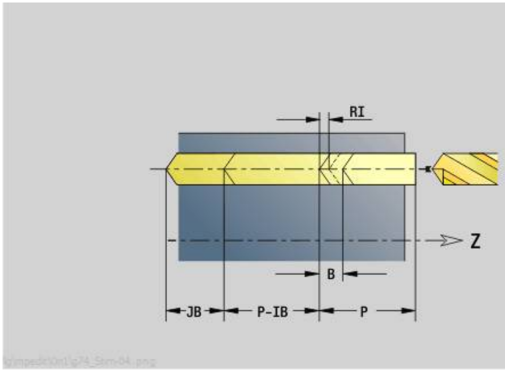
5

smart.Turn Units for the Y Axis (Option 9 and Option 70)

5.1 Units – Drilling / ICP Y

Unit G74 Drilling in ICP Y





The unit machines a single hole or a hole pattern in the XY or YZ plane. Using **ICP**, you define the holes as well as further details.

Unit name: **G74_ICP_Y** / cycle: **G74**

Further information: "Deep boring G74", Page 428

Pattern form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section

Cycle form:

- **E**: **Period of dwell** at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **DFF**: **Retraction feed rate**
- **V**: **Feed reduction**
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **AB**: **Drilling lengths** (default: 0)
- **P**: **1. Boring depth**
- **IB**: **Hole depth reduct. val** (default: 0)
- **JB**: **Minimum hole depth**
  If you have entered a hole depth reduction value, the hole depth is reduced only to the value entered in **JB**.
- **B**: **Return distance** – value by which the tool is retracted after reaching the respective hole depth
- **RI**: **Safety clearance** internal – distance for reapproach inside the hole (default: **Safety clearance SCK**)
- **RB**: **Return plane** (default: back to start position)

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **Drilling**
- Affected parameters: **F**, **S**

Unit G73 Tapping in ICP Y

The unit machines a single tap hole or a hole pattern in the XY or YZ plane. Using **ICP**, you define the tap holes as well as further details.

Unit name: **G73_ICP_Y** / cycle: **G73**

Further information: "Tapping G73", Page 426

Pattern form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section

Cycle form:

- **F1**: **Thread pitch**
- **B**: **Run-in lgth** to obtain the programmed spindle speed and feed rate (default: 2 * **Thread pitch F1**)
- **L**: **Retract length** when using floating tap holders (default: 0)
- **SR**: **Return speed** (default: tapping speed)
- **SP**: **Chip breaking depth**
- **SI**: **Retraction distance**
- **RB**: **Return plane**

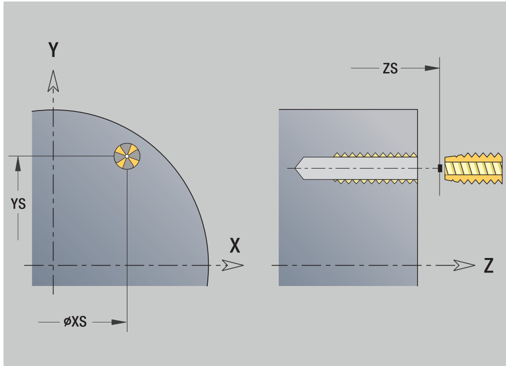
Further forms:

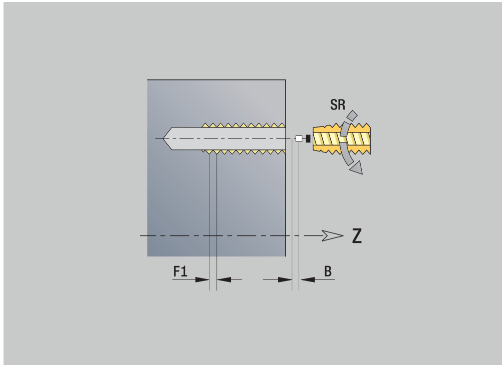
Further information: "smart.Turn unit", Page 98

Use the **Retract length** 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 tap is pulled away from the chuck by the retraction length. With this method, you can achieve higher service life of the taps.

Access to the technology database:

- Machining operation: **Tapping**
- Affected parameter: **S**





> If you interrupt program run during a tapping cycle, you can manually retract the tool from the hole in the Z axis. The control moves the spindle on a path matching the traverse.
>
> If the optional machine parameter **CfgBackTrack** (no. 122000) is active, use the **Start blck search** soft key to resume program run after the manual traverse.

Unit G72 Boring, cntrsinkg ICP Y

The unit machines a single hole or a hole pattern in the XY or YZ plane. Using **ICP**, you define the hole positions as well as further details for boring or countersinking.

Unit name: **G72_ICP_Y** / cycle: **G72**

Further information: "Boring/cnt-sink G72", Page 425

Pattern form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section

Cycle form:

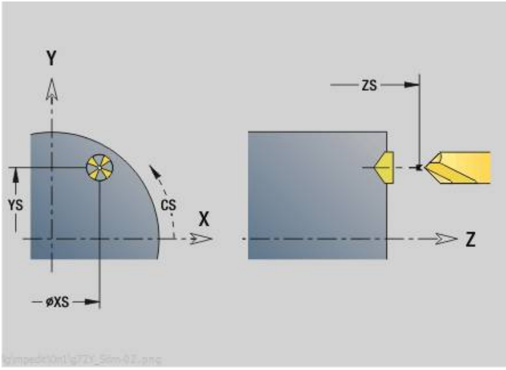
- **E**: **Period of dwell** at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **DFF**: **Retraction feed rate**
- **RB**: **Return plane** (default: back to start position)

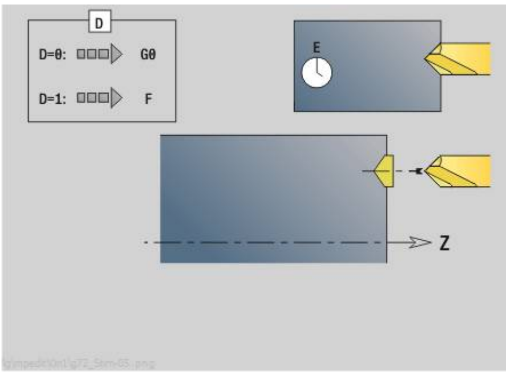
Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **Drilling**
- Affected parameters: **F**, **S**

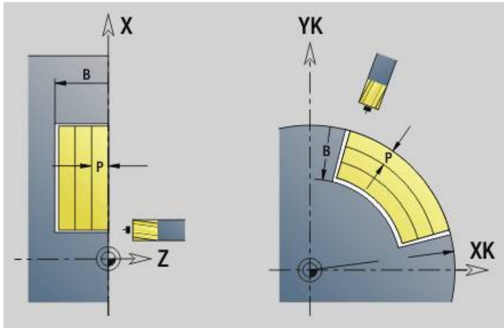




Units G75 Bore milling Y

Unit G75 Bore milling ICP Y face

The unit machines a single hole or a hole pattern on the face. Using **ICP**, you define the positions of the holes as well as further details.



> Only the contour description (ICP) for the C axis or Y axis is used for bore milling.

Unit name: **G75_BF_ICP_Y** / cycle: **G75**

Further information: "Bore milling G75", Page 431

Contour form:

- **FK**: **Finished part contour** – name of the contour to be machined
- **NS**: **Starting block no. of contour** – beginning of contour section
- **FZ**: **Approach feed** (default: active feed rate)
- **B**: **Milling depth** (default: depth from the contour description)

Cycle form:

- **QK**: **Machining operation**
  - **0: Roughing**
  - **1: Finishing**
  - **2: Roughing and finishing**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Max. approach** (default: milling in one infeed)
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **WB**: **Diameter of the helix** (default: helix diameter = 1.5 * milling diameter)
- **EW**: **Plunging angle**
- **U**: **Overlap factor** – overlap of milling paths = **U** * milling diameter (default: 0.5)
- **RB**: **Return plane** (default: retract to starting position or to safety clearance; diameter value with radial holes and holes in the YZ plane)

Further forms:

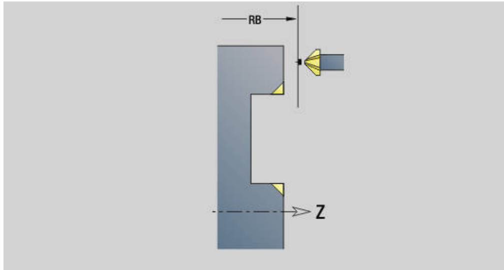
Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G75 Deburring ICP Y face

The unit deburs a single hole or a hole pattern on the face. Using **ICP**, you define the positions of the holes as well as further details.



> Only the contour description (ICP) for the C axis or Y axis is used for bore milling.

Unit name: **G75_EN_ICP_Y** / cycle: **G75**

Further information: "Bore milling G75", Page 431

Contour form:

- **FK**: **Finished part contour** – name of the contour to be machined
- **NS**: **Starting block no. of contour** – beginning of contour section
- **B**: **Milling depth** (default: countersinking depth from the contour definition)

Cycle form:

- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **RB**: **Return plane** (default: retract to starting position or to safety clearance; diameter value with radial holes and holes in the YZ plane)

Further forms:

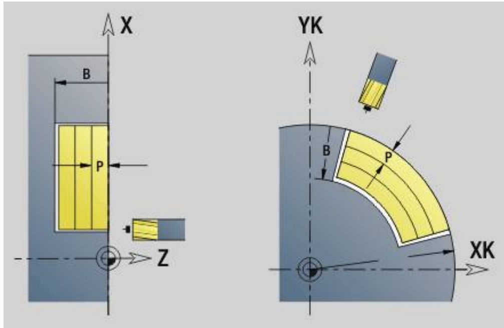
Further information: "smart.Turn unit", Page 98

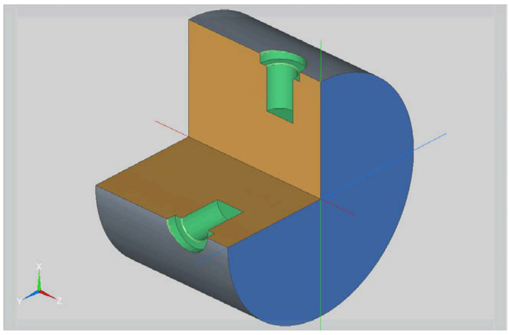
Access to the technology database:

- Machining operation: **Deburring**
- Affected parameters: **F**, **S**

Unit G75 Bore milling ICP Y lateral

The unit machines a single hole or a hole pattern on the lateral surface. Using **ICP**, you define the positions of the holes as well as further details.





> Only the contour description (ICP) for the C axis or Y axis is used for bore milling.

Unit name: **G75_BF_ICP_Y_MANT** / cycle: **G75**

Further information: "Bore milling G75", Page 431

Contour form:

- **FK**: **Finished part contour** – name of the contour to be machined
- **NS**: **Starting block no. of contour** – beginning of contour section
- **FZ**: **Approach feed** (default: active feed rate)
- **B**: **Milling depth** (default: depth from the contour description)

Cycle form:

- **QK**: **Machining operation**
  - **0: Roughing**
  - **1: Finishing**
  - **2: Roughing and finishing**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Max. approach** (default: milling in one infeed)
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **WB**: **Diameter of the helix** (default: helix diameter = 1.5 * milling diameter)
- **EW**: **Plunging angle**
- **U**: **Overlap factor** – overlap of milling paths = **U** * milling diameter (default: 0.5)
- **RB**: **Return plane** (default: retract to starting position or to safety clearance; diameter value with radial holes and holes in the YZ plane)

Further forms:

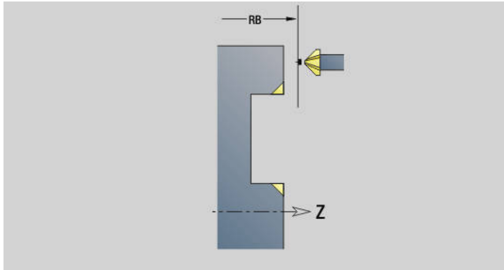
Further information: "smart.Turn unit", Page 98

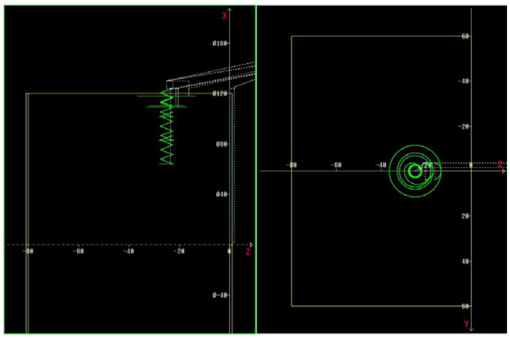
Access to the technology database:

- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G75 Deburring ICP Y lateral

The unit deburs a single hole or a hole pattern on the lateral surface. Using **ICP**, you define the positions of the holes as well as further details.





> Only the contour description (ICP) for the C axis or Y axis is used for bore milling.

Unit name: **G75_EN_ICP_Y_MANT** / cycle: **G75**

Further information: "Bore milling G75", Page 431

Contour form:

- **FK**: **Finished part contour** – name of the contour to be machined
- **NS**: **Starting block no. of contour** – beginning of contour section
- **B**: **Milling depth** (default: countersinking depth from the contour definition)

Cycle form:

- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **RB**: **Return plane** (default: retract to starting position or to safety clearance; diameter value with radial holes and holes in the YZ plane)

Further forms:

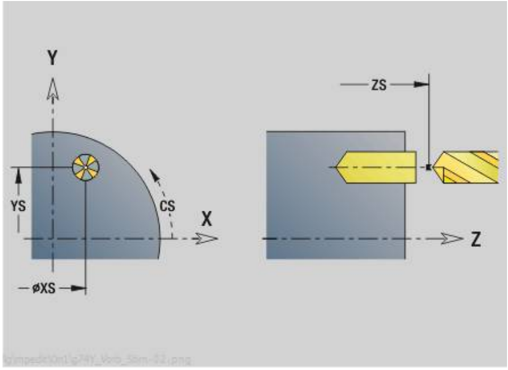
Further information: "smart.Turn unit", Page 98

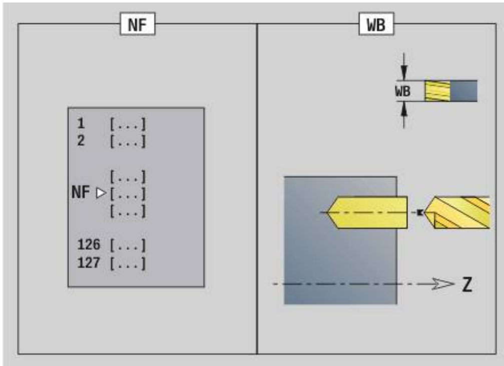
Access to the technology database:

- Machining operation: **Deburring**
- Affected parameters: **F**, **S**

5.2 Units – Drilling / Predrilling, milling in Y

Unit G840 Predrill, contour mill, ICP on face in Y





The unit determines the hole position and machines the hole. The subsequent milling cycle obtains the hole position from the reference stored in **NF**. If the milling contour consists of multiple sections, the unit machines a hole for each section.

Unit name: **DRILL_STI_840_Y** / cycles: **G840 A1**; **G71**

Further information: "G840 – calculating hole positions", Page 464

Further information: "Simple drilling G71", Page 423

Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
- **Z1**: **Upper edge of milling**
- **P2**: **Depth of contour**

Cycle form:

- **JK**: **Cutter position**
  - **0: On the contour**
  - **1: Within/left of contour**
  - **2: Outside/right of contour**
  - **3: Depending on H and MD**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **R**: **Approach radius**
- **WB**: **Milling diameter**
- **NF**: **Position mark** – reference at which the cycle stores the predrill hole positions (range: 1 to 127)
- **E**: **Period of dwell** at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **V**: **Feed reduction**
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **AB**: **Drilling lengths** (default: 0)
- **RB**: **Return plane** (default: back to start position)

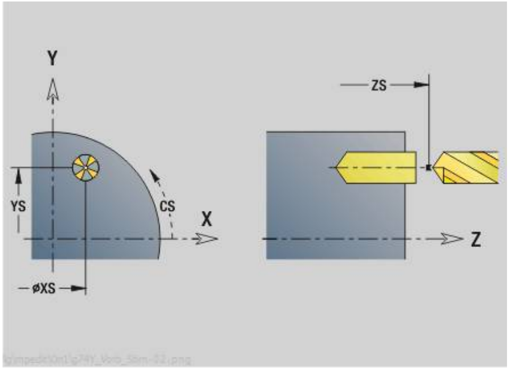
Further forms:

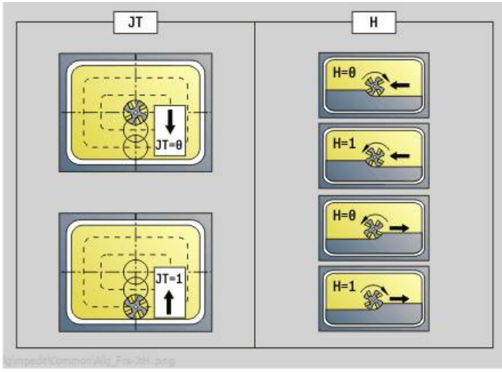
Further information: "smart.Turn unit", Page 98

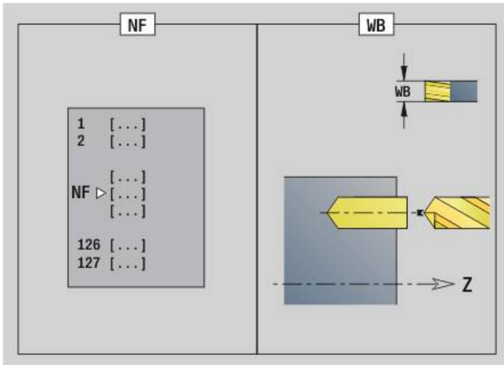
Access to the technology database:

- Machining operation: **Drilling**
- Affected parameters: **F**, **S**

Unit G845 Predrill, pocket mill, ICP on face in Y







The unit determines the hole position and machines the hole. The subsequent milling cycle obtains the hole position from the reference stored in **NF**. If the pocket consists of multiple sections, the unit machines a hole for each section.

Unit name: **DRILL_STI_845_Y** / Cycles: **G845**; **G71**

Overvw. form:

- **AP**: **Predrilling position**
  - **1: Determine predrilling pos**
  - **2: predrill. pos. fig. center**

Further information: "G845 – calculating hole positions", Page 473

Further information: "Simple drilling G71", Page 423

Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
- **Z1**: **Upper edge of milling**
- **P2**: **Depth of contour**

Cycle form:

- **JT**: **Machining direction**
  - **0: From the inside out**
  - **1: From the outside in**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
- **WB**: **Milling diameter**
- **NF**: **Position mark** – reference at which the cycle stores the predrill hole positions (range: 1 to 127)
- **E**: **Period of dwell** at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **V**: **Feed reduction**
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **AB**: **Drilling lengths** (default: 0)
- **RB**: **Return plane** (default: back to start position)

Further forms:

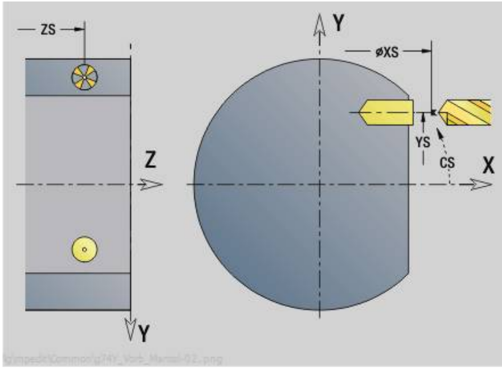
Further information: "smart.Turn unit", Page 98

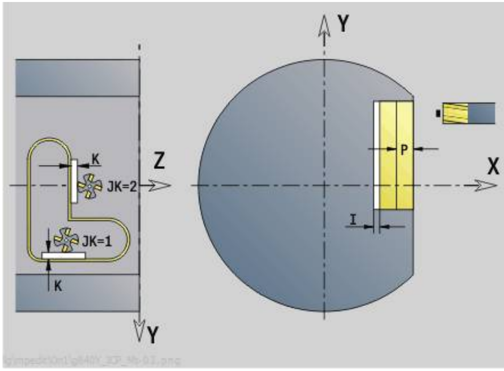
Access to the technology database:

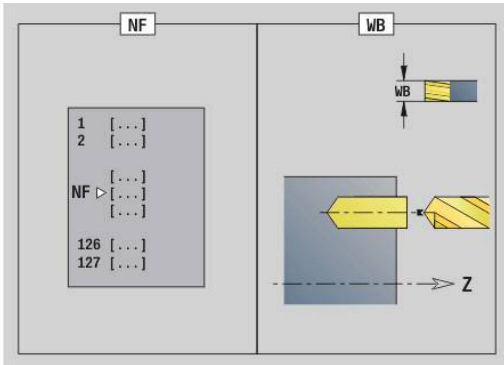
- Machining operation: **Drilling**
- Affected parameters: **F**, **S**

Unit G840 Predrill, contour mill, ICP on lateral in Y

The unit determines the hole position and machines the hole. The subsequent milling cycle obtains the hole position from the reference stored in **NF**. If the milling contour consists of multiple sections, the unit machines a hole for each section.







Unit name: **DRILL_MAN_840_Y** / cycles: **G840 A1**; **G71**

Further information: "G840 – calculating hole positions", Page 464

Further information: "Simple drilling G71", Page 423

Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
- **X1**: **Upper edge of milling**
- **P2**: **Depth of contour**

Cycle form:

- **JK**: **Cutter position**
  - **0: On the contour**
  - **1: Within/left of contour**
  - **2: Outside/right of contour**
  - **3: Depending on H and MD**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **R**: **Approach radius**
- **WB**: **Milling diameter**
- **NF**: **Position mark** – reference at which the cycle stores the predrill hole positions (range: 1 to 127)
- **E**: **Period of dwell** at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **V**: **Feed reduction**
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **AB**: **Drilling lengths** (default: 0)
- **RB**: **Return plane** (default: back to start position)

Further forms:

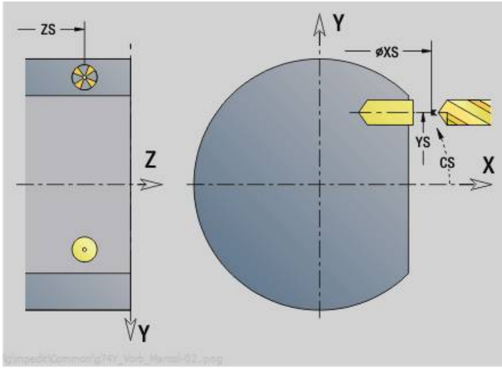
Further information: "smart.Turn unit", Page 98

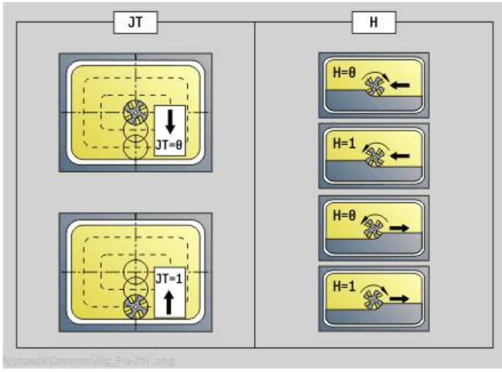
Access to the technology database:

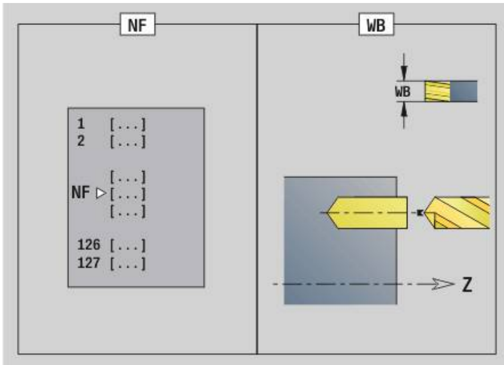
- Machining operation: **Drilling**
- Affected parameters: **F**, **S**

Unit G845 Predrill, pocket mill, ICP on lateral in Y

The unit determines the hole position and machines the hole. The subsequent milling cycle obtains the hole position from the reference stored in **NF**. If the pocket consists of multiple sections, the unit machines a hole for each section.







Unit name: **DRILL_MAN_845_Y** / Cycles: **G845**

Overvw. form:

- **AP**: **Predrilling position**
  - **1: Determine predrilling pos**
  - **2: predrill. pos. fig. center**

Further information: "G845 – calculating hole positions", Page 473

Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
- **X1**: **Upper edge of milling**
- **P2**: **Depth of contour**

Cycle form:

- **JT**: **Machining direction**
  - **0: From the inside out**
  - **1: From the outside in**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
- **WB**: **Milling diameter**
- **NF**: **Position mark** – reference at which the cycle stores the predrill hole positions (range: 1 to 127)
- **E**: **Period of dwell** at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **V**: **Feed reduction**
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **AB**: **Drilling lengths** (default: 0)
- **RB**: **Return plane** (default: back to start position)

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **Drilling**
- Affected parameters: **F**, **S**

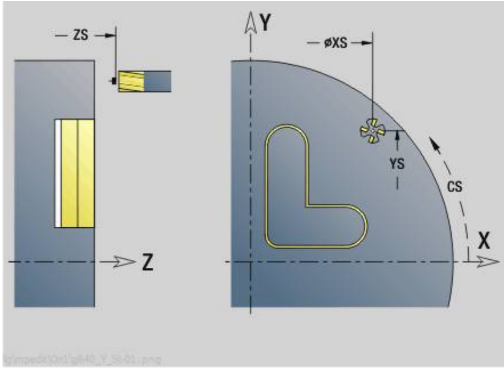
5.3 Units – Millg / Y axis, face, Y axis, latrl.

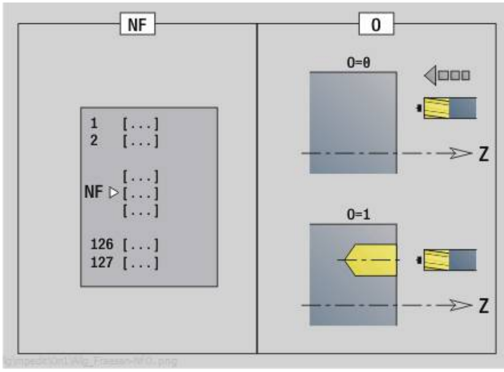
Unit G840 ICP contour milling on face in Y

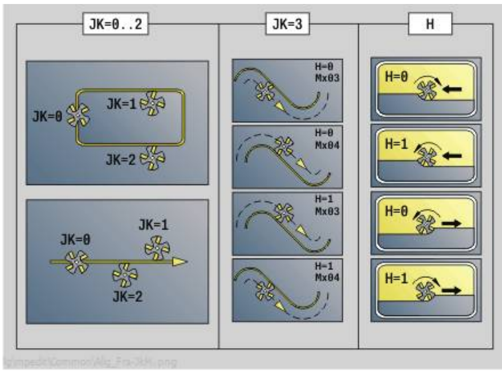
The unit mills the contour defined with **ICP** in the XY plane.

Unit name: **G840_Kon_Y_Stirn** / cycle: **G840**

Further information: "G840 – milling", Page 466







Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
- **BF**: **Machine form elements** (default: 0)

  A chamfer/rounding arc is machined
  - **0: No machining**
  - **1: At beginning**
  - **2: At end**
  - **3: At beginning and end**
  - **4: Only chamfer/rounding** is machined—not the basic element (requirement: the contour section consists of a single element)
- **Z1**: **Upper edge of milling**
- **P2**: **Depth of contour**

Cycle form:

- **JK**: **Cutter position**
  - **0: On the contour**
  - **1: Within/left of contour**
  - **2: Outside/right of contour**
  - **3: Depending on H and MD**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Maximum infeed**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **FZ**: **Approach feed** (default: active feed rate)
- **E**: **Reduced feed**
- **R**: **Approach radius**
- **O**: **Plunging behavior** (default: 0)
  - **0: Straight** – the cycle moves the tool to the starting point; the tool plunges at feed rate and mills the contour
  - **1: In predrilling** – the control positions the tool above the predrill hole position; the tool plunges and mills the contour
- **NF**: **Position mark** (only with **O** = 1)
- **RB**: **Return plane** (default: back to start position)

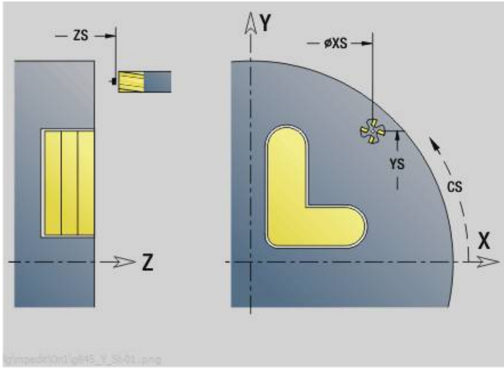
Further forms:

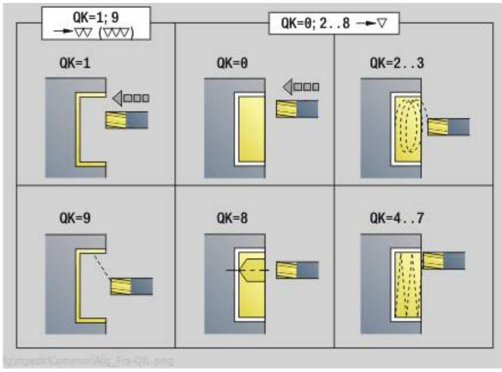
Further information: "smart.Turn unit", Page 98

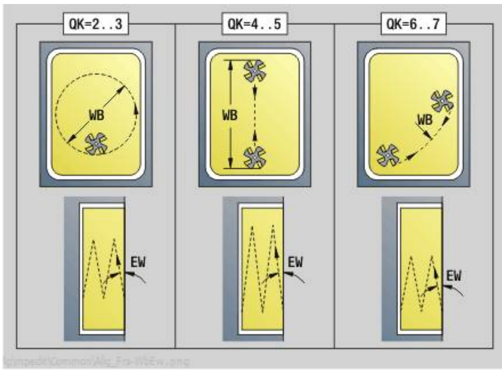
Access to the technology database:

- Machining operation: Finish-milling
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G845 ICP pocket milling on face in Y







The unit mills the pocket defined with **ICP** in the XY plane. In **QK** (machining operation), select whether a roughing or finishing operation is to be executed. For roughing, define the plunging strategy.

Unit name: **G845_Tas_Y_Stirn** / cycles: **G845**; **G846**

Further information: "G845 – milling", Page 474

Further information: "Pocket milling - finishing G846", Page 478

Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **Z1**: **Upper edge of milling**
- **P2**: **Depth of contour**
- **NF**: **Position mark** (only with **QK** = 8)
- **FZ**: **Approach feed** (default: active feed rate)
- **E**: **Reduced feed**
- **FP**: **Infeed rate in the plane**

Cycle form:

- **QK**: **Machining operation** and plunging strategy
  - **0: Roughing**
  - **1: Finishing**
  - **2: Helical roughing, manual**
  - **3: Helical roughing, automatic**
  - **4: Recipr. linear roughing, manual**
  - **5: Recipr. linear roughing, auto**
  - **6: Reciproc. circ. roughing, manual**
  - **7: Recipr. circular roughing, auto**
  - **8: Plunge rough at predrill pos.**
  - **9: Finishing with 3-D approach arc**
- **JT**: **Machining direction**
  - **0: From the inside out**
  - **1: From the outside in**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Maximum infeed**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **R**: **Approach radius**
- **WB**: **Plunging length**
- **EW**: **Plunging angle**

- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
- **RB**: **Return plane** (default: back to start position)

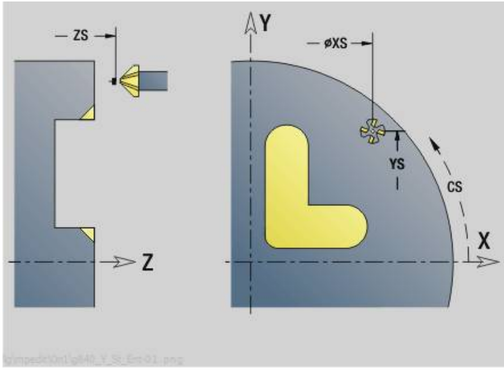
Further forms:

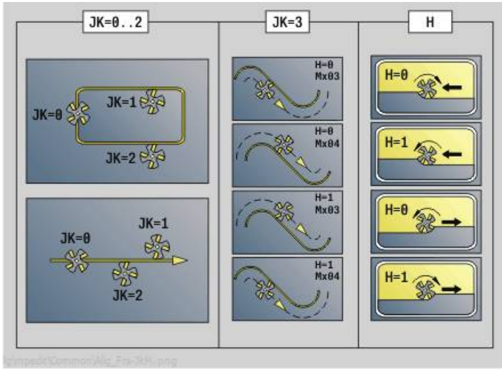
Further information: "smart.Turn unit", Page 98

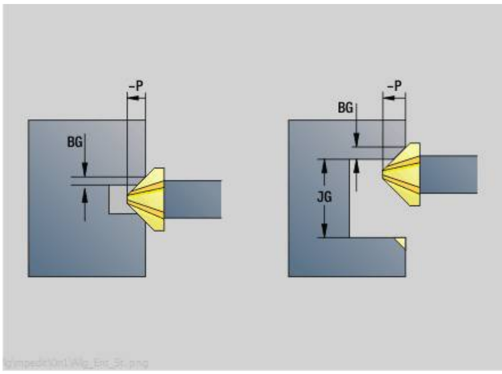
Access to the technology database:

- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G840 ICP deburring on front face in Y







The unit deburrs the contour defined with **ICP** in the XY plane.

Unit name: **G840_ENT_Y_STIRN** / cycle: **G840**

Further information: "G840 – deburring", Page 470

Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
- **BF**: **Machine form elements** (default: 0)

  A chamfer/rounding arc is machined
  - **0: No machining**
  - **1: At beginning**
  - **2: At end**
  - **3: At beginning and end**
  - **4: Only chamfer/rounding** is machined—not the basic element (requirement: the contour section consists of a single element)
- **Z1**: **Upper edge of milling**

Cycle form:

- **JK**: **Cutter position**
  - **0: On the contour**
  - **1: Within/left of contour**
  - **2: Outside/right of contour**
  - **3: Depending on H and MD**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **BG**: **Cham. width** for deburring
- **JG**: **Preparation diameter**
- **P**: **Plunging depth** (indicated as a negative value)
- **I**: **Contour-parallel oversize**
- **R**: **Approach radius**
- **FZ**: **Approach feed** (default: active feed rate)
- **E**: **Reduced feed**
- **RB**: **Return plane** (default: back to start position)

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **Deburring**
- Affected parameters: **F**, **S**

Unit G841 Single surface in Y axis on face

The unit mills a single surface defined with **ICP** in the XY plane.

Unit name: **G841_Y_STI** / cycles: **G841**; **G842**

Further information: "Area milling - roughing G841", Page 655

Further information: "Area milling - finishing G842", Page 656

Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section

Cycle form:

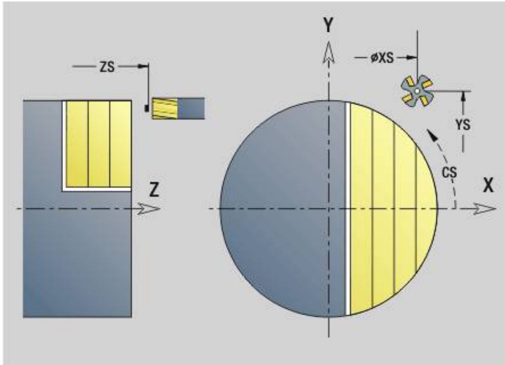
- **QK**: **Machining operation**
  - Roughing
  - Finishing
- **P**: **Maximum infeed**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
- **V**: **Overrun factor** – defines the distance by which the tool should pass the outside radius of the workpiece (default: 0.5)
- **FZ**: **Approach feed** (default: active feed rate)
- **RB**: **Return plane** (default: back to start position)

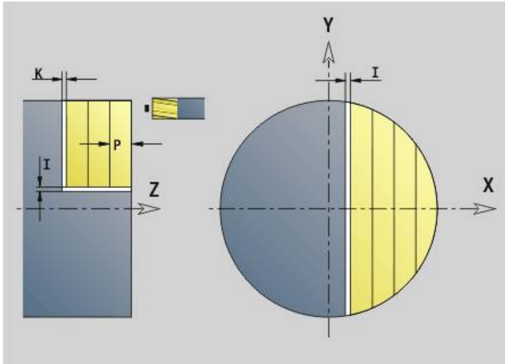
Further forms:

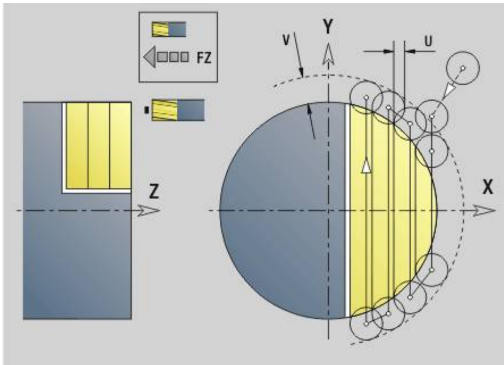
Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**







Unit G843 Centric polygon, Y axis on face

The unit mills the centric polygon defined with **ICP** in the XY plane.

Unit name: **G843_Y_STI** / cycles: **G843**; **G844**

Further information: "Centric polygon - roughing G843", Page 657

Further information: "Centric polygon - finishing G844", Page 658

Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section

Cycle form:

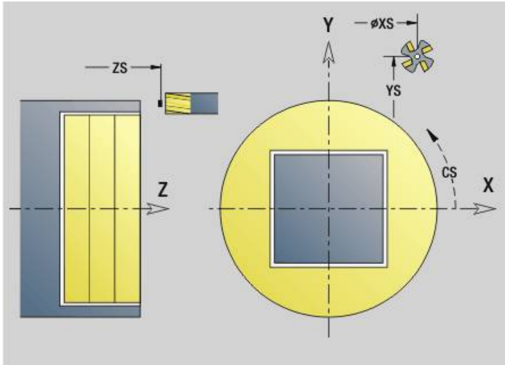
- **QK**: **Machining operation**
  - Roughing
  - Finishing
- **P**: **Maximum infeed**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
- **V**: **Overrun factor** – defines the distance by which the tool should pass the outside radius of the workpiece (default: 0.5)
- **FZ**: **Approach feed** (default: active feed rate)
- **RB**: **Return plane** (default: back to start position)

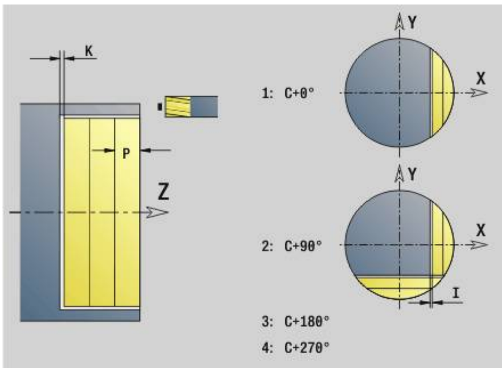
Further forms:

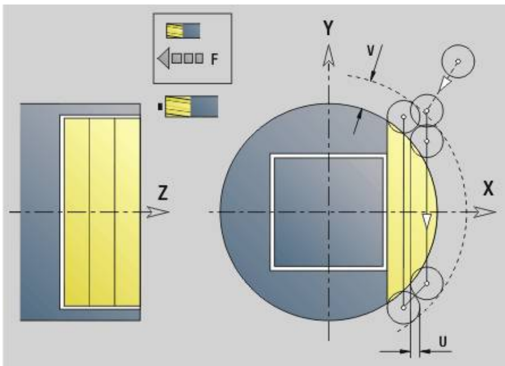
Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

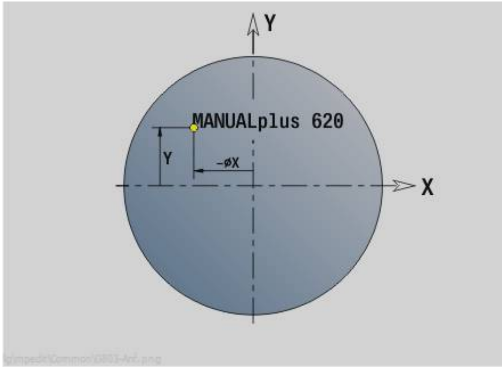


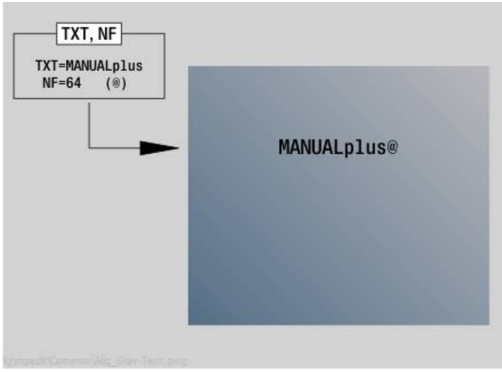


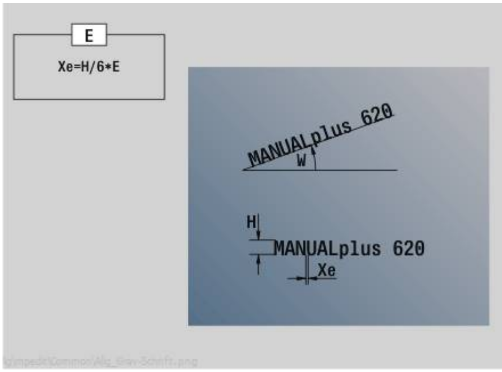


Unit G803 Engraving in Y axis on face

The unit engraves character strings aligned linearly in the XY plane. Diacritics and special characters that you cannot enter in **smart.Turn** operating mode can be defined, character by character, in **NF**. If you program **Q** = 1 (**Continue from last text**), then the tool change and pre-positioning are suppressed. The technological data of the previous engraving cycle apply.







Unit name: **G803_GRA_Y_STIRN** / cycle: **G803**

Further information: "Engraving in XY plane G803", Page 666

Position form:

- **X**, **Y**: **Start point**
- **Z**: **Final point** – Z position, infeed depth during milling
- **RB**: **Return plane**
- **APP**: **Type of approach**
- **DEP**: **Type of departure**

Cycle form:

- **TXT**: **Text** to be engraved
- **NF**: **Char. no.** – ASCII code of the character to be engraved
- **H**: **Font height**
- **E**: **Distance factor**
  The spacing between the characters is calculated according to the following formula: **H** / 6 * **E**
- **W**: **Inclinat. ang.** of the character string
- **FZ**: **Plunging feed rate factor** (plunging feed rate = current feed rate * **FZ**)
- **Q**: **Continue from last text**
  - **0** (**No**): Engraving starts at the starting point
  - **1** (**Yes**): Engraving starts at the tool position
- **O**: **Mirror writing**
  - **0** (**No**): Engraving is not mirrored
  - **1** (**Yes**): Engraving is mirrored (mirror writing)
- **NS**: **Block number of contour** – reference to the contour description

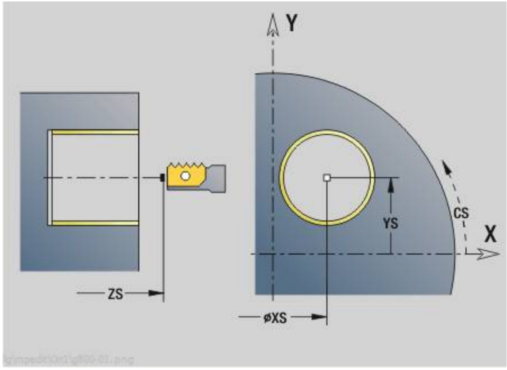
Further forms:

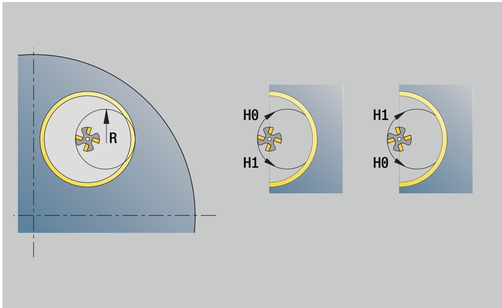
Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **Engraving**
- Affected parameters: **F**, **S**

Unit G800 Thread milling on face in Y





The unit mills a thread in existing holes in the XY plane.

Unit name: **G800_GEW_Y_STIRN** / cycle: **G800**

Further information: "Thread milling in XY plane G800", Page 668

Position form:

- **APP**: **Type of approach**
- **CS**: **Approach position C** – C-axis position that is approached before cycle call with **G110**
- **Z1**: **Start point drill**
- **P2**: **Thread depth**
- **I**: **Thread diameter**
- **F1**: **Thread pitch**

Cycle form:

- **J**: **Direction of thread:**
  - **0: Right-hand thread**
  - **1: Left-hand thread**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **V**: **Milling method**
  - **0: One revolution** – the thread is milled in a 360-degree helix
  - **1: Two or more revolutions** – the thread is milled in several helix paths (single-point tool)
- **R**: **Approach radius**
- **FK**: **Finished part contour** – name of the contour to be machined
- **NS**: **Block number of contour** – reference to the contour description

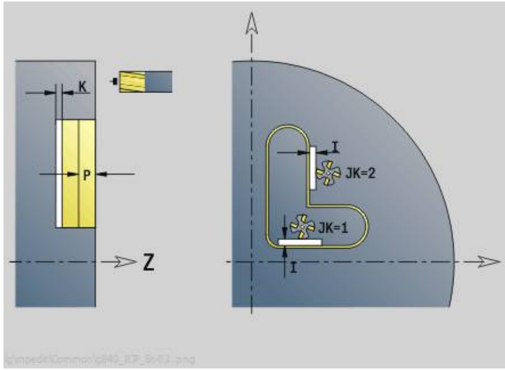
Further forms:

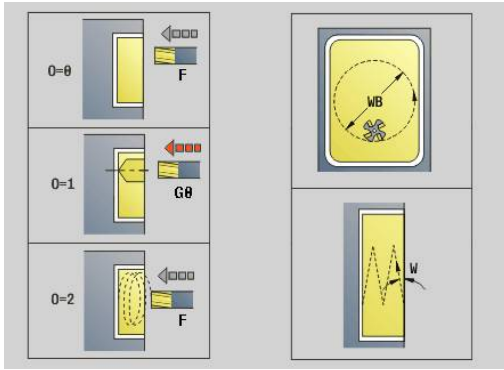
Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: Finish-milling
- Affected parameters: **F**, **S**

Unit G847 ICP Troch. contour mllng, front face Y





The unit uses trochoidal milling to rough out the open or closed contour on the front face of the workpiece defined with **ICP**.

Unit name: **G847_KON_Y_STIRN** / cycle: **G847**

Further information: "Trochoidal contour milling G847 ", Page 480

Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
- **BF**: **Machine form elements** (default: 0)

  A chamfer/rounding arc is machined
  - **0: No machining**
  - **1: At beginning**
  - **2: At end**
  - **3: At beginning and end**
  - **4: Only chamfer/rounding** is machined—not the basic element (requirement: the contour section consists of a single element)
- **Z1**: **Upper edge of milling**
- **P2**: **Depth of contour**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **RB**: **Return plane** (default: back to start position)
- **NF**: **Position mark** (only with **O** = 1)

Cycle form:

- **JK**: **Cutter position**
  - **0: On the contour**
  - **1: Within/left of contour**
  - **2: Outside/right of contour**
- **H**: **Mill cutting direction** (default: 1)
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Maximum infeed**
- **BR**: **Trochoid width**
- **R**: **Radius for return**
- **FP**: **Feed rate for return** (default: active feed rate)
- **AL**: **Retraction path for return**

- **O**: **Plunging behavior** (default: 2)
  - **O** = 0 (vertical plunging): The cycle moves the tool to the starting point; the tool plunges at the feed rate for infeed and mills the contour
  - **O** = 1 (vertical plunging, e.g. at the predrilled position):
    - If **NF** is programmed: The control positions the milling cutter above the first predrill hole position; the tool plunges at rapid traverse to safety clearance and mills the first area. If applicable, the cycle positions the tool to the next predrill hole position and mills the next area etc.
    - If **NF** is not programmed: The cycle plunges at the current position at rapid traverse and mills the area. If applicable, position the tool to the next predrill hole position and mill the next area etc.
  - **O** = 2 (plunge in helical motion): The tool plunges at the current position at the angle **W** and mills full circles with the diameter **WB**.
- **FZ**: **Approach feed** (default: active feed rate)
- **EW**: **Plunging angle**
- **WB**: **Diameter of the helix** (default: helix diameter = 1.5 * milling diameter)
- **U**: **Overlap factor** – overlap of milling paths = **U** * milling diameter (default: 0.9)
- **HCC**: **Contour smoothing**
  - **0: No smoothing cut**
  - **1: With smoothing cut**

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

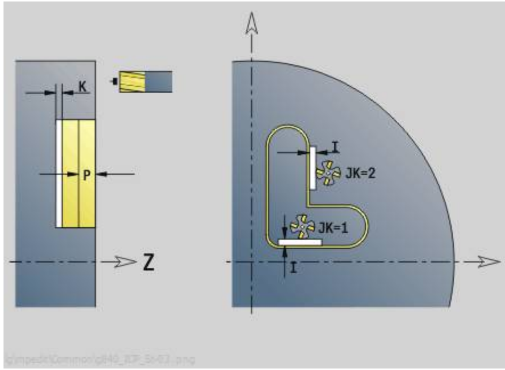
- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

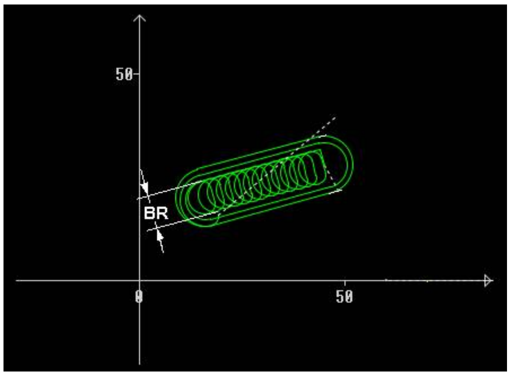
Unit G848 ICP Troch. pocket mllng, front face Y

The unit uses trochoidal milling to rough out the figure or figure pattern on the front face defined with **ICP**.

Unit name: **G848_TAS_Y_STIRN** / cycle: **G848**

Further information: "Trochoidal pocket milling G847 ", Page 482





Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **Z1**: **Upper edge of milling**
- **P2**: **Depth of contour**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **RB**: **Return plane** (default: back to start position)
- **NF**: **Position mark** (only with **O** = 1)

Cycle form:

- **H**: **Mill cutting direction** (default: 1)
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Maximum infeed**
- **BR**: **Trochoid width**
- **R**: **Radius for return**
- **FP**: **Feed rate for return** (default: active feed rate)
- **AL**: **Retraction path for return**
- **O**: **Plunging behavior** (default: 2)
  - **O**=0 (vertical plunging): The cycle moves the tool to the starting point; the tool plunges at the feed rate for infeed and mills the figure
  - **O** = 1 (vertical plunging, e.g. at the predrilled position):
    - If **NF** is programmed: The control positions the milling cutter above the first predrill hole position; the tool plunges at rapid traverse to safety clearance and mills the first area. If applicable, the cycle positions the tool to the next predrill hole position and mills the next area etc.
    - If **NF** is not programmed: The cycle plunges at the current position at rapid traverse and mills the area. If applicable, position the tool to the next predrill hole position and mill the next area etc.
  - **O** = 2 (plunge in helical motion): The tool plunges at the current position at the angle **W** and mills full circles with the diameter **WB**.
- **FZ**: **Approach feed** (default: active feed rate)
- **EW**: **Plunging angle**
- **WB**: **Diameter of the helix** (default: helix diameter = 1.5 * milling diameter)
- **U**: **Overlap factor** – overlap of milling paths = **U** * milling diameter (default: 0.9)

- **J**: **Machining operation**
  - **0: Complete**
  - **1: W/o corner machining**
  - **2: Only corner machining**

> For slots and rectangles, you need to program width **BR** of the trochoidal tool path; for circles and polygons, this is not necessary.

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G840 ICP contour mill, latrl. surf. Y

The unit mills the contour defined with **ICP** in the YZ plane.

Unit name: **G840_Kon_Y_Mant** / cycle: **G840**

Further information: "G840 – milling", Page 466

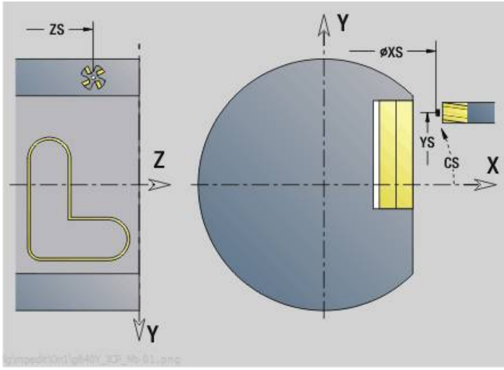
Contour form:

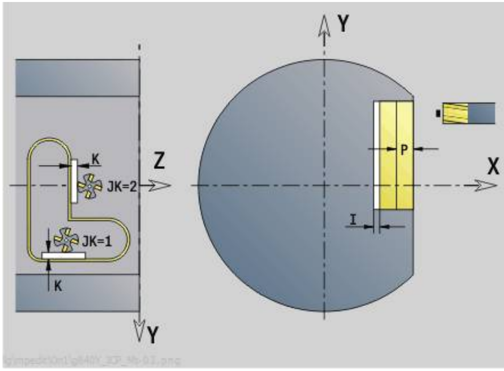
- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
- **BF**: **Machine form elements** (default: 0)

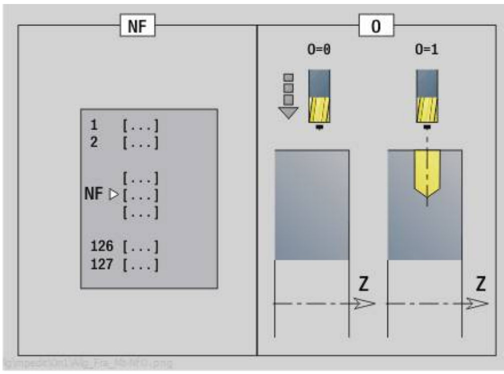
  A chamfer/rounding arc is machined
  - **0: No machining**
  - **1: At beginning**
  - **2: At end**
  - **3: At beginning and end**
  - **4: Only chamfer/rounding** is machined—not the basic element (requirement: the contour section consists of a single element)
- **X1**: **Upper edge of milling**
- **P2**: **Depth of contour**

Cycle form:

- **JK**: **Cutter position**
  - **0: On the contour**
  - **1: Within/left of contour**
  - **2: Outside/right of contour**
  - **3: Depending on H and MD**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Maximum infeed**
- **I**: **Infeed-direction oversize**
- **K**: **Contour-parallel oversize**
- **FZ**: **Approach feed** (default: active feed rate)
- **E**: **Reduced feed**
- **R**: **Approach radius**
- **O**: **Plunging behavior** (default: 0)
  - **0: Straight** – the cycle moves the tool to the starting point; the tool plunges at feed rate and mills the contour
  - **1: In predrilling** – the control positions the tool above the predrill hole position; the tool plunges and mills the contour
- **NF**: **Position mark** (only with **O** = 1)
- **RB**: **Return plane** (default: back to start position)







Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: Finish-milling
- Affected parameters: **F**, **S**, **FZ**, **P**

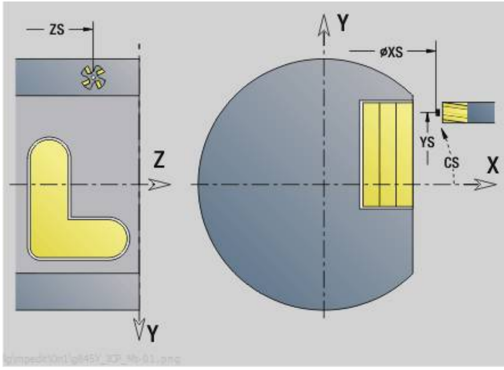
Unit G845 ICP pocket mill, lateral surf. Y

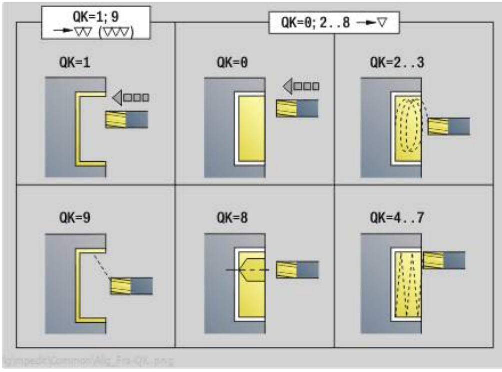
The unit mills the pocket defined with **ICP** in the YZ plane. In **QK** (machining operation), select whether a roughing or finishing operation is to be executed. For roughing, define the plunging strategy.

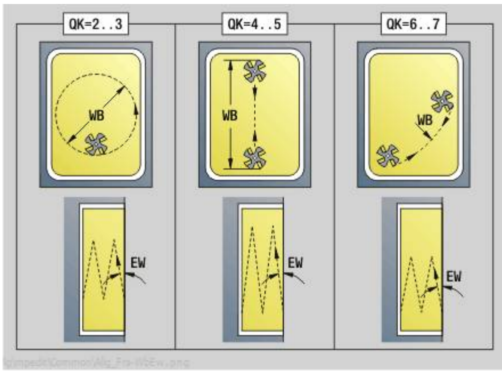
Unit name: **G845_Tas_Y_Mant** / cycles: **G845**; **G846**

Further information: "G845 – milling", Page 474

Further information: "Pocket milling - finishing G846", Page 478







Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **X1**: **Upper edge of milling**
- **P2**: **Depth of contour**
- **NF**: **Position mark** (only with **QK** = 8)
- **FZ**: **Approach feed** (default: active feed rate)
- **E**: **Reduced feed**
- **FP**: **Infeed rate in the plane**

Cycle form:

- **QK**: **Machining operation** and plunging strategy
  - **0: Roughing**
  - **1: Finishing**
  - **2: Helical roughing, manual**
  - **3: Helical roughing, automatic**
  - **4: Recipr. linear roughing, manual**
  - **5: Recipr. linear roughing, auto**
  - **6: Reciproc. circ. roughing, manual**
  - **7: Recipr. circular roughing, auto**
  - **8: Plunge rough at predrill pos.**
  - **9: Finishing with 3-D approach arc**
- **JT**: **Machining direction**
  - **0: From the inside out**
  - **1: From the outside in**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Maximum infeed**
- **I**: **Infeed-direction oversize**
- **K**: **Contour-parallel oversize**
- **R**: **Approach radius**
- **WB**: **Plunging length**
- **EW**: **Plunging angle**
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
- **RB**: **Return plane** (default: back to start position)

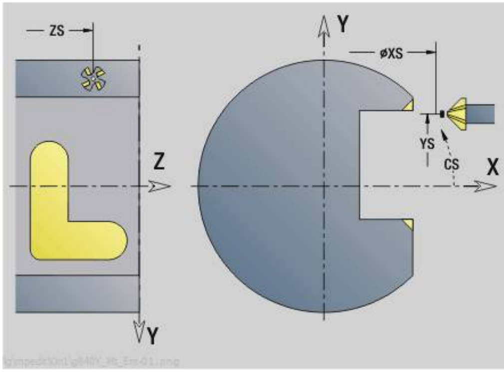
Further forms:

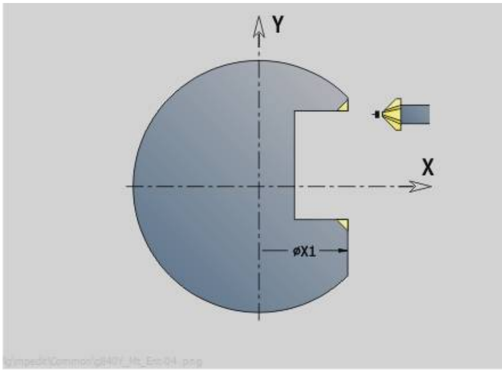
Further information: "smart.Turn unit", Page 98

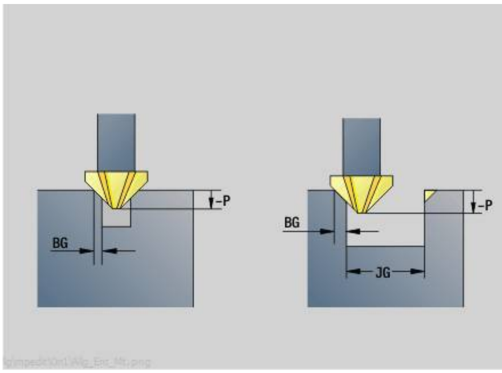
Access to the technology database:

- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G840 ICP deburring on lateral surf. Y







The unit deburrs the contour defined with **ICP** in the YZ plane.

Unit name: **G840_ENT_Y_MANT** / cycle: **G840**

Further information: "G840 – deburring", Page 470

Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
- **BF**: **Machine form elements** (default: 0)
  A chamfer/rounding arc is machined
  - **0: No machining**
  - **1: At beginning**
  - **2: At end**
  - **3: At beginning and end**
  - **4: Only chamfer/rounding** is machined—not the basic element (requirement: the contour section consists of a single element)
- **X1**: **Upper edge of milling**

Cycle form:

- **JK**: **Cutter position**
  - **0: On the contour**
  - **1: Within/left of contour**
  - **2: Outside/right of contour**
  - **3: Depending on H and MD**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **BG**: **Cham. width** for deburring
- **JG**: **Preparation diameter**
- **P**: **Plunging depth** (indicated as a negative value)
- **K**: **Contour-parallel oversize**
- **R**: **Approach radius**
- **FZ**: **Approach feed** (default: active feed rate)
- **E**: **Reduced feed**
- **RB**: **Return plane** (default: back to start position)

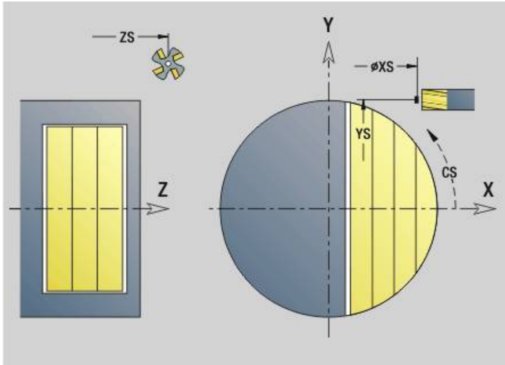
Further forms:

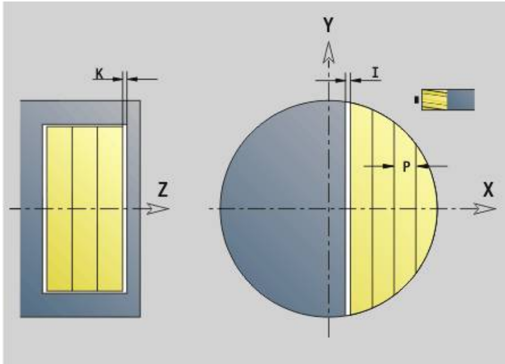
Further information: "smart.Turn unit", Page 98

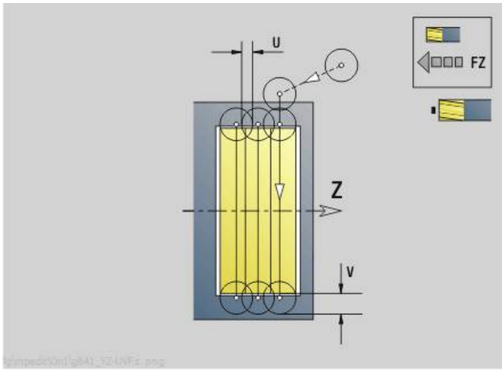
Access to the technology database:

- Machining operation: **Deburring**
- Affected parameters: **F**, **S**

Unit G841 Single surface in Y axis, latrl.







The unit mills a single surface defined with **ICP** in the YZ plane.

Unit name: **G841_Y_MANT** / cycles: **G841**, **G842**

Further information: "Area milling - roughing G841", Page 655

Further information: "Area milling - finishing G842", Page 656

Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section

Cycle form:

- **QK**: **Machining operation**
  - Roughing
  - Finishing
- **P**: **Maximum infeed**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
- **V**: **Overrun factor** – defines the distance by which the tool should pass the outside radius of the workpiece (default: 0.5)
- **FZ**: **Approach feed** (default: active feed rate)
- **RB**: **Return plane** (default: back to start position)

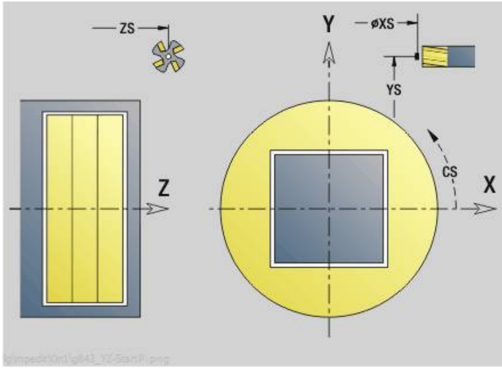
Further forms:

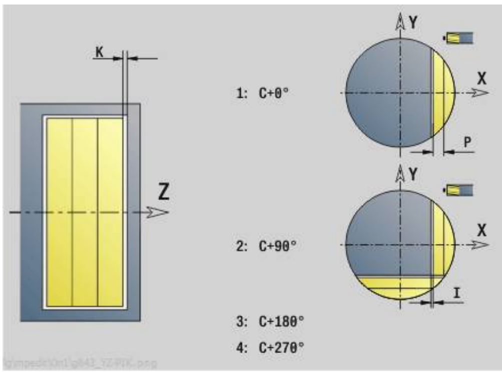
Further information: "smart.Turn unit", Page 98

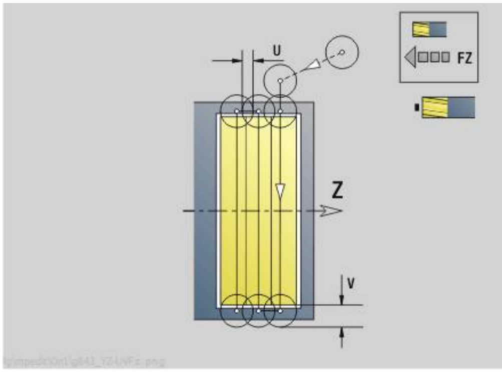
Access to the technology database:

- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G843 Centric polygon Y axis, latrl.







The unit mills the centric polygon defined with **ICP** in the YZ plane.

Unit name: **G843_Y_MANT** / cycles: **G843**; **G844**

Further information: "Centric polygon - roughing G843", Page 657

Further information: "Centric polygon - finishing G844", Page 658

Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section

Cycle form:

- **QK**: **Machining operation**
  - Roughing
  - Finishing
- **P**: **Maximum infeed**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
- **V**: **Overrun factor** – defines the distance by which the tool should pass the outside radius of the workpiece (default: 0.5)
- **FZ**: **Approach feed** (default: active feed rate)
- **RB**: **Return plane** (default: back to start position)

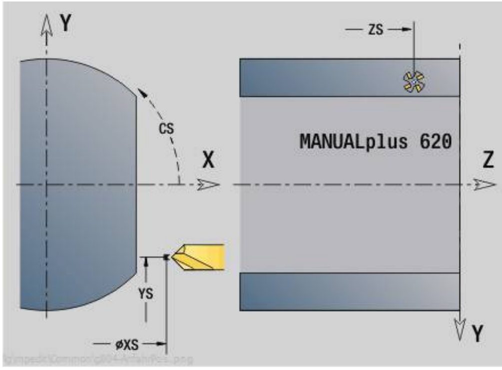
Further forms:

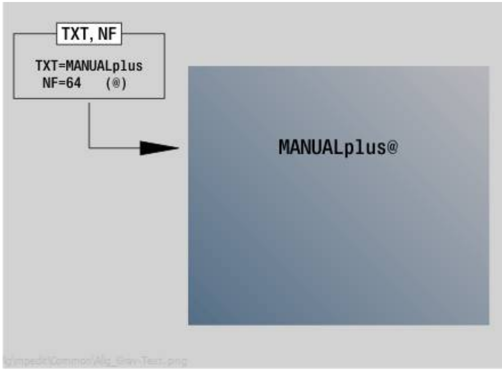
Further information: "smart.Turn unit", Page 98

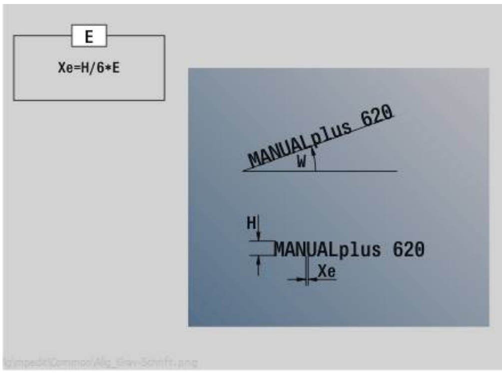
Access to the technology database:

- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

Unit G804 Engraving in Y on latrl. surface







The unit engraves character strings aligned linearly in the YZ plane. Diacritics and special characters that you cannot enter in **smart.Turn** operating mode can be defined, character by character, in **NF**. If you program **Q** = 1 (**Continue from last text**), then the tool change and pre-positioning are suppressed. The technological data of the previous engraving cycle apply.

Unit name: **G804_GRA_Y_MANT** / cycle: **G804**

Further information: "Engraving in YZ plane G804", Page 667

Position form:

- **Y**, **Z**: **Start point**
- **X**: **Final point** – X position, infeed depth during milling (diameter value)
- **RB**: **Return plane**

Cycle form:

- **TXT**: **Text** to be engraved
- **NF**: **Char. no.** – ASCII code of the character to be engraved
- **H**: **Font height**
- **E**: **Distance factor**
  The spacing between the characters is calculated according to the following formula: **H** / 6 * **E**
- **W**: **Inclinat. ang.** of the character string
- **FZ**: **Plunging feed rate factor** (plunging feed rate = current feed rate * **FZ**)
- **Q**: **Continue from last text**
  - **0** (**No**): Engraving starts at the starting point
  - **1** (**Yes**): Engraving starts at the tool position
- **O**: **Mirror writing**
  - **0** (**No**): Engraving is not mirrored
  - **1** (**Yes**): Engraving is mirrored (mirror writing)
- **NS**: **Block number of contour** – reference to the contour description

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **Engraving**
- Affected parameters: **F**, **S**

Unit G806 Thread milling, lateral in Y

The unit mills a thread in existing holes in the YZ plane.

Unit name: **G806_GEW_Y_MANT** / cycle: **G806**

Further information: "Thread milling in YZ plane G806", Page 669

Position form:

- **APP**: **Type of approach**
- **CS**: **Approach position C** – C-axis position that is approached before cycle call with **G110**
- **X1**: **Start point drill** (diameter value)
- **P2**: **Thread depth**
- **I**: **Thread diameter**
- **F1**: **Thread pitch**

Cycle form:

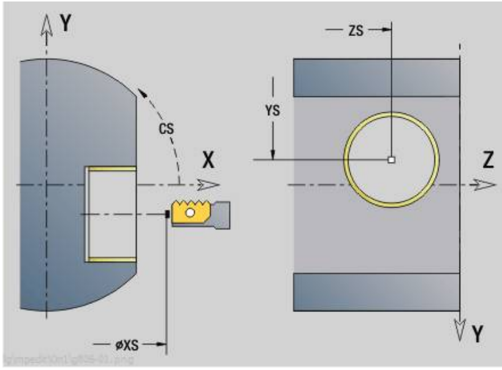
- **J**: **Direction of thread:**
  - **0: Right-hand thread**
  - **1: Left-hand thread**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **V**: **Milling method**
  - **0: One revolution** – the thread is milled in a 360-degree helix
  - **1: Two or more revolutions** – the thread is milled in several helix paths (single-point tool)
- **R**: **Approach radius**
- **FK**: **Finished part contour** – name of the contour to be machined
- **NS**: **Block number of contour** – reference to the contour description

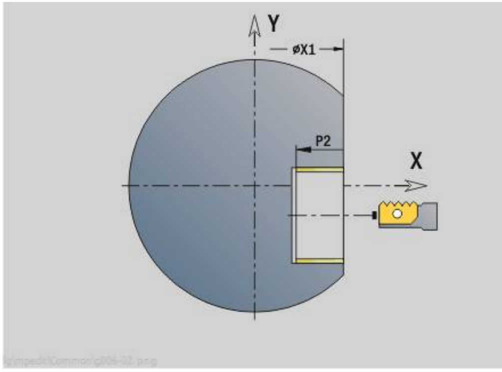
Further forms:

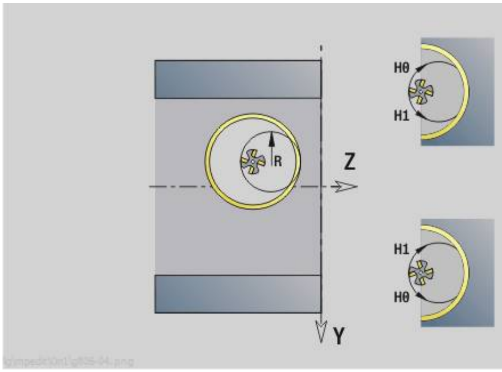
Further information: "smart.Turn unit", Page 98

Access to the technology database:

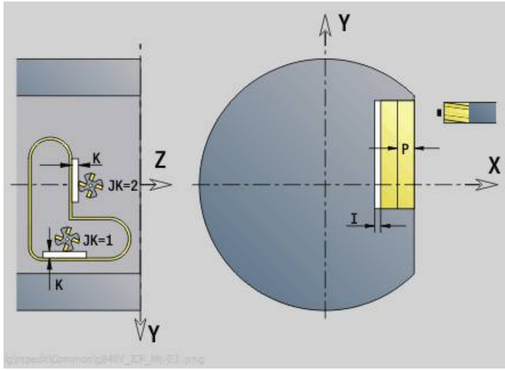
- Machining operation: Finish-milling
- Affected parameters: **F**, **S**

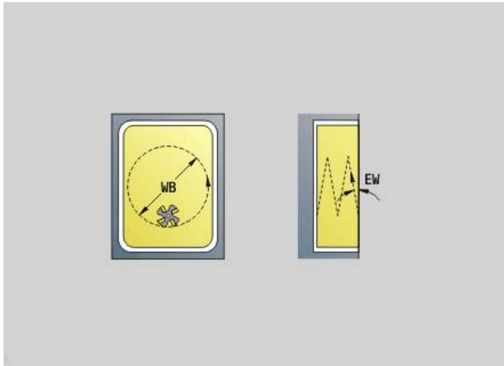






Unit G847 ICP Troch. contour mllng, lat. surf. Y





The unit uses trochoidal milling to rough out the open or closed contour on the lateral surface of the workpiece defined with **ICP**.

Unit name: **G847_KON_Y_MANT** / cycle: **G847**

Further information: "Trochoidal contour milling G847 ", Page 480

Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
- **BF**: **Machine form elements** (default: 0)

  A chamfer/rounding arc is machined
  - **0: No machining**
  - **1: At beginning**
  - **2: At end**
  - **3: At beginning and end**
  - **4: Only chamfer/rounding** is machined—not the basic element (requirement: the contour section consists of a single element)
- **X1**: **Upper edge of milling** (diameter value; default: **Starting point X**)
- **P2**: **Depth of contour**
- **I**: **Infeed-direction oversize**
- **K**: **Contour-parallel oversize**
- **RB**: **Return plane** (default: back to start position)
- **NF**: **Position mark** (only with **O** = 1)

Cycle form:

- **JK**: **Cutter position**
  - **0: On the contour**
  - **1: Within/left of contour**
  - **2: Outside/right of contour**
- **H**: **Mill cutting direction** (default: 1)
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Maximum infeed**
- **BR**: **Trochoid width**
- **R**: **Radius for return**
- **FP**: **Feed rate for return** (default: active feed rate)
- **AL**: **Retraction path for return**

- **O**: **Plunging behavior** (default: 2)
  - **O** = 0 (vertical plunging): The cycle moves the tool to the starting point; the tool plunges at the feed rate for infeed and mills the contour
  - **O** = 1 (vertical plunging, e.g. at the predrilled position):
    - If **NF** is programmed: The control positions the milling cutter above the first predrill hole position; the tool plunges at rapid traverse to safety clearance and mills the first area. If applicable, the cycle positions the tool to the next predrill hole position and mills the next area etc.
    - If **NF** is not programmed: The cycle plunges at the current position at rapid traverse and mills the area. If applicable, position the tool to the next predrill hole position and mill the next area etc.
  - **O** = 2 (plunge in helical motion): The tool plunges at the current position at the angle **W** and mills full circles with the diameter **WB**.
- **FZ**: **Approach feed** (default: active feed rate)
- **EW**: **Plunging angle**
- **WB**: **Diameter of the helix** (default: helix diameter = 1.5 * milling diameter)
- **U**: **Overlap factor** – overlap of milling paths = **U** * milling diameter (default: 0.9)
- **HCC**: **Contour smoothing**
  - **0: No smoothing cut**
  - **1: With smoothing cut**

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

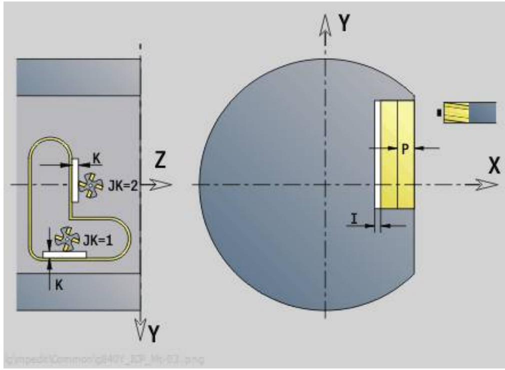
- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

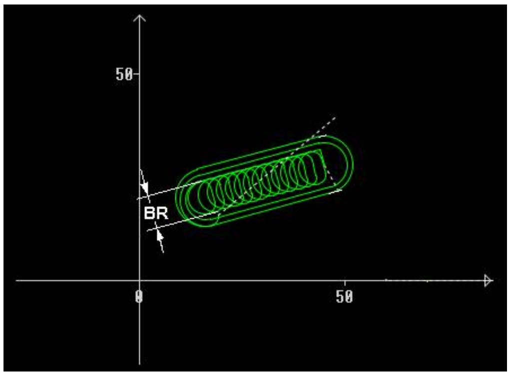
Unit G848 ICP Troch. pocket mllng, lat. surf. Y

The unit uses trochoidal milling to rough out the figures defined on the lateral surface with **ICP**.

Unit name: **G848_TAS_Y_MANT** / cycle: **G848**

Further information: "Trochoidal pocket milling G847 ", Page 482





Contour form:

- **FK**: **ICP contour number**
- **NS**: **Starting block no. of contour** – beginning of contour section
- **X1**: **Upper edge of milling**
- **P2**: **Depth of contour**
- **I**: **Contour-parallel oversize**
- **K**: **Infeed-direction oversize**
- **RB**: **Return plane** (default: back to start position)
- **NF**: **Position mark** (only with **O** = 1)

Cycle form:

- **H**: **Mill cutting direction** (default: 1)
  - **0: Up-cut**
  - **1: Climb**
- **P**: **Maximum infeed**
- **BR**: **Trochoid width**
- **R**: **Radius for return**
- **FP**: **Feed rate for return** (default: active feed rate)
- **AL**: **Retraction path for return**
- **O**: **Plunging behavior** (default: 2)
  - **O**=0 (vertical plunging): The cycle moves the tool to the starting point; the tool plunges at the feed rate for infeed and mills the figure
  - **O** = 1 (vertical plunging, e.g. at the predrilled position):
    - If **NF** is programmed: The control positions the milling cutter above the first predrill hole position; the tool plunges at rapid traverse to safety clearance and mills the first area. If applicable, the cycle positions the tool to the next predrill hole position and mills the next area etc.
    - If **NF** is not programmed: The cycle plunges at the current position at rapid traverse and mills the area. If applicable, position the tool to the next predrill hole position and mill the next area etc.
  - **O** = 2 (plunge in helical motion): The tool plunges at the current position at the angle **W** and mills full circles with the diameter **WB**.
- **FZ**: **Approach feed** (default: active feed rate)
- **EW**: **Plunging angle**
- **WB**: **Diameter of the helix** (default: helix diameter = 1.5 * milling diameter)
- **U**: **Overlap factor** – overlap of milling paths = **U** * milling diameter (default: 0.9)

- **J**: **Machining operation**
  - **0: Complete**
  - **1: W/o corner machining**
  - **2: Only corner machining**

> For slots and rectangles, you need to program width **BR** of the trochoidal tool path; for circles and polygons, this is not necessary.

Further forms:

Further information: "smart.Turn unit", Page 98

Access to the technology database:

- Machining operation: **milling**
- Affected parameters: **F**, **S**, **FZ**, **P**

6

DIN/ISO Programming

6.1 Programming in ISO Mode

Geometry and machining commands

The control also supports structured programming in **ISO Mode**.

The **G** commands are divided into:

- **Geometry commands** for describing the contours of the workpiece blanks and finished parts
- **Machining commands** for the **MACHINING** section.

> Some **G** codes are used both for workpiece blank/finished-part definition and in the **MACHINING** section. When copying or shifting NC blocks, keep in mind that **geometry** functions are used exclusively for describing a contour, while **machining** functions are used only in the **MACHINING** section.

Example: Structured DIN PLUS program

PROGRAM HEAD		
#MATERIAL	Steel	
#MACHINE	Automatic lathe	
#DRAWING	356_787.9	
#CLAMP. PRESS.	20	
#SLIDE	$1	
#COMPANY	Turn & Co	
#UNIT	METRIC	
TURRET 1		
T1 ID"342-300.1"		
T2 ID"111-80-080.1"		
. . .		
BLANK		
N1 G20 X120 Z120 K2		
FINISHED PART		
N2 G0 X60 Z-115		
N3 G1 Z-105		
. . .		
MACHINING		
N22 G59 Z282		
N25 G14 Q0		
		[Drilling]
N26 T1		
N27 G97 S1061 G95 F0.25 M4		
. . .		
END		

Contour programming

The "contour follow-up" function and use of contour-related turning cycles require the previous description of the contour of the workpiece blank and of the finished part. With milling and drilling, contour definition is a precondition for the use of machining cycles.

> Use **ICP** (Interactive Contour Programming) for describing blank and finished parts.

Contours for turning:

- Describe a continuous contour
- The direction of the contour description is independent of the direction of machining
- Contour descriptions must not extend beyond the turning center
- The contour of the finished part must lie within the contour of the blank part
- When machining bars, define only the required section as the workpiece blank
- Contour definitions apply to the entire NC program, even if the workpiece is rechucked for machining the rear face
- In the fixed cycles, the defined contour is used to program **reference values**

To describe **workpiece blanks** and **auxiliary workpiece blanks**, use:

- **G20** "Blank part macro" for standard parts (cylinder, hollow cylinder)
- **G21** "Cast-part macro" for blank-part contours based on finished-part contours **G21** is only used for describing workpiece blanks
- Individual contour elements (such as are used for finished-part contours) where use of **G20** or **G21** is not possible

To describe finished parts, use individual contour elements and form elements. The contour elements or the complete contour can be assigned attributes accounted for during the machining of the workpiece (example: oversizes, additive compensation, special feed rates, etc.). The control always uses paraxial elements to close finished parts.

For intermediate machining steps, define auxiliary contours. Auxiliary contours are programmed in the same way as finished-part descriptions. One contour description is possible per **Auxiliary contour**. An **Auxiliary contour** is assigned a name (**ID**) that can be referenced by the cycles. Auxiliary contours are not closed automatically.

Contours for C-axis machining:

- Contours for C-axis machining are programmed within the **FINISHED PART** section.
- Identify the contours as **FRONT** or **LATERAL**. You can use section codes more than once or program multiple contours within one section code

Block references: When editing **G** codes related to the contour (**MACHINING** section), load the block references from the displayed contour.

Load the block reference:

- Place the cursor in the input box (**NS**)
- Switch to the contour display
- Place the cursor on the desired contour element.
- Switch to **NE**
- Place the cursor on the desired contour element.
- Press the **Take over** soft key to return to the dialog

NC blocks of the DIN/ISO program

An NC block contains NC commands such as positioning, switching, or organizational commands. Traversing and switching commands begin with **G** or **M** followed by a number (**G1**, **G2**, **G81**, **M3**, **M30**, ...) and the address parameters. Organizational commands consist of **key words** (**WHILE** , **RETURN**, etc.) or of a combination of letters and numbers.

You can also program NC blocks containing only variable calculations.

You can program several NC commands in one NC block, provided they have different address letters and **do not have opposing** functions.

Examples:

- Permissible combination: **N10 G1 X100 Z2 M8**
- Non-permissible combination: **N10 G1 X100 Z2 G2 X100 Z2 R30** – same address letters are used more than once or **N10 M3 M4** – opposing functionality

To describe **workpiece blanks** and **auxiliary workpiece blanks**, use:

- **G20** "Blank part macro" for standard parts (cylinder, hollow cylinder)
- **G21** "Cast-part macro" for blank-part contours based on finished-part contours **G21** is only used for describing workpiece blanks
- Individual contour elements (such as are used for finished-part contours) where use of **G20** or **G21** is not possible

NC address parameters – the address parameters consist of 1 or 2 letters followed by:

- A value
- A mathematical expression
- A **question mark** (VGP simplified geometry programming)
- A letter **i** to designate incremental address parameters (examples: **Xi**..., **Ci**..., **XKi**..., **YKi**..., etc.)
- A **#** variable
- A constant (_constname)

Examples:

- **X20** [Absolute dimension]
- **Zi-35.675** [Incremental dimension]
- **X?** [Simplified geometry programming]
- **X#l1** [Variable programming]
- **X(#g12+1)** [Variable programming]
- **X(37+2)*SIN (30)** [Mathematical expression]
- **X(20*_pi)** [Expression with constant]

Creating, editing and deleting NC blocks

Create NC block:

INS

- ▶ Press the **INS** key
- > The control creates a new NC block below the cursor position.
- ▶ Alternatively, you can program the NC command directly.
- > The control creates a new NC block or inserts the NC command into the existing NC block.

Delete the NC block:

- ▶ Position the cursor on the NC block to be deleted.

DEL

- ▶ Press the **DEL** key
- > The control deletes the NC block.

Add an NC element:

- ▶ Position the cursor on an element of the NC block (NC block number, **G** command, **M** command, address parameter, etc.)
- ▶ Insert NC element (**G**, **M**, **T** function etc.)

Change NC element:

- ▶ Position the cursor on an element of the NC block (NC block number, **G** command, **M** command, address parameter, etc.) or the section code.

ENT

- ▶ Press the **ENT** key
- ▶ Or double-click with the left mouse key
- > The control activates a dialog box displaying the block number, the number of the **G** or **M** function, or the address parameters, which can then be edited.

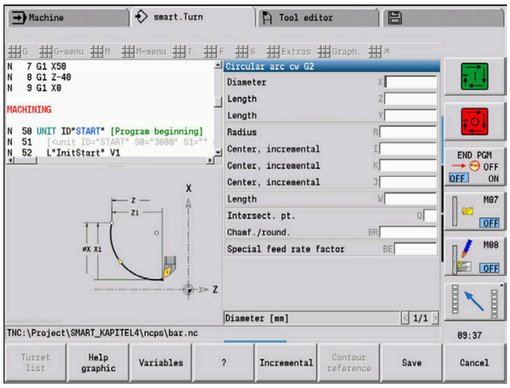
Delete NC element:

- ▶ Position the cursor on an element of the NC block (NC block number, **G** command, **M** command, address parameter, etc.) or the section code.

DEL

- ▶ Press the **DEL** key
- > The NC element highlighted by the cursor and all the related elements are deleted. Example: If the cursor is placed on a **G** command, the address parameters are also deleted..

Address parameters



Coordinates can be programmed absolutely or incrementally. If you do not make any entry for **X**, **Y**, **Z**, **XK**, **YK**, **C**, the coordinates of the block previously executed will be retained (modal).

The control calculates missing coordinates in the principal axes X, Y, or Z if you program **?** (simplified geometry programming).

The machining functions **G0**, **G1**, **G2**, **G3**, **G12**, and **G13** are modal. This means that the control uses the previous **G** command if the address parameters **X**, **Y**, **Z**, **I**, or **K** in the following block have been programmed without a **G** code. However, the address parameters must have been programmed as absolute values.

The control supports the use of variables and mathematical expressions as address parameters.

Editing address parameters:

- Call the dialog box.
  - Place the cursor in the input field.
  - Enter or edit values
  - As an alternative, use the additional input options provided by the soft keys:
    - **?** Simplified geometry programming
    - Switch from incremental to absolute
    - Activate variable input
    - Load the contour reference

Simplified geometry programming:

?

- Press the **?** soft key

?

- Press the **?** soft key again to display further options

Simplified geometry programming provides the following options:

- **?**: The control calculates the value
- **?>**: The control calculates the value. If there are two solutions, the control uses the higher value.
- **?<**: The control calculates the value. If there are two solutions, the control uses the lower value

Soft keys in the G dialog	
Help graphic	Alternately shows and hides the help graphics
Variables	Opens the alphabetic keyboard for entering variables (**GOTO** key)
?	Inserts the question mark for activating the simplified geometry programming
Incremental	Activates incremental programming for the current input parameter
Contour reference	Allows transferring the contour references for **NS** and **NE**

Machining cycles

HEIDENHAIN recommends programming a machining cycle as follows:

- Insert the tool
- Define the cutting data
- Position the tool in front of the machining area
- Define the safety clearance
- Call the cycle
- Retract the tool
- Move to the tool change position

NOTICE

Danger of collision!

Some parameters are non-volatile, e.g. special feed rates or approach and departure variants
With missing program steps (parameters are not redefined) the control uses the last programmed values for all subsequent machining operations. This may cause undesired constellations, e.g. feed rate for finishing with recessing cycles.

- Always use the recommended program structure
- Define all relevant parameters for each machining operation

Typical structure of a machining cycle

. . .	
MACHINING	
N.. G59 Z..	Datum shift
N.. G26 S..	Define the speed limit
N.. G14 Q..	Move to tool change point
. . .	
N.. T..	Insert the tool
N.. G96 S.. G95 F.. M4	Define the technology data
N.. G0 X.. Z..	Pre-position
N.. G47 P..	Define the safety clearance
N.. G810 NS.. NE..	Cycle call
N.. G0 X.. Z..	If necessary, retract
N.. G14 Q0	Move to tool change point
. . .	

Subprograms and expert programs

Subprograms are used to program the contour or the machining process.

In the subprogram, transfer parameters are available as variables. You can fix the designation of the transfer parameters and illustrate them in help graphics.

Further information: "Subprograms", Page 542

In every subprogram, the local variables **#l1** to **#l99** are available for internal calculations.

> Channel-dependent, initialized variables that, starting from the initialization programming level, are also effective in subprograms called from this level, are available in addition to the local variables.
>
> **Further information:** "General variables", Page 515

Subprograms can be nested up to six times. **Nesting** means that a subprogram calls a further subprogram, etc.

If a subprogram is to be run repeatedly, enter the number of times the subprogram is to be repeated in the **Q** parameter.

The control distinguishes between local and external subprograms:

- Local subprograms are in the file of the NC main program. Local subprograms can only be called in from their corresponding main programs
- External subprograms are stored in separate NC files and can be called in from any NC main program or other NC subprograms

Expert programs – an expert program is a subprogram that executes complex processes and is adapted to the machine configurations. Expert programs are usually provided by the machine tool builder.

NC program interpretation

For programming and user communication, keep in mind that the control interprets the NC program up to the fixed word MACHINING in the program selection.

The MACHINING section is not interpreted until you select **NC start**.

ISO programs of predecessor controls

The ISO program formats of the predecessor controls MANUALplus 4110 and CNC PILOT 4290 differ from the format of your current control. However, you can use the program converter to adapt programs of the predecessor control to the new control.

When opening an NC program, the control recognizes the programs of predecessor controls. The program concerned will be converted after a confirmation prompt. **CONV_...** will be prefixed to the program name.

This converter is also part of the **Transfer** submode.

When converting ISO programs, the different solutions for tool management, technology data, etc. must be taken into account, as well as contour description and variable programming.

Remember the following when converting **ISO programs of the MANUALplus 4110**:

- **Tool call**: The loading of the tool number depends on whether the program is a multifix program (2-digit tool number) or turret program (4-digit tool number):
  - 2-digit tool number: The tool number is loaded as **ID** and entered as the tool number **T1**
  - 4-digit tool number (**Tddpp**): The first two digits of the tool 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 **AUX. BLANK**
- **Contour descriptions**: In MANUALplus 4110 programs, the machining cycles are followed by the contour description. During conversion, the contour description is converted to an **AUX. BLANK**. 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 specified in the target program. You will have to add it manually later.

Remember the following when converting **ISO programs of the CNC PILOT 4290**:

- **Tool call** (**T** commands of the **TURRET** section):
  - **T** commands containing a reference to the tool database are left unchanged (example: **T1 ID"342-300.1"**)
  - **T** commands containing tool data cannot be converted
- **Variable programming**: Variable accesses to tool data, machine dimensions, **D** compensation values, parameter data and events cannot be converted. These program sequences have to be adapted
- **M functions** are left unchanged
- **Names of external subprograms:** When an external subprogram is called, the converter prefixes **CONV_...** to the name

> If the ISO 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.

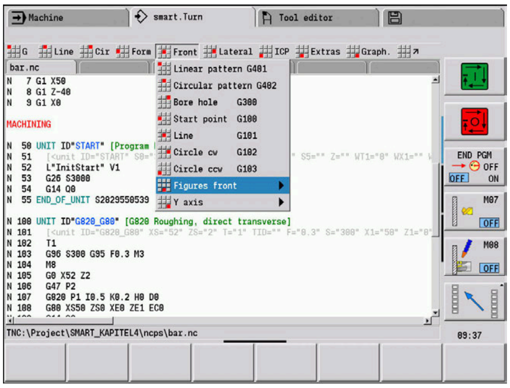
NOTICE

Danger of collision!

Converted NC programs may have incorrectly converted contents (machine-dependent) or nonconvertible contents. Danger of collision during machining!

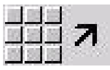
- Adapt converted NC programs to the current control
- Test the NC program in **Simulation** submode, using the graphic displayed there

Geometry menu item



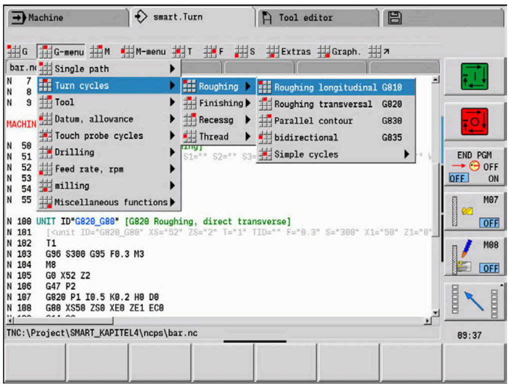
The **Geo»** (geometry) menu item contains functions for contour description. In **ISO Mode**, press the **Geo»** menu item to call the following menu items.

- **G**: Direct entry of a **G** function
- **Line**: Input of a line (**G1**)
- **Cir**: Description of a circular arc (**G2**, **G3**, **G12**, **G13**)
- **Form**: description of form elements
- **Front**: Functions for contour descriptions on the front face
- **Lateral**: Functions for contour descriptions on the lateral surface
- **ICP**, **Extras**, **Graph.**:
  Further information: "Shared menu items", Page 72



▶ Back to the DIN/ISO main menu

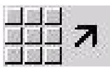
Machining menu item



The **Mach»** (machining) menu item contains functions for programming the machining operation. In **ISO Mode**, press the **Mach»** menu item to call the following menu items.

- **G**: Direct entry of a **G** function
- **G-menu**: Menu items for machining tasks
- **M**: Direct entry of an **M** function
- **M-menu**: Menu items for switching tasks
- **T**: Direct tool call
- **F**: Feed per revolution **G95**
- **S**: Cutting speed **G96**
- **Extras**, **Graph.**:
  Further information: "Shared menu items", Page 72

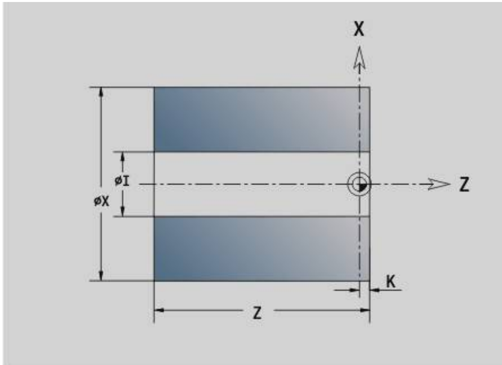
> Refer to your machine manual.
> Your machine tool builder can also provide his own **G**-code functions. You will find these functions under **Misc. functions** in the **G-menu**.



▶ Back to the ISO main menu

6.2 Definition of workpiece blank

Chuck part bar or tube G20-Geo



G20 defines the contour of a cylinder or hollow cylinder.

Parameters:

- **X**: **Diameter**
  - Cylinder/hollow cylinder diameter
  - Diameter of circumference of a polygonal blank
- **Z**: **Length** of workpiece blank
- **K**: **Righthnd edge** – distance between workpiece datum and right edge
- **I**: **Diam. inside**

Example: G20-Geo

. . .	
BLANK	
N1 G20 X80 Z100 K2 I30	
. . .	

Casting G21-Geo

G21 generates the contour of the blank part from the contour of the finished part—plus the equidistant **Ov.size P**.

Parameters:

- **P**: Equidistant **Allowance** (reference: finished part contour)
- **Q**: **Boring Y/N** (default: 0)
  - **0: No**
  - **1: Yes**

> **G21 cannot** be used to describe an auxiliary blank.

Example: G21-Geo

. . .	
BLANK	
N1 G21 P5 Q1	
. . .	
FINISHED PART	
N2 G0 X30 Z0	
N3 G1 X50 BR-2	
N4 G1 Z-40	
N5 G1 X65	
N6 G1 Z-70	
. . .	

6.3 Basic elements of turning contour

Starting point of turning contour G0–Geo

G0 defines the **Start point** of a turning contour.

Parameters:

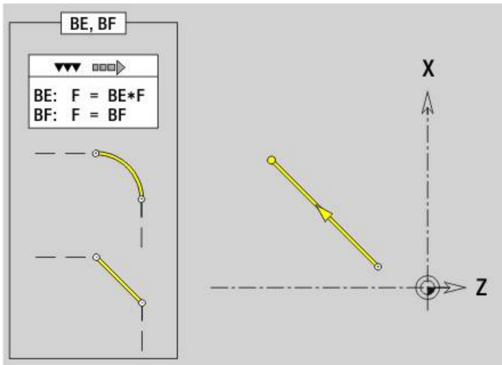
- **X**: Contour **Start point** (diameter value)
- **Z**: **Start point** of contour
- **PZ**: **Start point** (polar radius)
- **W**: **Start point** (polar angle)

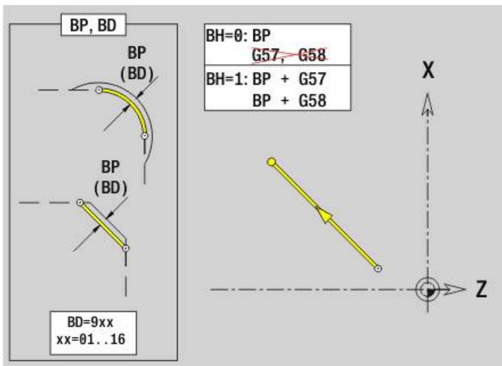
Example: G21-Geo

. . .	
FINISHED PART	
N2 G0 X30 Z0	
N3 G1 X50 BR-2	
N4 G1 Z-40	
N5 G1 X65	
N6 G1 Z-70	
. . .	

Machining attributes for form elements

All the basic contour elements contain the **Chamf./round.** form element **BR**. You can define machining attributes for this form element and all other form elements, such as recesses and undercuts.





Parameters:

- **BE**: **Special feed rate factor** for **Chamf./round.** (default: 1)
  Special feed rate = active feed rate * **BE** (range: 0 < **BE** <= 1)
- **BF**: **Feed per rot.** – special feed rate for **Chamf./round.** during the finishing cycle (default: no special feed rate)
- **BD**: **Corr. additiv** for **Chamf./round.** (range: 901 to 916)
- **BP**: Equidistant **Allowance** (at constant distance) for **Chamf./round.**
- **BH**: **Absolut=0,Add=1** – type of allowance for **Chamf./round.**
  - 0: Absolute oversize
  - 1: Additive oversize

Line segment in a turning contour G1–Geo

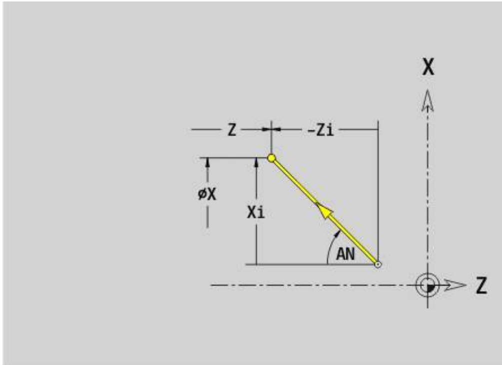
G1 defines a line segment in a turning contour.

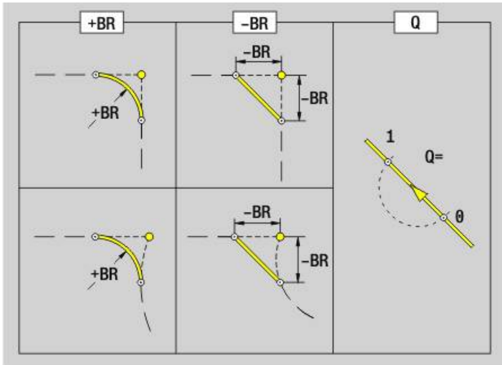
Parameters:

- **X**: **Final point** (diameter value)
- **Z**: **Final point**
- **AN**: **Angle** to rotary axis
- **Q**: **Intersect. pt.** or **Final point** if the line segment intersects a circular arc (default: 0)
  - 0: Near point of intersection
  - 1: Far point of intersection
- **BR**: **Chamf./round.** – defines the transition to the next contour element
  When entering a **Chamf./round.**, program the theoretical end point.
  - No entry: Tangential transition
  - **BR** = 0: No tangential transition
  - **BR** > 0: Rounding radius
  - **BR** < 0: Width of chamfer
- **PZ**: **Final point** (polar radius; reference: workpiece datum)
- **W**: **Final point** (polar angle; reference: workpiece datum)
- **AR**: **Incr.angle to foregoer ARi** (**AR** corresponds to **AN**)
- **R**: **Line length**
- **FP**: **Do not machine the element** (required only for **TURN PLUS**)
  - **1: Basic element** (line) do not machine
  - **2: Overlay element** (chamfer or rounding arc) do not machine
  - **3: Basic/overlay element** do not machine
- **IC**: **Measuring cut oversize**
- **KC**: **Length of measuring cut**
- **HC**: **Measuring cut counter** – Number of workpieces after which a measurement is performed

BE, **BF**, **BD**, **BP** and **BH**.

Further information: "Machining attributes for form elements", Page 288



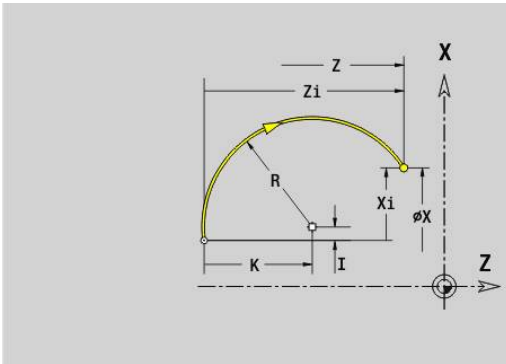


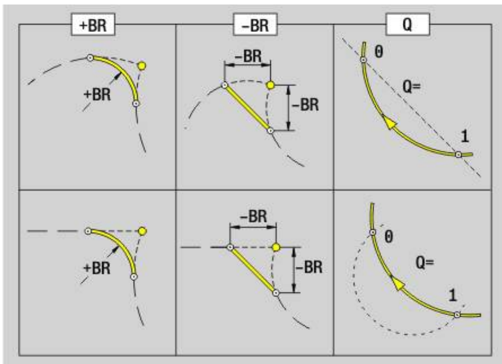
> Programming:
> - **X**, **Z**: Absolute, incremental, modal or **?**
> - **ARi**: Angle to the previous element
> - **ANi**: Angle to the subsequent element

Example: G1-Geo

. . .	
FINISHED PART	
N2 G0 X0 Z0	Starting point
N3 G1 X50 BR-2	Vertical line with chamfer
N4 G1 Z-20 BR2	Horizontal line with radius
N5 G1 X70 Z-30	Oblique cut with absolute target coordinates
N6 G1 Zi-5	Horizontal line segment, incremental
N7 G1 Xi10 AN30	Incremental and angle
N8 G1 X92 Zi-5	Incremental and absolute mixed
N9 G1 X? Z-80	Calculate the X coordinate
N10 G1 X100 Z-100 AN10	End point and angle with unknown starting point
. . .	

Circular arc of turning contour G2-/G3-Geo





G2 and **G3** define a circular arc in a contour with incremental center dimensioning.

Direction of rotation:

- **G2**: In clockwise direction
- **G3**: In counterclockwise direction

Parameters:

- **X**: **Final point** (diameter value)
- **Z**: **Final point**
- **R**: **Radius**
- **I**: **Center, incremental** – distance from starting point to center point (radius value)
- **K**: **Center, incremental** – distance from starting point to center point
- **Q**: **Intersect. pt.** or **Final point** if the line segment intersects a circular arc (default: 0)
  - 0: Near point of intersection
  - 1: Far point of intersection
- **BR**: **Chamf./round.** – defines the transition to the next contour element
  When entering a **Chamf./round.**, program the theoretical end point.
  - No entry: Tangential transition
  - **BR** = 0: No tangential transition
  - **BR** > 0: Rounding radius
  - **BR** < 0: Width of chamfer
- **FP**: **Do not machine the element** (required only for **TURN PLUS**)
  - **1: Basic element** (line) do not machine
  - **2: Overlay element** (chamfer or rounding arc) do not machine
  - **3: Basic/overlay element** do not machine

BE, **BF**, **BD**, **BP** and **BH**.

Further information: "Machining attributes for form elements", Page 288

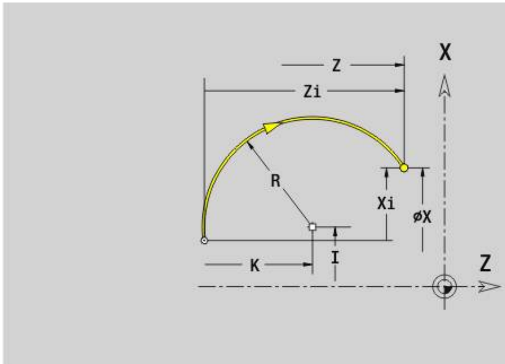
> Programming:
> - **X** and **Z** absolute, incremental, modal or **?**

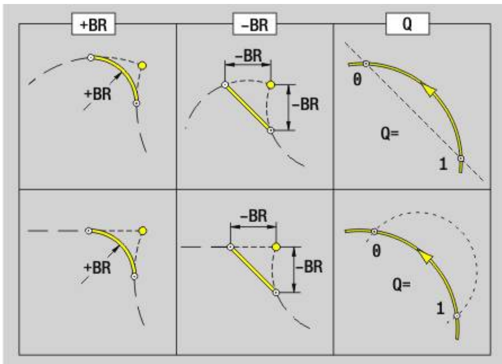
Example: G2, G3-Geo

. . .	
FINISHED PART	
N1 G0 X0 Z-10	
N2 G3 X30 Z-30 R30	Target point and radius
N3 G2 X50 Z-50 I19.8325 K-2.584	Target point and center, incremental
N4 G3 Xi10 Zi-10 R10	Target point (incremental) and radius
N5 G2 X100 Z? R20	Unknown target point coordinate
N6 G1 Xi-2.5 Zi-15	
. . .	

Circular arc of turning contour G12-/G13-Geo

G12 and **G13** define a circular arc in a contour with absolute center dimensioning.





Direction of rotation:

- **G12**: In clockwise direction
- **G13**: In counterclockwise direction

Parameters:

- **X**: **Final point** (diameter value)
- **Z**: **Final point**
- **I**: **Center** absolute (radius value)
- **K**: **Center** absolute
- **R**: **Radius**
- **Q**: **Intersect. pt.** or **Final point** if the line segment intersects a circular arc (default: 0)
  - 0: Near point of intersection
  - 1: Far point of intersection
- **BR**: **Chamf./round.** – defines the transition to the next contour element
  When entering a **Chamf./round.**, program the theoretical end point.
  - No entry: Tangential transition
  - **BR** = 0: No tangential transition
  - **BR** > 0: Rounding radius
  - **BR** < 0: Width of chamfer
- **PZ**: **Final point** (polar radius; reference: workpiece datum)
- **W**: **Final point** (polar angle; reference: workpiece datum)
- **PM**: **Center** (polar radius; reference: workpiece datum)
- **WM**: **Center** (polar angle; reference: workpiece datum)
- **AR**: **Start angle** – tangential angle to rotary axis
- **AN**: **Final angle** – tangential angle to rotary axis

- **FP**: **Do not machine the element** (required only for **TURN PLUS**)
  - **1: Basic element** (line) do not machine
  - **2: Overlay element** (chamfer or rounding arc) do not machine
  - **3: Basic/overlay element** do not machine

BE, **BF**, **BD**, **BP** and **BH**.

Further information: "Machining attributes for form elements", Page 288

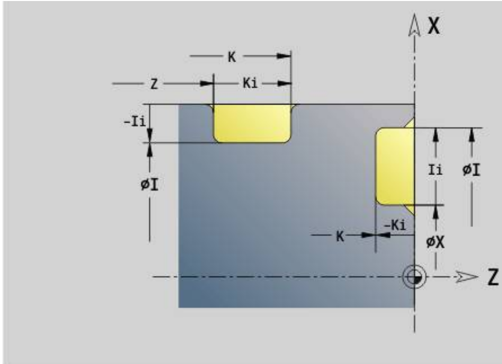
> Programming:
> - **X**, **Z**: Absolute, incremental, modal or **?**
> - **ARi**: Angle to the previous element
> - **ANi**: Angle to the subsequent element

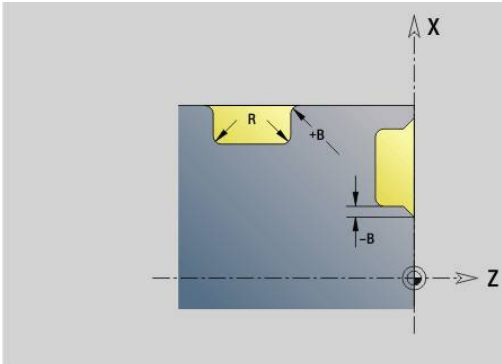
Example: G12, G13-Geo

. . .	
FINISHED PART	
N1 G0 X0 Z-10	
. . .	
N7 G13 Xi-15 Zi15 R20	Target point (incremental) and radius
N8 G12 X? Z? R15	Only the radius is known
N9 G13 X25 Z-30 R30 BR10 Q1	Rounding arc in transition and selection of intersections
N10 G13 X5 Z-10 I22.3325 K-12.584	Target point and center, absolute
. . .	

6.4 Form elements of turning contour

Recess (standard) G22–Geo





G22 defines a recess on the previously programmed paraxial reference element.

Parameters:

- **X**: **Start point** for recess on plane surface (diameter value)
- **Z**: **Start point** for recess on lateral surface
- **I**: **Inner corner** (diameter value)
  - Recess on face: End point of the recess
  - Recess on lateral surface: Recess base
- **Ii**: **Inner corner** incremental (pay attention to algebraic sign!)
  - Recess on face: Recess width
  - Recess on lateral surface: Recess depth
- **K**: **Inner corner**
  - Recess on face: Recess base
  - Recess on lateral surface: End point of the recess
- **Ki**: **Inner corner** incremental (pay attention to algebraic sign!)
  - Recess on face: Recess depth
  - Recess on lateral surface: Recess width
- **B**: **Outs.rad./cham.** at both sides of the recess (default: 0)
  - **B** > 0: Radius of rounding
  - **B** < 0: Width of chamfer
- **R**: **Inner radius** in both corners of the recess (default: 0)
- **FP**: **Do not machine the element** (required only for **TURN PLUS**)
  - **1: Yes**

BE, **BF**, **BD**, **BP** and **BH**.

Further information: "Machining attributes for form elements", Page 288

> Program only **X** or **Z** for the **Start point**.

Example: G22-Geo

. . .	
FINISHED PART	
N1 G0 X40 Z0	
N2 G1 X80	
N3 G22 X60 I70 Ki-5 B-1 R0.2	Recess on face, depth is incremental
N4 G1 Z-80	
N5 G22 Z-20 I70 K-28 B1 R0.2	Longitudinal recess, width is absolute
N6 G22 Z-50 Ii-8 Ki-12 B0.5 R0.3	Longitudinal recess, width is incremental
N7 G1 X40	
N8 G1 Z0	
N9 G22 Z-38 Ii6 K-30 B0.5 R0.2	Longitudinal recess, inside
. . .	

Recess (general) G23–Geo

G23 defines a recess on the previously programmed linear reference element. The reference element can also be oblique.

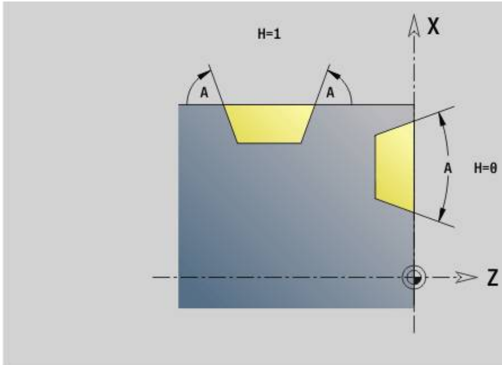
Parameters:

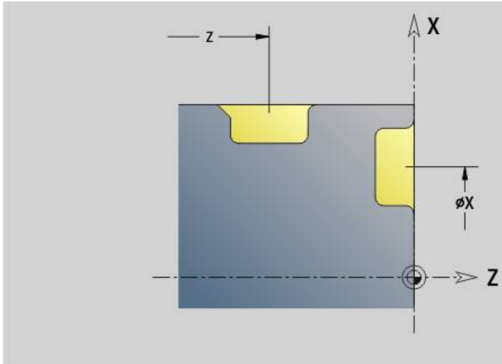
- **H**: **Kind of recess** (default: 0)
  - **0: Symmetrical recess**
  - **1: Relief turn**
- **X**: **Center** point for recess on plane surface (no input: position is calculated; diameter value)
- **Z**: **Center** point for recess on lateral surface (no input: position is calculated)
- **I**: **Depth**
  - **I** > 0: Recess at the right of the reference element
  - **I** < 0: Recess at the left of the reference element
- **K**: **Width** (without **Chamf./round.**)
- **U**: **Recess diameter** – diameter of recess base
  Use **U** only if the reference element runs parallel to the Z axis.
- **A**: **Angle** (default: 0°)
  - **H** = 0: Angle between recess edges (range: 0° <= **A** < 180°)
  - **H** = 1: Angle between reference line and recess edge (range: 0° < **A** <= 90°)
- **B**: **Outs.rad./cham.** at near corner relative to starting point (default: 0)
  - **B** > 0: Radius of rounding
  - **B** < 0: Width of chamfer
- **P**: **Outs.rad./cham.** at far corner relative to starting point (default: 0)
  - **P** > 0: Radius of the rounding arc
  - **P** < 0: Chamfer width
- **R**: **Inner radius** in both corners of the recess (default: 0)
- **FP**: **Do not machine the element** (required only for **TURN PLUS**)
  - **1: Yes**

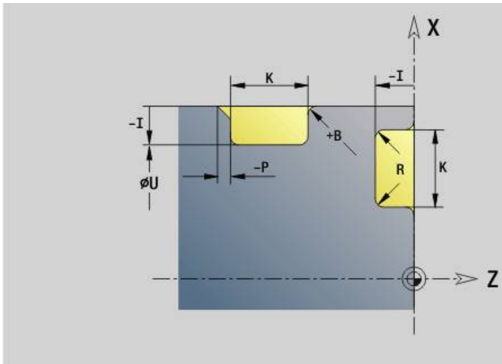
BE, **BF**, **BD**, **BP** and **BH**.

Further information: "Machining attributes for form elements", Page 288

> **Depth** always references the reference element. The recess base runs parallel to the reference element.



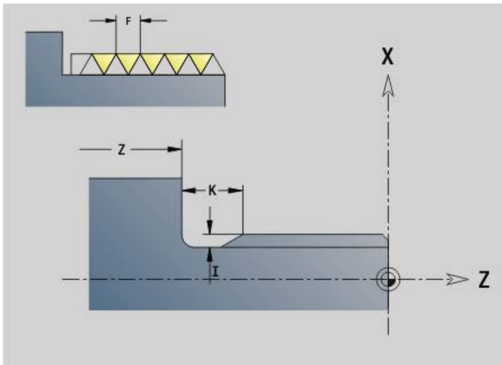




Example: G23-Geo

. . .	
FINISHED PART	
N1 G0 X40 Z0	
N2 G1 X80	
N3 G23 H0 X60 I-5 K10 A20 B-1 P1 R0.2	Recess on face, depth is incremental
N4 G1 Z-40	
N5 G23 H1 Z-15 K12 U70 A60 B1 P-1 R0.2	Longitudinal recess, width is absolute
N6 G1 Z-80 A45	
N7 G23 H1 X120 Z-60 I-5 K16 A45 B1 P-2 R0.4	Longitudinal recess, width is incremental
N8 G1 X40	
N9 G1 Z0	
N10 G23 H0 Z-38 I-6 K12 A37.5 B-0.5 R0.2	Longitudinal recess, inside
. . .	

Thread with undercut G24–Geo



G24 defines a linear basic element with a longitudinal thread and subsequent thread undercut (DIN 76). The thread is an outside or inside thread (metric ISO fine-pitch thread DIN 13 Part 2, Series 1).

Parameter:

- **F**: **Thread pitch**
- **I**: **Undercut depth**
- **K**: **Underc. breadth**
- **Z**: **Final point** of the undercut
- **FP**: **Do not machine the element** (required only for **TURN PLUS**)
  - **1: Yes**

BE, **BF**, **BD**, **BP** and **BH**.

Further information: "Machining attributes for form elements", Page 288

- Program **G24** only in closed contours
- The thread is machined with **G31**

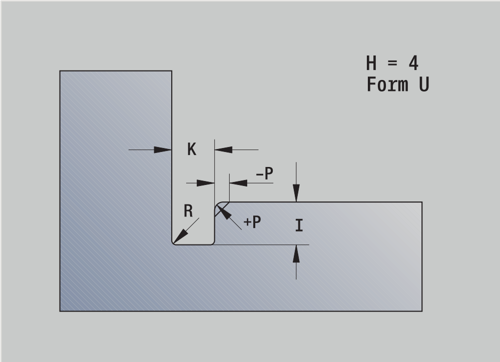
Example: G24-Geo

. . .	
FINISHED PART	
N1 G0 X40 Z0	
N2 G1 X40 BR-1.5	Starting point for thread
N3 G24 F2 I1.5 K6 Z-30	Thread with undercut
N4 G1 X50	Next transverse element
N5 G1 Z-40	
. . .	

Undercut G25–Geo

G25 generates the undercut contours listed below. The undercuts are only possible in inside contour corners in which the planar element is parallel to the X axis. Program **G25** after the first element. Specify the **Cut type** in the **H** parameter.

Undercut type U (H=4)



Parameter:

- **H**: **Cut type** U (**H** = 4)
- **I**: **Undercut depth**
- **K**: **Underc. breadth**
- **R**: **Radius** – **Inner radius** in both corners of the recess (default: 0)
- **P**: **Trav.dpth** – **outside radius** or **Chamfer** (default: 0)
  - **P** > 0: Radius of the rounding arc
  - **P** < 0: Chamfer width
- **FP**: **Do not machine the element** (required only for **TURN PLUS**)
  - **1: Yes**

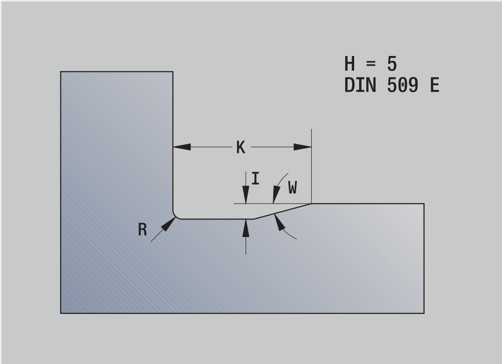
BE, **BF**, **BD**, **BP** and **BH**.

Further information: "Machining attributes for form elements", Page 288

Example: Call G25-Geo type U

. . .	
N.. G1 Z-15	Longitudinal element
N.. G25 H4 I2 K4 R0.4 P-0.5	Type U
N.. G1 X20	Transverse element
. . .	

Undercut DIN 509 E (H=0.5)



Parameter:

- **H**: **Cut type** DIN 509 E (**H** = 0 or **H** = 5)
- **I**: **Undercut depth**
- **K**: **Underc. breadth**
- **R**: **Radius** in the undercut corner
- **W**: **Angle** – **Undercut angle**

BE, **BF**, **BD**, **BP** and **BH**.

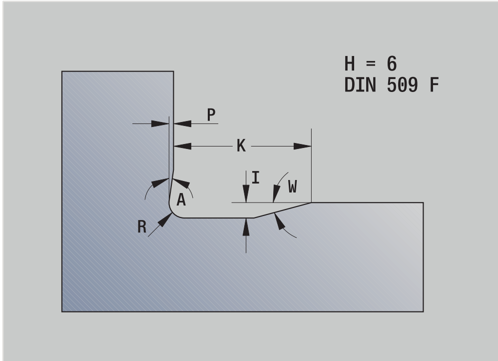
Further information: "Machining attributes for form elements", Page 288

> The control uses the diameter to calculate the parameters that you do not define.

Example: Call G25-Geo DIN 509 E

. . .	
N.. G1 Z-15	Longitudinal element
N.. G25 H5	DIN 509 E
N.. G1 X20	Transverse element
. . .	

Undercut DIN 509 F (H=6)



Parameter:

- **H**: **Cut type** DIN 509 F (**H** = 6)
- **I**: **Undercut depth**
- **K**: **Underc. breadth**
- **R**: **Radius** in the undercut corner
- **P**: **Trav.dpth**
- **W**: **Angle** – **Undercut angle**
- **A**: **Angle** – **Trav.angle**

BE, **BF**, **BD**, **BP** and **BH**.

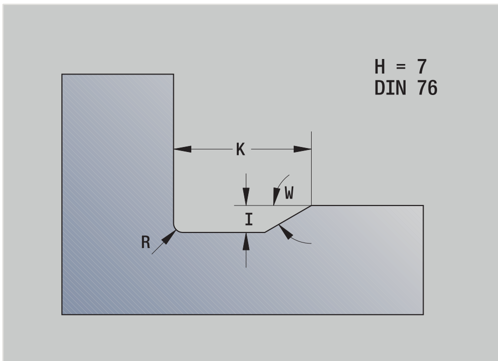
Further information: "Machining attributes for form elements", Page 288

> The control uses the diameter to calculate the parameters that you do not define.

Example: Call G25-Geo DIN 509 F

. . .	
N.. G1 Z-15	Longitudinal element
N.. G25 H6	DIN 509 F
N.. G1 X20	Transverse element
. . .	

Undercut DIN 76 (H=7)



Program only **FP**. All other values that have not been programmed are automatically taken from the standard table, derived from the **Thread pitch** you specified.

Parameter:

- **H**: **Cut type** DIN 76 (**H** = 7)
- **I**: **Undercut depth**
- **K**: **Underc. breadth**
- **R**: **Radius** in the undercut corner (default: **R** = 0.6 * **I**)
- **W**: **Angle** – **Undercut angle** (default: 30°)
- **FP**: **Thread pitch**

BE, **BF**, **BD**, **BP** and **BH**.

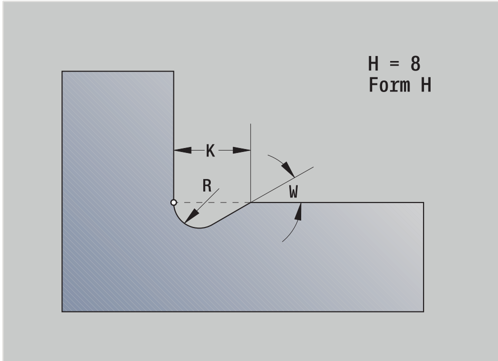
Further information: "Machining attributes for form elements", Page 288

Example: Call G25-Geo DIN 76

. . .	
N.. G1 Z-15	Longitudinal element
N.. G25 H7 FP2	DIN 76
N.. G1 X20	Transverse element
. . .	

Undercut form H (H=8)

If you do not enter **W**, the **Angle** will be calculated on the basis of **K** and **R**. The final point of the undercut is then located at the **Contour corner**.



Parameter:

- **H**: **Cut type** H (**H** = 8)
- **K**: **Underc. breadth**
- **R**: **Radius** – **Undercut radius** (no input: the circular element is not machined)
- **W**: **Angle** – **Undercut angle**

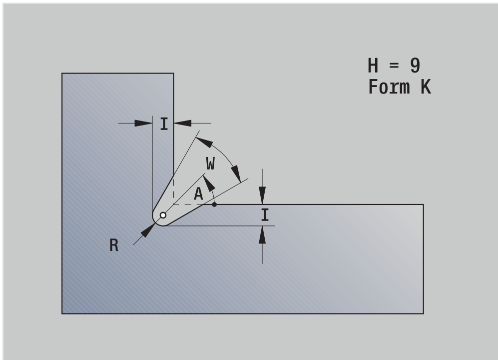
BE, **BF**, **BD**, **BP** and **BH**.

Further information: "Machining attributes for form elements", Page 288

Example: Call G25-Geo Form H

. . .	
N.. G1 Z-15	Longitudinal element
N.. G25 H8 K4 R1 W30	Type H
N.. G1 X20	Transverse element
. . .	

Undercut form K (H=9)



Parameter:

- **H**: **Cut type** K (**H** = 9)
- **I**: **Undercut depth**
- **R**: **Radius** – **Undercut radius** (no input: the circular element is not machined)
- **W**: **Angle** – **Undercut angle**
- **A**: **Angle** to longitudinal axis (default: 45°)

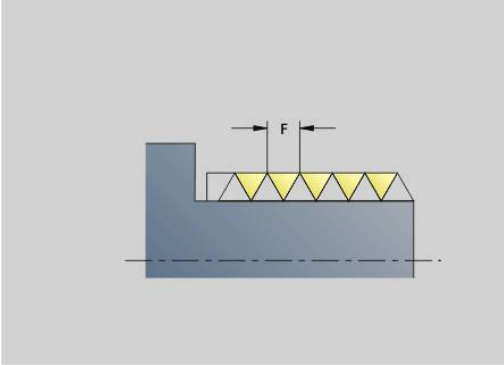
BE, **BF**, **BD**, **BP** and **BH**.

Further information: "Machining attributes for form elements", Page 288

Example: Call G25-Geo type K

. . .	
N.. G1 Z-15	Longitudinal element
N.. G25 H9 I1 R0.8 W40	Type K
N.. G1 X20	Transverse element
. . .	

Thread (standard) G34–Geo



G34 defines a simple or an interlinked external or internal thread (metric ISO fine-pitch thread DIN 13 Series 1). The control calculates all the required values.

Parameter:

- **F**: **Thread pitch**

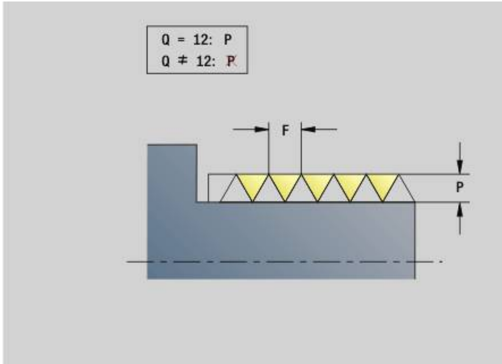
Threads are concatenated by programming several **G1**/**G34** blocks after each other.

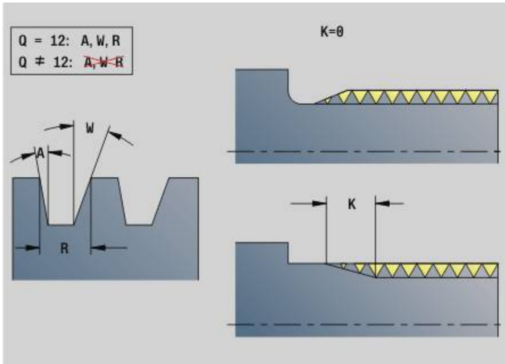
> - You need to program a linear contour element as a reference before **G34** or in the NC block containing **G34**
> - Machine the thread with **G31**

Example: G34

...	
FINISHED PART	
N1 G0 X0 Z0	
N2 G1 X20 BR-2	
N3 G1 Z-30	
N4 G34	Metric ISO
N5 G25 H7 I1.7 K7	
N6 G1 X30 BR-1.5	
N7 G1 Z-40	
N8 G34 F1.5	Metric ISO fine-pitch thread
N9 G25 H7 I1.5 K4	
N10 G1 X40	
N11 G1 Z-60	
...	

Thread (general) G37–Geo





G37 defines the different types of thread. Multi-start threads and concatenated threads are possible. Threads are concatenated by programming several **G01**/**G37** blocks after each other.

Parameter:

- **Q**: **Sort of thread** (default: 1)
  - **1: ISO fine DIN 13**
  - **2: ISO DIN 13**
  - **3: Taper DIN 158**
  - **4: Fine taper DIN 158**
  - **5: ISO trapezoid DIN 103**
  - **6: Trapezoid DIN 380**
  - **7: Sawing DIN 513**
  - **8: Round DIN 405**
  - **9: Cylindrical DIN 11**
  - **10: Taper DIN 2999**
  - **11: Pipe DIN 259**
  - **12: Not standardized**
  - **13: UNC US coarse**
  - **14: UNF US fine-pitch**
  - **15: UNEF US extra-fine-pitch**
  - **16: NPT US taper pipe**
  - **17: NPTF US Dryseal pipe**
  - **18: NPSC US pipe** (with lubricant)
  - **19: NPFS US pipe** (without lubricant)
  - **20: Helical slot**
- **F**: **Thread pitch**
  - Required for **Q** = 1, 3-7, 12
  - For other thread types, **F** is calculated from the diameter if it was not programmed
- **P**: **Thread depth** (only with **Q** = 12)
- **K**: **End. length** for threads without undercut (default: 0)
- **D**: **Reference pt.** (default: 0)
  - 0: Runout of thread at the end of the reference element
  - 1: Runout of thread at the beginning of the reference element
- **H**: **No.threads** (default: 1)
- **A**: **Flank left** – enter the thread angle at left (only for **Q** = 12)
- **W**: **Flank right** – enter the thread angle at right (only for **Q** = 12)
- **R**: **Width** (enter only for **Q** = 12)
- **E**: **Variable gr.** (default: 0)
  Increases/decreases the pitch per revolution by **E**.
- **V**: **Direction of thread:**
  - **0: Right-hand thread**
  - **1: Left-hand thread**

- Before **G37**, program a linear contour element as a reference
- Machine the thread with **G31**
- For standard threads, the parameters **P**, **R**, **A** and **W** are defined by the control
- Use **Q**=12 if you wish to use individual parameters

NOTICE

Danger of collision!

The thread is generated to the length of the reference element. The control does not check for collisions with the workpiece contour (e.g. contour of the finished part). Danger of collision during machining!

- Another linear element without undercut is to be programmed as overrun.

Example: G37

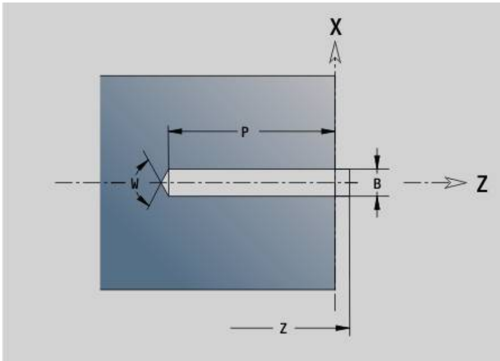
. . .	
FINISHED PART	
N1 G0 X0 Z0	
N2 G1 X20 BR-2	
N3 G1 Z-30	
N4 G37 Q2	Metric ISO
N5 G25 H7 I1.7 K7	
N6 G1 X30 BR-1.5	
N7 G1 Z-40	
N8 G37 F1.5	Metric ISO fine-pitch thread
N9 G25 H7 FP1.5	
N10 G1 X40	
N11 G1 Z-60	
. . .	

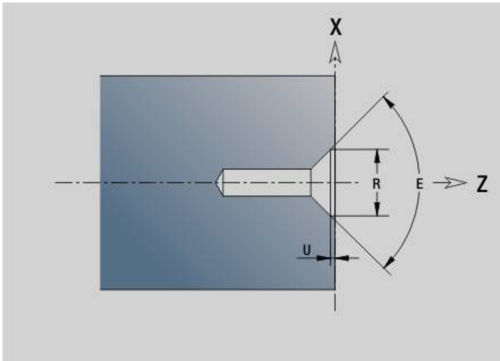
Example: G37 Concatenated

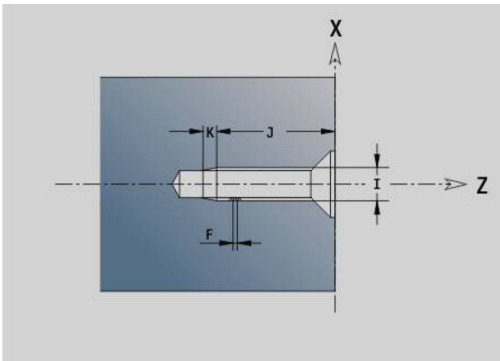
. . .	
AUXILIARY CONTOUR ID"G37_Kette"	
N37 G0 X0 Z0	
N 38 G1 X20	
N 39 G1 Z-30	
N 40 G37 F2	Metric ISO
N 41 G1 X30 Z-40	
N 42 G37 Q2	
N 43 G1 Z-70	
N 44 G37 F2	
. . .	

Bore hole (centric) G49–Geo

G49 defines a single hole with countersink and thread at the turning center (front or rear face). The **G49** hole is a form element, not part of the contour.







Parameter:

- **Z**: **Position** Starting position for hole (reference point)
- **B**: **Diameter**
- **P**: **Depth** excluding point
- **W**: **Point angle** (default: 180°)
- **R**: **Sink diam.**
- **U**: **Sink depth**
- **E**: **Sink angle**
- **I**: **Thread diameter**
- **J**: **Thread depth**
- **K**: **Start of thread** – run-out length
- **F**: **Thread pitch**
- **V**: **Direction of thread:** (default: 0)
  - **0: Right-hand thread**
  - **1: Left-hand thread**
- **A**: **Angle** – position of the first hole (default: 0°)
  - **A** = 0°: Front face
  - **A** = 180°: Rear side
- **O**: **Center. diam.**

- Program **G49** in the **FINISHED PART** section, and not in **AUXILIARY CONTOUR**, **FRONT**, or **REAR SIDE**
- Machine the **G49** hole with **G71**..**G74**

6.5 Attributes for contour description

Overview of attributes for contour description

G function	Description of function	Page
G10	Surface roughness for basic contour elements – modal	Page 304
G38	**Special feed rate factor** for basic elements and form elements – modal	Page 305
G52	Equidistant **Allowance** for basic elements and form elements – modal	Page 306
G95	Finishing feed rate for basic elements and form elements—modal	Page 307
G149	**Additive correction** for basic elements and form elements – modal	Page 308

- Once programmed, **G10**, **G38**, **G52**, **G95**, and **G149**-Geo remain in effect for **all contour elements** until the function is programmed again without parameter definition
- For form elements, you can program different attributes directly in the definition of the form element
  Further information: "Machining attributes for form elements", Page 288
- The **attributes for contour description** influence the finishing feed rate of the cycles **G869** and **G890**, not the finishing feed rate in recessing cycles.

Surface roughness G10–Geo

G10 influences the finishing feed rate of **G890**. The surface roughness (peak-to-valley height) applies only to basic contour elements.

Parameters:

- **H**: **Kind of rough.** – peak-to-valley height (DIN 4768)
  - H = **1**: General surface roughness (total height of the profile) **Rt1**
  - H = **2**: Arithmetic mean roughness **Ra**
  - H = **3**: Maximum height of roughness profile **Rz**
- **RH**: **Roughness**

- **G10** is a modal function
- Program **G10** or **G95** without any parameters to switch the surface roughness function off
- **G10 RH...** overwrites the surface roughness block by block.
- **G38** overwrites the surface roughness block by block.

Feed reduction G38-Geo

G38 activates the **Sp. Feed Fact.** for the finishing cycle **G890**. The **Sp. Feed Fact.** applies to basic contour elements and form elements. It is a modal function.

Parameter:

- **E**: **Special feed rate factor** (default: 1)
  Special feed rate = active feed rate * **E**

> - **G38** is a modal function
> - Program **G38** before the contour element for which it is intended
> - **G38** replaces a **Sp. Feed Fact.**
> - To cancel the special feed factor, program **G38** without parameters.

Attributes for superimposed elements G39-Geo

G39 influences the finishing feed rate of **G890** with the form elements:

- Chamfers/rounding arcs (for connecting basic elements)
- Undercuts
- Recesses

Affected machining:

- **Sp. Feed Fact.**
- **Roughness**
- Additive D compensation
- Equidistant **Ov.size**

Parameter:

- **F**: **Feed per revolution**
- **V**: **Kind of rough.** – Peak-to-valley height (DIN 4768)
  - **1**: General surface roughness (profile depth) **Rt1**
  - **2**: Surface roughness **Ra**
  - **3**: Surface roughness **Rz**
- **RH**: **Roughness** (µm, inch mode: µinch)
- **D**: **Corr. additiv** (range: 901 <= **D** <= 916)
- **P**: **Allowance** (radius value)
- **H**: **Absolut=0,Add=1** – **P** applies as an absolute or additive value (default: 0)
  - **0**: **P** replaces **G57**-/**G58** oversizes
  - **1**: **P** is added to **G57**-/**G58** oversizes
- **E**: **Special feed rate factor** (default: 1)
  Special feed rate = active feed rate * **E**

- Use **Kind of rough. V**, **Roughness RH**, **Feed per rotation F** and special feed rate **E** alternatively!
- **G39** is a non-modal function
- Program G39 **before** the contour element for which it is intended
- **G50** preceding a cycle (**MACHINING** section) cancels oversizes programmed for that cycle with **G39**

Function **G39** can be replaced by directly entering the attributes in the contour elements dialog. The function is necessary to execute imported programs correctly.

Separation point G44

During automatic program creation with **TURN PLUS**, you can define the **Separation point** for rechucking with function **G44**.

Parameter:

- **D**: **Location of separation point**
  - **0: Start of basic element**
  - **1: Target of basic element**

If no **Separation point** was defined, **TURN PLUS** uses the largest diameter as separation point for outside machining and the smallest diameter as **Separation point** for inside machining.

Oversize G52-Geo

G52 defines a contour-parallel **Ov.size** that applies to basic contour elements and form elements and is taken into account in **G810**, **G820**, **G830**, **G860**, and **G890**.

Parameter:

- **P**: **Allowance** (radius value)
- **H**: **Absolut=0,Add=1** – **P** applies as an absolute or additive value (default: 0)
  - **0**: **P** replaces **G57**-/**G58** oversizes
  - **1**: **P** is added to **G57**-/**G58** oversizes

- **G52** is a modal function
- Program **G52** in the NC block for which it is intended.
- **G50** preceding a cycle (**MACHINING** section) cancels an oversize programmed for that cycle with **G52**

Feed per revolution G95-Geo

G95 influences the finishing feed rate of **G890** for basic contour elements and form elements.

Parameter:

- **F**: **Feed per revolution**

> - The **G95** finishing feed rate replaces a finishing feed rate defined in the machining section.
> - **G95** is modal
> - To cancel a finishing feed rate set with **G95**, program G95 without an input value.
> - **G10** disables the finishing feed rate **G95**

Example: Attributes in contour description G95

. . .	
FINISHED PART	
N1 G0 X0 Z0	
N2 G1 X20 BR-1	
N3 G1 Z-20	
N4 G25 H5 I0.3 K2.5 R0.6 W15	
N5 G1 X40 BR-1	
N6 G95 F0.08	
N7 G1 Z-40	
N8 G25 H5 I0.3 K2.5 R0.6 W15 BF0	
N9 G95	
N10 G1 X58 BR-1	
N11 G1 Z-60	
. . .	

Additive correction G149-Geo

G149 followed by a **D number** activates or deactivates **Additive correction**. The control manages 16 tool-independent compensation values in an internal table. The compensation values are managed in **Program run** submode.

Further information: User's Manual

Parameter:

- **D**: **Corr. additiv** (default: 900)
  - **D** = 900: Deactivates the additive compensation
  - **D** = 901-916: Activates the additive compensation **D**

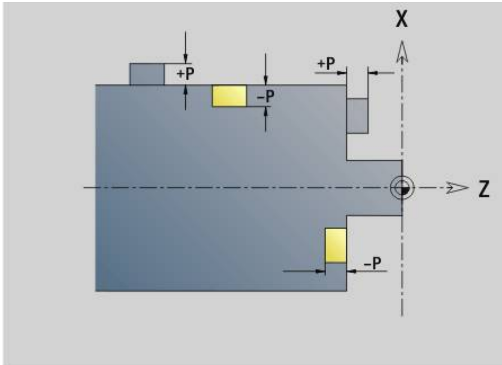
> - Note the direction of contour description
> - **Additive correction** is effective from the block in which **G149** is programmed
> - An **Additive correction** remains in effect until:
>   - The next **G149 D900**
>   - Up to the end of the finished part description

Example: Attributes in contour description G149

. . .	
FINISHED PART	
N1 G0 X0 Z0	
N2 G1 X20 BR-1	
N3 G1 Z-20	
N4 G25 H5 I0.3 K2.5 R0.6 W15	
N5 G1 X40 BR-1	
N6 G149 D901	
N7 G1 Z-40	
N8 G25 H5 I0.3 K2.5 R0.6 W15 BD900	
N9 G149 D900	
N10 G1 X58 BR-1	
N11 G1 Z-60	
. . .	

6.6 C-axis contours—fundamentals

Position of milling contours



Define the **reference plane** or the **Reference diameter** in the section code.

Specify the **Depth** and **Position** of a milling contour (pocket, island) in the contour definition:

- With **Depth/Height P** in the previously programmed **G308** cycle
- Alternatively on figures: Cycle parameter **Depth P**

The algebraic sign of **P** defines the **Position** of the milling contour:

- **P** < 0: Pocket
- **P** > 0: Island

Position of milling contour

Section	P	Surface	Milling floor
FRONT	**P** < 0 **P** > 0	**Z** **Z + P**	**Z + P** **Z**
REAR SIDE	**P** < 0 **P** > 0	**Z** **Z - P**	**Z - P** **Z**
LATERAL	**P** < 0 **P** > 0	**X** **X + (P * 2)**	**X + (P * 2)** **X**

- **X**: **Reference diameter** from the section code
- **Z**: **Reference plane** from the section code
- **P**: **Depth/Height** from **G308** or from cycle parameter

> The area milling cycles mill the surface specified in the contour definition. **Islands** within this surface are not taken into consideration.

Contours in more than one plane (hierarchically nested contours):

- A plane begins with **G308** and ends with **G309**
- **G308** defines a new **reference plane**/**Reference diameter**. The first **G308** uses the **reference plane** defined in the section code. Each following **G308** defines a new plane. Calculation: New **reference plane** = **Reference plane** + **P** (from previous **G308**)
- **G309** switches back to the previous reference plane.

Beginning of pocket/island G308-Geo

G308 defines a new **reference plane** or **Reference diameter** in hierarchically nested contours.

Parameter:

- **ID**: **Milling contour** – name of the milling contour
- **P**: **Depth/Height** – depth for pockets, height for islands
- **HC**: **Milling/drilling attribute**
  - **1: Contour milling G840/G847**
  - **2: Pocket milling G845/G848**
  - **3: Area milling G841-G844**
  - **4: Deburring G840**
  - **5: Engraving G801-G804**
  - **6: Contour + deburring G840/G847**
  - **7: Pocket + deburring G845/G848**
  - **8: face milling G797**
  - **9: face milling + deburring G797**
  - **10: bore milling G75**
  - **11: thread milling G799/G800/G806**
  - **12: bore and thread milling G75/G799..**
  - **14: Do not machine**
- **D**: **Cutter diameter**
- **Q**: **Cutter position**
  - **0: On the contour**
  - **1: Inside / left**
  - **2: Outside / right**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **O**: **Plunging behavior** (default: 0)
  - 0 / no input – **vertical plunging**
  - **1: Helical plunging**
    - In roughing cycles, the tool plunges in a reciprocating movement when milling pockets, otherwise—i.e. when milling slots—in a helical movement.
    - In finishing cycles, the tool plunges in a 3-D approach arc when milling pockets.
  - **2: Reciprocating plunge**
    - In roughing cycles, the tool plunges in a reciprocating movement when milling pockets.
    - In finishing cycles, the tool plunges in a 3-D approach arc when milling pockets.
- **I**: **Limit diameter**
- **RC**: **Trochoid width**
- **RB**: **Return plane**
- **W**: **Angle**
  - Angle of the chamfer
  - If **HC = 5**: Tool angle
- **BR**: **Chamfer width**

End of pocket/island G309-Geo

G309 defines the end of a **reference plane**. Every reference plane defined with **G308** must be ended with **G309**.

Further information: "Position of milling contours", Page 309

Example: G308/G309

. . .	
FINISHED PART	
. . .	
FRONT Z0	Define reference plane
N7 G308 ID“Rechteck“ P-5 O1	Starting point of rectangle with depth –5 and helical plunge
N8 G305 XK-5 YK-10 K50 B30 R3 A0	Rectangle
N9 G308 ID“Kreis“ P-10 O1	Starting point of full circle in rectangle with depth –10 and helical plunge
N10 G304 XK-3 YK-5 R8	Full circle
N11 G309	End of full circle
N12 G309	End of rectangle
LATERAL X100	Define reference diameter
N13 G311 Z-10 C45 A0 K18 B8 P-5	Linear slot with depth –5
. . .	

Circular pattern with circular slots

For circular slots in circular patterns you program the pattern positions, the center of curvature, the curvature radius and the **position** of the slots.

The control positions the slots as follows:

- Slots are arranged at the distance of the **pattern radius** about the **pattern center** if
  - Pattern center = Center of curvature **and**
  - Pattern radius = curvature radius
- Slots are arranged at the distance of the **pattern radius + curvature radius** about the **pattern center** if
  - Pattern center <> center of curvature **or**
  - Pattern radius <> curvature radius

In addition, the **position** influences the arrangement of the slots:

- **Normal position:**
  - The starting angle of the slot applies as a **relative value** to the pattern position
  - The starting angle is added to the pattern position
- **Original position:**
  - The starting angle of the slot applies as an **absolute value**

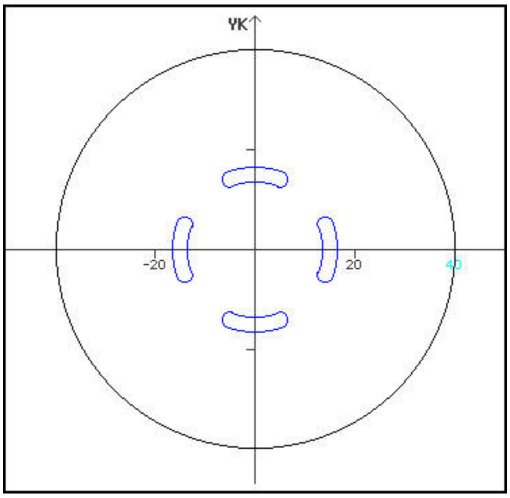
The following examples show the programming of a circular pattern with circular slots.

Slot centerline as reference and normal position

Programming:

- Pattern center = center of curvature
- Pattern radius = curvature radius
- Normal position

These commands arrange the slots at the distance of the **pattern radius** about the pattern center.



Example: Slot centerline as reference, normal position

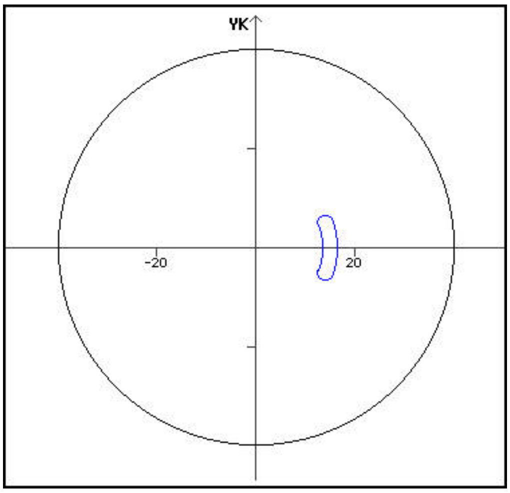
N.. G402 Q4 K30 A0 XK0 YK0 H0	Circular pattern, normal position
N.. G303 I0 J0 R15 A-20 W20 B3 P1	Circular slot

Slot centerline as reference and original position

Programming:

- Pattern center = center of curvature
- Pattern radius = curvature radius
- Original position

These commands arrange all slots at the same position.



Example: Slot centerline as reference, original position

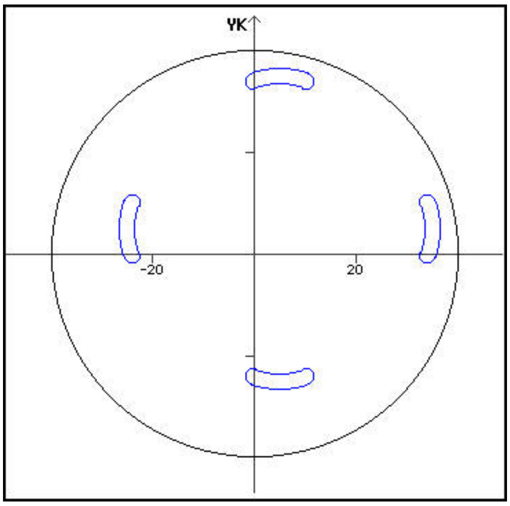
N.. G402 Q4 K30 A0 XK0 YK0 H1	Circular pattern, original position
N.. G303 I0 J0 R15 A-20 W20 B3 P1	Circular slot

Center of curvature as reference and normal position

Programming:

- Pattern center <> center of curvature
- Pattern radius = curvature radius
- Normal position

These commands arrange the slots at the distance of the **pattern radius plus curvature radius** about the pattern center.



Example: Center of curvature as reference, normal position

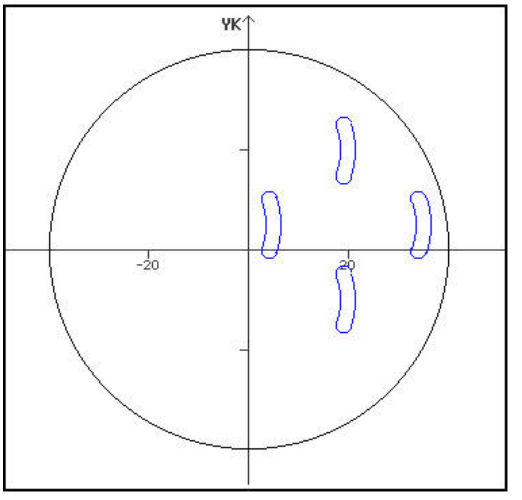
N.. G402 Q4 K30 A0 XK5 YK5 H0	Circular pattern, normal position
N.. G303 I0 J0 R15 A-20 W20 B3 P1	Circular slot

Center of curvature as reference and original position

Programming:

- Pattern center <> center of curvature
- Pattern radius = curvature radius
- Original position

These commands arrange the slots at the distance of the **pattern radius plus curvature radius** about the pattern center while keeping the starting and ending angle.



Example: Center of curvature as reference and original position

N.. G402 Q4 K30 A0 XK5 YK5 H1	Circular pattern, original position
N.. G303 I0 J0 R15 A-20 W20 B3 P1	Circular slot

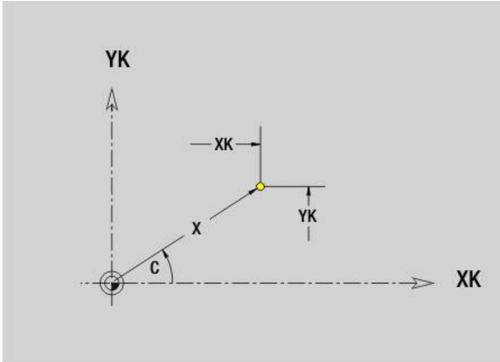
6.7 Front/Rear face contours

Starting point of front/rear face contour G100-Geo

G100 defines the **Start point** of a front or rear face contour.

Parameter:

- **X**: **Start point** (in polar coordinates)
- **C**: **Start angle** (angle, in polar coordinates)
- **XK**: **Start point** (in Cartesian coordinates)
- **YK**: **Start point** (in Cartesian coordinates)

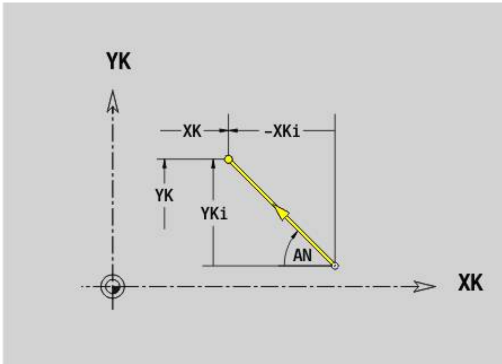


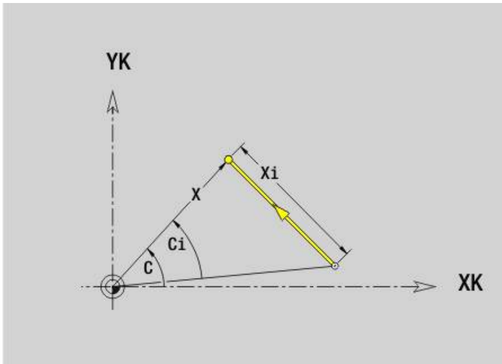
Line segment in front/rear face contour G101-Geo

G101 defines a line segment in a contour on the front face/rear face.

Parameter:

- **X**: **Final point** (in polar coordinates; diameter value)
- **C**: **End angle** (in polar coordinates)
- **XK**: **Final point** (in Cartesian coordinates)
- **YK**: **Final point** (in Cartesian coordinates)
- **AN**: **Angle** to positive XK axis
- **Q**: **Intersect. pt.** or **Final point** if the line segment intersects a circular arc (default: 0)
  - 0: Near point of intersection
  - 1: Far point of intersection
- **BR**: **Chamf./round.** – defines the transition to the next contour element
  When entering a **Chamf./round.**, program the theoretical end point.
  - No entry: Tangential transition
  - **BR** = 0: No tangential transition
  - **BR** > 0: Rounding radius
  - **BR** < 0: Width of chamfer
- **AR**: **Incr.angle to foregoer ARi** (**AR** corresponds to **AN**)
- **R**: **Line length**





> Programming:
> - **XK**, **YK**: Absolute, incremental, modal or **?**
> - **X**, **C**: Absolute, incremental, modal, or **?**
> - **ARi**: Angle to the previous element
> - **ANi**: Angle to the subsequent element

Circular arc in front/rear face contour G102-/G103-Geo

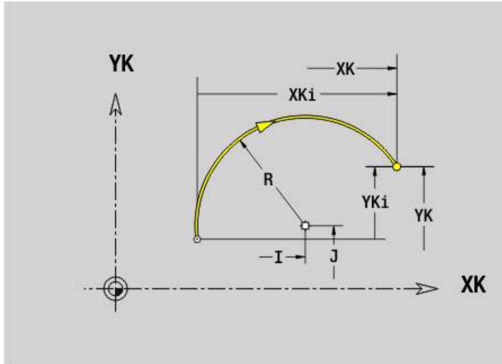
G102 and **G103** define a circular arc in a front or rear face contour.

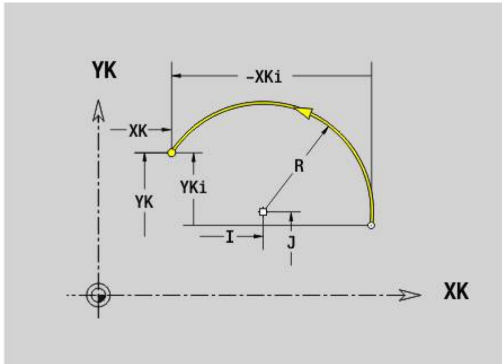
Direction of rotation:

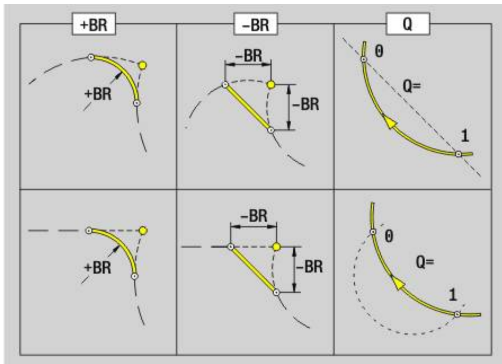
- **G102**: In clockwise direction
- **G103**: In counterclockwise direction

Parameter:

- **X**: **Final point** (in polar coordinates; diameter value)
- **C**: **End angle** (in polar coordinates)
- **XK**: **Final point** (in Cartesian coordinates)
- **YK**: **Final point** (in Cartesian coordinates)
- **R**: **Radius**
- **I**: **Center** (in Cartesian coordinates)
- **J**: **Center** (in Cartesian coordinates)
- **Q**: **Intersect. pt.** or **Final point** if the circular arc intersects a line segment or another circular arc (default: 0)
  - 0: Near point of intersection
  - 1: Far point of intersection
- **BR**: **Chamf./round.** – defines the transition to the next contour element
  When entering a **Chamf./round.**, program the theoretical end point.
  - No entry: Tangential transition
  - **BR** = 0: No tangential transition
  - **BR** > 0: Rounding radius
  - **BR** < 0: Width of chamfer
- **XM**: **Center** (polar radius; reference: workpiece datum)
- **CM**: **Center** – polar angle (reference: workpiece datum)
- **AR**: **Start angle** – tangential angle to rotary axis
- **AN**: **Final angle** – tangential angle to rotary axis







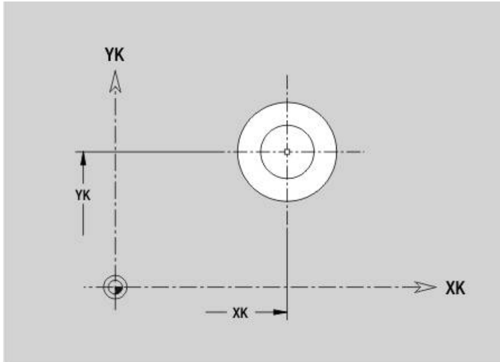
> Programming:
> - **XK**, **YK**: Absolute, incremental, modal or **?**
> - **X**, **C**: Absolute, incremental, modal, or **?**
> - **I**, **J**: Absolute, incremental or **?**
> - **XM**, **CM**: Absolute or incremental
> - **ARi**: Angle to the previous element
> - **ANi**: Angle to the subsequent element
>
> End point must not be the starting point (no full circle).

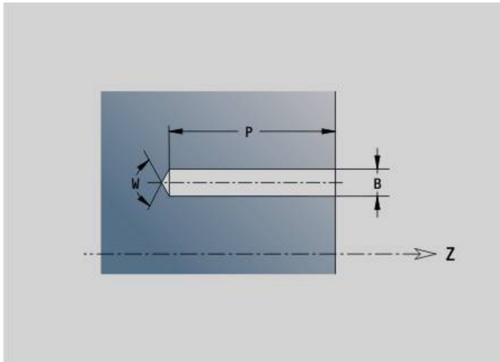
Bore hole on front/rear face G300-Geo

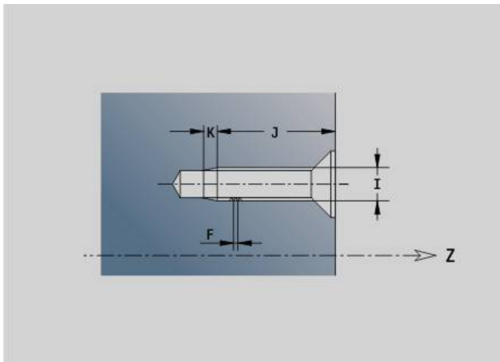
G300 defines a hole with countersinking and thread in a front or rear face contour.

Parameter:

- **XK**: **Center** (in Cartesian coordinates)
- **YK**: **Center** (in Cartesian coordinates)
- **B**: **Diameter**
- **P**: **Depth** excluding point
- **W**: **Point angle** (default: 180°)
- **R**: **Sink diam.**
- **U**: **Sink depth**
- **E**: **Sink angle**
- **I**: **Thread diameter**
- **J**: **Thread depth**
- **K**: **Start of thread** – run-out length
- **F**: **Thread pitch**
- **V**: **Direction of thread:** (default: 0)
  - **0: Right-hand thread**
  - **1: Left-hand thread**
- **A**: **Angle** to Z axis – angle of the hole
  - Front face (range: –90° < **A** < 90°; default: 0°)
  - Rear face (range: 90° < **A** < 270°; default: 180°)
- **O**: **Center. diam.**







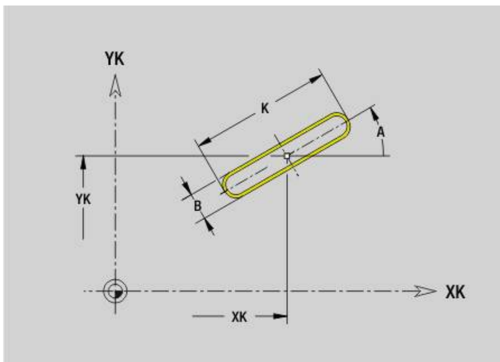
> Machine the **G300** holes with **G71**..**G74**.

Linear slot on front/rear face G301-Geo

G301 defines a linear slot in a contour on the front or rear face.

Parameters:

- **XK**: **Center** (in Cartesian coordinates)
- **YK**: **Center** (in Cartesian coordinates)
- **X**: **Diameter** – **Center** (in polar coordinates)
- **C**: **Angle** – **Center** (in polar coordinates)
- **A**: **Angle** to XK axis (default: 0°)
- **K**: **Length**
- **B**: **Width**
- **P**: **Depth/Height** (default: **P** from **G308**)
  - **P** < 0: Pocket
  - **P** > 0: Island



Circular slot on front/rear face G302-/G303-Geo

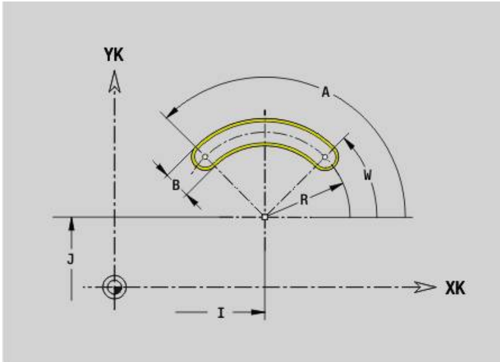
G302 and **G303** define a circular slot in a contour on the front face/ rear face.

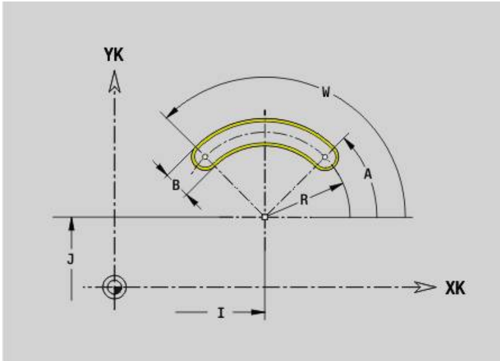
Direction of rotation:

- **G302**: Circular slot clockwise
- **G303**: Circular slot counterclockwise

Parameters:

- **I**: **Center** (in Cartesian coordinates)
- **J**: **Center** (in Cartesian coordinates)
- **X**: **Diameter** – **Center** (in polar coordinates)
- **C**: **Angle** – **Center** (in polar coordinates)
- **R**: **Radius** – Curvature radius (reference: center point path of the slot)
- **A**: **Start angle** to XK axis (default: 0°)
- **W**: **Final angle** to XK axis (default: 0)
- **B**: **Width**
- **P**: **Depth/Height** (default: **P** from **G308**)
  - **P** < 0: Pocket
  - **P** > 0: Island



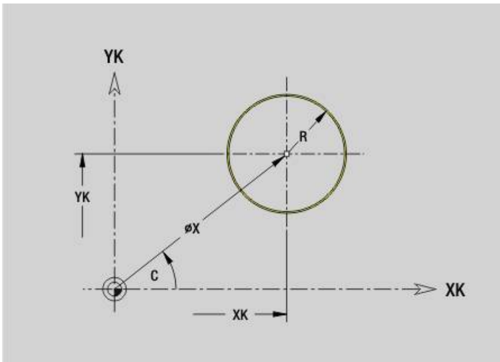


Full circle on front/rear face G304-Geo

G304 defines a **Full circle** in a contour on the front face/rear face.

Parameters:

- **XK**: **Center** (in Cartesian coordinates)
- **YK**: **Center** (in Cartesian coordinates)
- **X**: **Diameter** – **Center** (in polar coordinates)
- **C**: **Angle** – **Center** (in polar coordinates)
- **R**: **Radius**
- **P**: **Depth/Height** (default: **P** from **G308**)
  - **P** < 0: Pocket
  - **P** > 0: Island

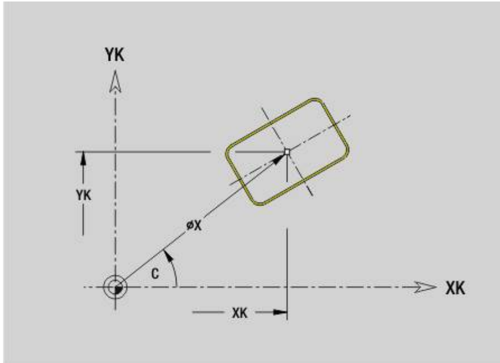


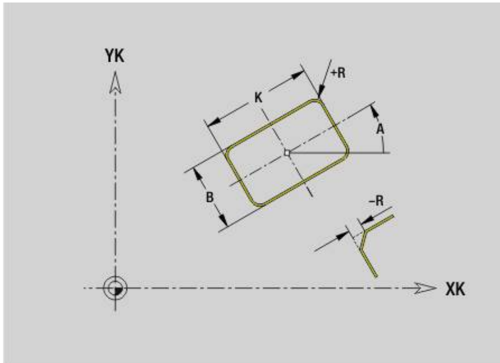
Rectangle on front/rear face G305-Geo

G305 defines a rectangle in a contour on the front face/rear face.

Parameters:

- **XK**: **Center** (in Cartesian coordinates)
- **YK**: **Center** (in Cartesian coordinates)
- **X**: **Diameter** – **Center** (in polar coordinates)
- **C**: **Angle** – **Center** (in polar coordinates)
- **A**: **Angle** to XK axis (default: 0°)
- **K**: **Length** of rectangle
- **B**: **Height** of rectangle
- **R**: **Chamf./round.** (default: 0)
  - **R** > 0: Radius of rounding arc
  - **R** < 0: Chamfer width
- **P**: **Depth/Height** (default: **P** from **G308**)
  - **P** < 0: Pocket
  - **P** > 0: Island



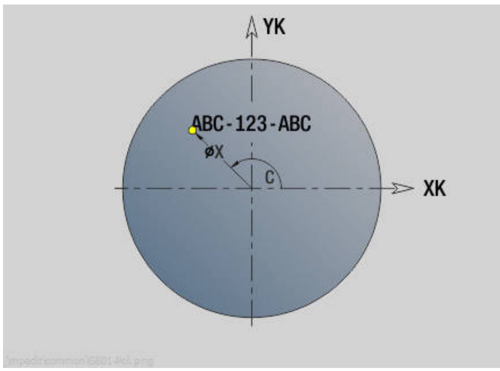


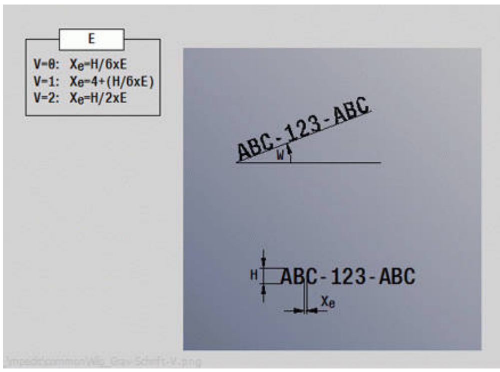
Text for front face CG306-Geo

G306 defines a text on the front face.

Parameters:

- **X**: **Start point** X
- **C**: **Start. angle**
- **XK**: **Start point** (in Cartesian coordinates)
- **YK**: **Start point** (in Cartesian coordinates)
- **ID**: **Text** to be engraved
- **NF**: **Char. no.** – ASCII code of the character to be engraved
- **P**: **Depth**
- **W**: **Inclinat. ang.** of the character string
- **H**: **Font height**
- **E**: **Distance factor**
  The spacing between the characters is calculated according to the following formula: **H** / 6 * **E**
- **V**: **Execution (linear/polar)**
  - **0: Linear**
  - **1: Arched above**
  - **2: Arched below**
- **D**: **Reference diameter**
- **F**: **Plunging feed rate factor** (plunging feed rate = current feed rate * **F**)
- **O**: **Mirror writing**
  - **0** (**No**): Engraving is not mirrored
  - **1** (**Yes**): Engraving is mirrored (mirror writing)



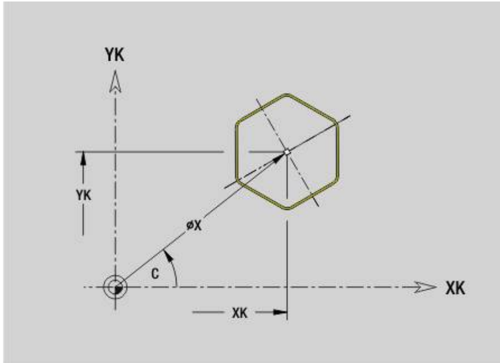


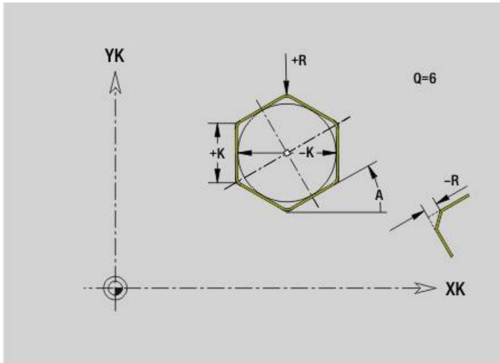
Polygon on front/rear face G307-Geo

G307 defines a polygon in a contour on the front face/rear face.

Parameters:

- **XK**: **Center** (in Cartesian coordinates)
- **YK**: **Center** (in Cartesian coordinates)
- **X**: **Diameter** – **Center** (in polar coordinates)
- **C**: **Angle** – **Center** (in polar coordinates)
- **A**: **Angle** to XK axis (default: 0°)
- **Q**: **Number edges**
- **K**: **+edge lngth/-width a. flats**
  - **K** > 0: Edge length
  - **K** < 0: Width across (Inside diameter)
- **R**: **Chamf./round.** (default: 0)
  - **R** > 0: Radius of rounding arc
  - **R** < 0: Chamfer width
- **P**: **Depth/Height** (default: **P** from **G308**)
  - **P** < 0: Pocket
  - **P** > 0: Island



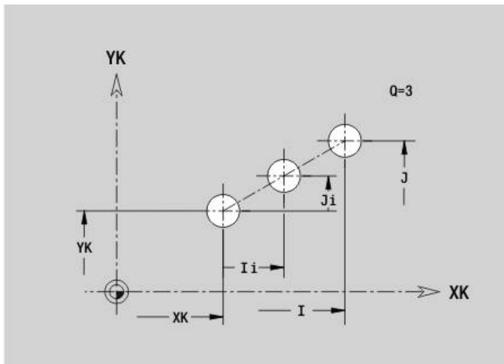


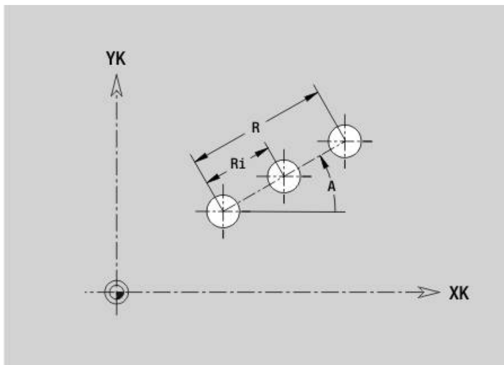
Linear pattern on front/rear face G401-Geo

G401 defines a linear hole pattern or figure pattern on the front or rear face. **G401** is effective for the hole/figure defined in the following block (**G300**..**G305**, **G307**).

Parameters:

- **Q**: **Number** of figures
- **XK**: **Start point** (in Cartesian coordinates)
- **YK**: **Start point** (in Cartesian coordinates)
- **I**: **Final point** (in Cartesian coordinates)
- **Ii**: **Final point** – distance between two figures (in X)
- **J**: **Final point** (in Cartesian coordinates)
- **Ji**: **Final point** – distance between two figures (in Y)
- **A**: **Angle** to XK axis (default: 0°)
- **R**: **Length** – total length of pattern
- **Ri**: **Length** – **Incremental distance**





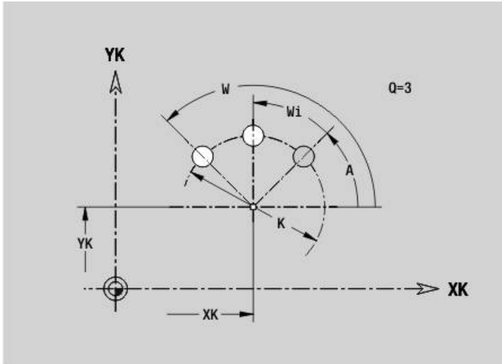
> Programming notes:
> - Program the hole or figure in the following block without a center
> - In the **MACHINING** section, the drilling or milling cycle calls the hole or figure in the following block—not the pattern definition

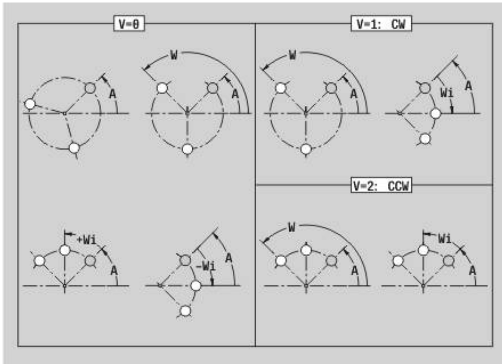
Circular pattern on front/rear face G402-Geo

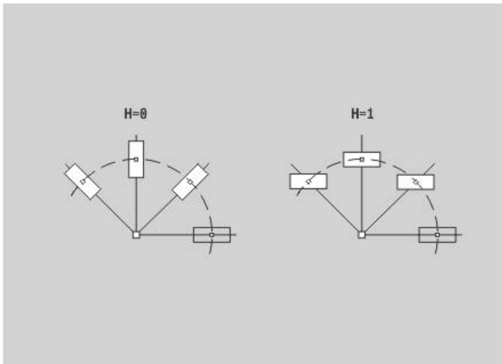
G402 defines a circular hole pattern or figure pattern on the front or rear face. **G402** is effective for the hole/figure defined in the following block (**G300**..**G305**, **G307**).

Parameters:

- **Q**: **Number** of figures
- **K**: **Pattern diameter**
- **A**: **Start angle** – position of the first figure (reference: positive XK axis; (default: 0°)
- **W**: **Final angle** – position of the last figure (reference: positive XK axis; (default: 360°)
- **Wi**: **Final angle** – **Angle** between two figures
- **V**: **Direction** – orientation (default: 0)
  - **V** = 0, without **W**: Figures are arranged on a full circle
  - **V** = 0, with **W**: Figures are arranged on the longer circular arc
  - **V** = 0, with **W**: The algebraic sign of **Wi** defines the direction (**W** < 0: clockwise)
  - **V** = 1, with **W**: Clockwise
  - **V** = 1, with **W**: Clockwise (algebraic sign of **W** has no effect)
  - **V** = 2, with **W**: Counterclockwise
  - **V** = 2, with **W**: Counterclockwise (algebraic sign of **W** has no effect)
- **XK**: **Center** (in Cartesian coordinates)
- **YK**: **Center** (in Cartesian coordinates)
- **H**: **0=Normal position** – position of the figures (default: 0)
  - **0**: Normal position – 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)







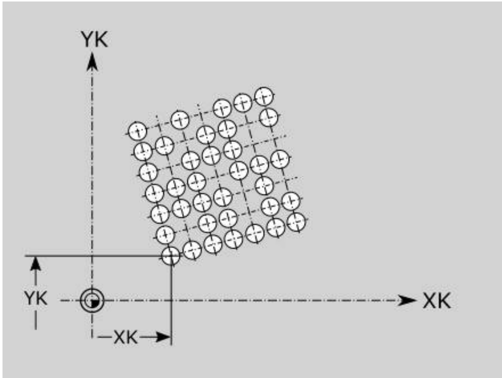
> Programming notes:
>
> - Program the hole or figure in the following block without a center. Exception: circular slot
>   **Further information:** "Circular pattern with circular slots", Page 312
> - The drilling or milling cycle (**MACHINING** section) calls the hole or figure in the following block—not the pattern definition

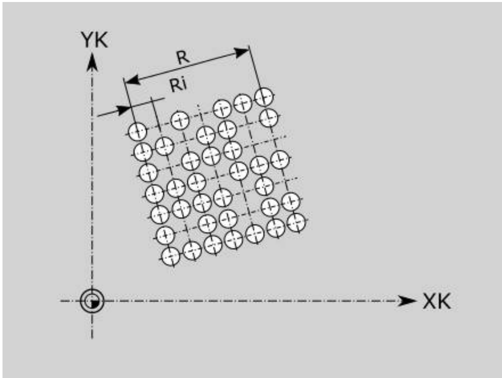
DataMatrix face C G405-Geo

G405 defines a pattern in DataMatrix code on the front face. **G405** is effective for the hole or figure defined in the following block (**G300**, **G304**, **G305**, or **G307**).

Parameters:

- **ID**: **Text** to be converted into DataMatrix code
- **XK**: **Start point** (in Cartesian coordinates)
- **YK**: **Start point** (in Cartesian coordinates)
- **A**: **Angle** to XK axis (default: 0°)
- **R**: **Length** – total length of pattern
- **Ri**: **Length** – distance to next hole or figure



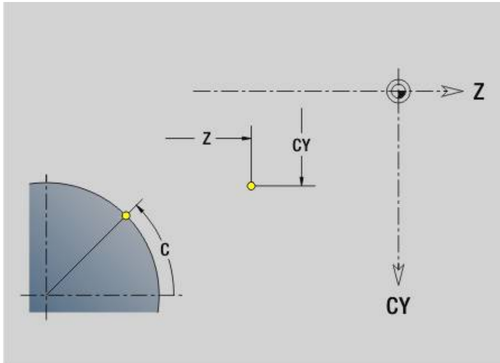


Programming notes

- If you do not enter the length, the control calculates the pattern so that the holes or figures touch each other.
- Program the hole or figure in the following block without a center
- In the **MACHINING** section, the drilling or milling cycle calls the hole or figure in the following block—not the pattern definition
- A maximum of 80 ASCII characters are allowed per DataMatrix code
- The G codes for rectangle and polygon are restricted to a square shape

6.8 Lateral surface contours

Starting point of lateral surface contour G110-Geo



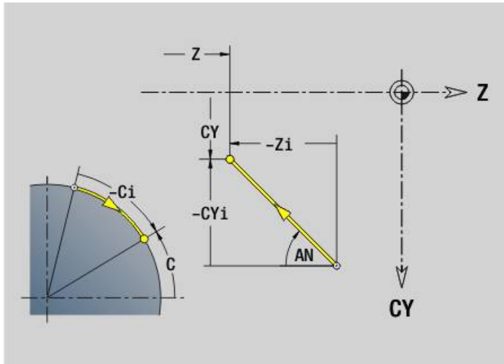
G110 defines the **Start point** of a lateral surface contour.

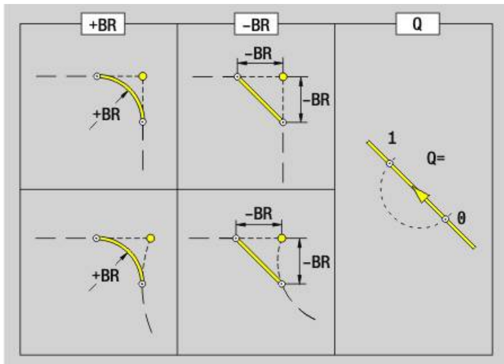
Parameters:

- **Z**: **Start point**
- **C**: **Start angle** (angle, in polar coordinates)
- **CY**: **Start point** as a linear dimension (reference: unrolled lateral surface, using the **Reference diameter**)
- **PZ**: **Start point** (polar radius)

> Program either **Z**, **C** or **Z**, **CY**.

Line segment in a lateral surface contour G111-Geo





G111 defines a line segment in a lateral-surface contour.

Parameters:

- **Z**: **Final point**
- **C**: **End angle**
- **CY**: **Final point** as a linear dimension (reference: unrolled lateral surface, using the **Reference diameter**)
- **AN**: **Angle** to positive Z axis
- **Q**: **Intersect. pt.** or **Final point** if the line segment intersects a circular arc (default: 0)
  - 0: Near point of intersection
  - 1: Far point of intersection
- **BR**: **Chamf./round.** – defines the transition to the next contour element
  When entering a **Chamf./round.**, program the theoretical end point.
  - No entry: Tangential transition
  - **BR** = 0: No tangential transition
  - **BR** > 0: Rounding radius
  - **BR** < 0: Width of chamfer
- **PZ**: **Final point** (polar radius; reference: workpiece datum)
- **AR**: **Incr.angle to foregoer ARi** (**AR** corresponds to **AN**)
- **R**: **Line length**

> Programming:
> - **Z**, **CY**: Absolute, incremental, modal or **?**
> - **C**: Absolute, incremental or modal
> - **ARi**: Angle to the previous element
> - **ANi**: Angle to the subsequent element

Circular arc in lateral surface contour G112-/G113-Geo

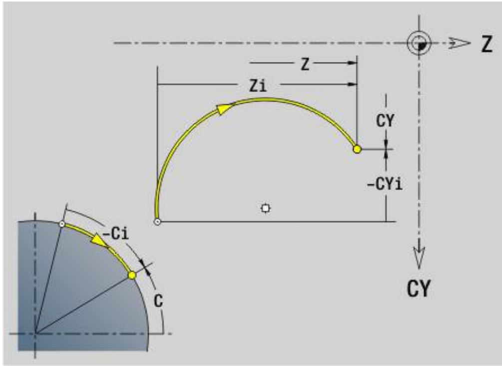
G112 and **G113** define a circular arc in a lateral-surface contour.

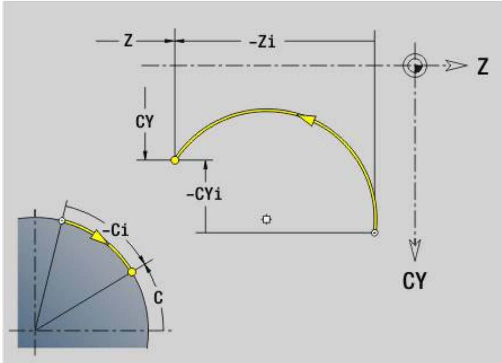
Direction of rotation:

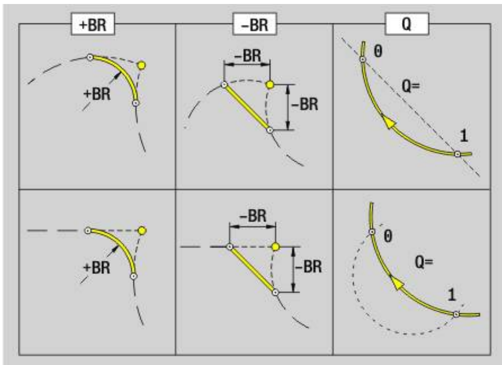
- **G112**: In clockwise direction
- **G113**: In counterclockwise direction

Parameters:

- **Z**: **Final point**
- **C**: **End angle** (in polar coordinates)
- **CY**: **Final point** as a linear dimension (reference: unrolled lateral surface, using the **Reference diameter**)
- **R**: **Radius**
- **K**: **Center** (in Z)
- **J**: **Center** – angle defined on the basis of the linear distance derived from the developed lateral surface and the reference diameter
- **Q**: **Intersect. pt.** or **Final point** if the circular arc intersects a line segment or another circular arc (default: 0)
  - 0: Near point of intersection
  - 1: Far point of intersection
- **BR**: **Chamf./round.** – defines the transition to the next contour element
  When entering a **Chamf./round.**, program the theoretical end point.
  - No entry: Tangential transition
  - **BR** = 0: No tangential transition
  - **BR** > 0: Rounding radius
  - **BR** < 0: Width of chamfer
- **PZ**: **Final point** (polar radius; reference: workpiece datum)
- **W**: **Center** (polar angle; reference: workpiece datum)
- **PM**: **Center** (polar radius; reference: workpiece datum)
- **AR**: **Start angle** – tangential angle to rotary axis
- **AN**: **Final angle** – tangential angle to rotary axis







Programming:

- **Z**, **CY**: Absolute, incremental, modal or **?**
- **C**: Absolute, incremental or modal
- **K**, **J**: Absolute or incremental
- **PZ**, **W**, **PM**: Absolute or incremental
- **ARi**: Angle to the previous element
- **ANi**: Angle to the subsequent element

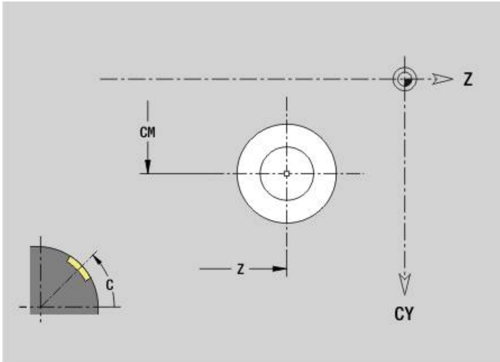
Hole on lateral surface G310-Geo

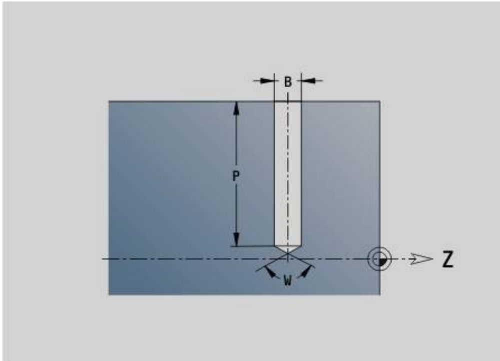
G310 defines a hole with countersink and thread in a lateral surface contour.

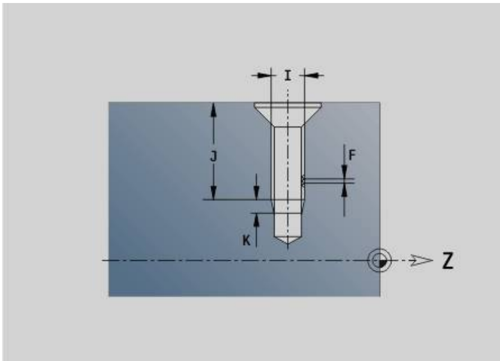
Parameters:

- **Z**: **Center** hole
- **CY**: **Center** as a linear dimension (reference: unrolled lateral surface, using the **Reference diameter**)
- **C**: **Center** (angle)
- **B**: **Diameter**
- **P**: **Depth** excluding point
- **W**: **Point angle** (default: 180°)
- **R**: **Sink diam.**
- **U**: **Sink depth**
- **E**: **Sink angle**
- **I**: **Thread diameter**
- **J**: **Thread depth**
- **K**: **Start of thread** – run-out length
- **F**: **Thread pitch**
- **V**: **Direction of thread:** (default: 0)
  - **0: Right-hand thread**
  - **1: Left-hand thread**
- **A**: **Angle** to Z axis (range: 0° < **A** < 180°; default: 90° = vertical hole)
- **O**: **Center. diam.**

> Machine the **G310** holes with **G71**..**G74**.





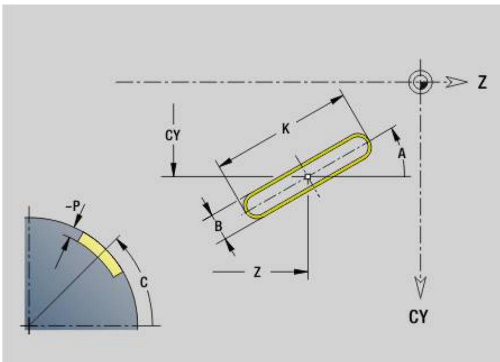


Linear slot on lateral surface G311-Geo

G311 defines a linear slot in a lateral-surface contour.

Parameters:

- **Z**: **Center** of slot
- **CY**: **Center** as a linear dimension (reference: unrolled lateral surface, using the **Reference diameter**)
- **C**: **Center** (angle)
- **A**: **Angle to Z axis** (default: 0°)
- **K**: **Length**
- **B**: **Width**
- **P**: **Depth** (default: **P** from **G308**)



Circular slot on lateral surface G312-/G313-Geo

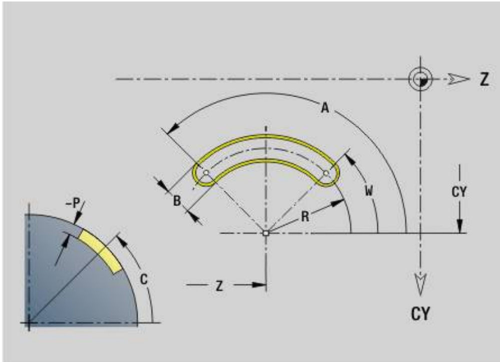
G312 and **G313** define a circular slot in a lateral-surface contour.

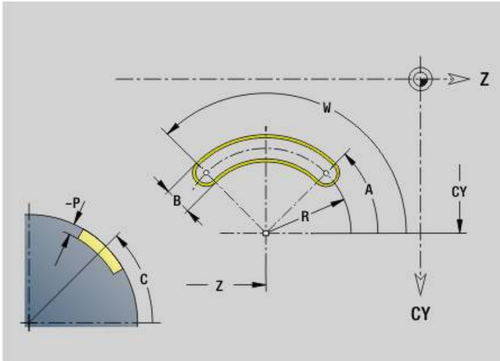
Direction of rotation:

- **G312**: Circular slot clockwise
- **G313**: Circular slot counterclockwise

Parameters:

- **Z**: **Center** of slot
- **CY**: **Center** as a linear dimension (reference: unrolled lateral surface, using the **Reference diameter**)
- **C**: **Center** (angle)
- **R**: **Radius** – Curvature radius (reference: center point path of the slot)
- **A**: **Start angle** to Z axis (default: 0°)
- **W**: **Final angle** to Z axis (default: 0°)
- **B**: **Width**
- **P**: **Depth** (default: **P** from **G308**)



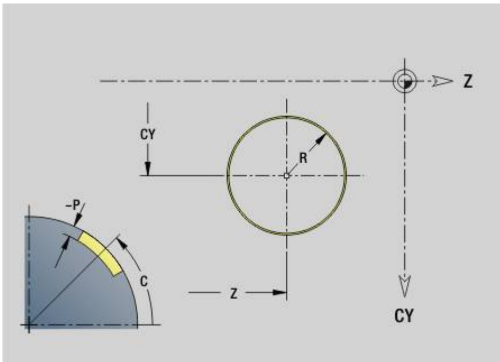


Full circle on lateral surface G314-Geo

G314 defines a full circle in a lateral-surface contour.

Parameters:

- **Z**: **Center**
- **CY**: **Center** as a linear dimension (reference: unrolled lateral surface, using the **Reference diameter**)
- **C**: **Center** (angle)
- **R**: **Radius**
- **P**: **Depth** (default: **P** from **G308**)

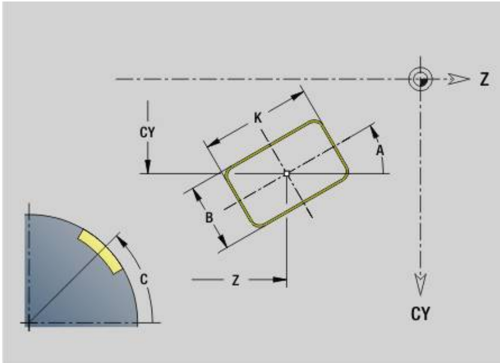


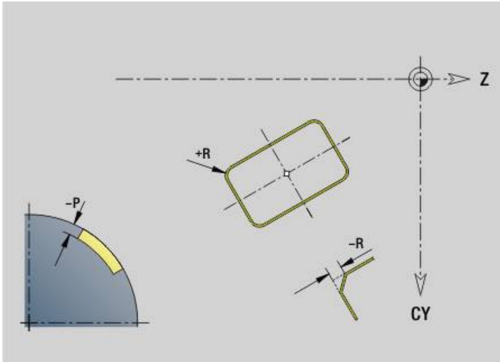
Rectangle, surface G315-Geo

G315 defines a rectangle in a lateral-surface contour.

Parameters:

- **Z**: **Center**
- **CY**: **Center** as a linear dimension (reference: unrolled lateral surface, using the **Reference diameter**)
- **C**: **Center** (angle)
- **A**: **Angle to Z axis** (default: 0°)
- **K**: **Length** of rectangle
- **B**: **Width** of rectangle
- **R**: **Chamf./round.** (default: 0)
  - **R** > 0: Radius of rounding arc
  - **R** < 0: Chamfer width
- **P**: **Depth** (default: **P** from **G308**)



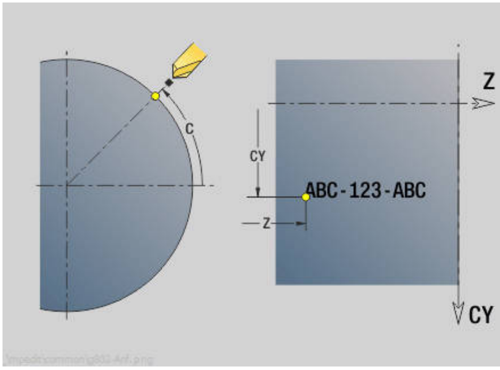


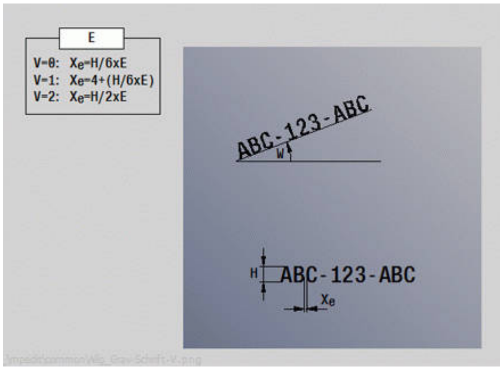
Text for lateral face C G316-Geo

G316 defines a text on the lateral surface.

Parameters:

- **Z**: **Start point**
- **C**: **Start. angle**
- **CY**: **Start point** of first character
- **ID**: **Text** to be engraved
- **NF**: **Char. no.** – ASCII code of the character to be engraved
- **P**: **Depth**
- **W**: **Inclinat. ang.** of the character string
- **H**: **Font height**
- **E**: **Distance factor**
  The spacing between the characters is calculated according to the following formula: **H** / 6 * **E**
- **F**: **Plunging feed rate factor** (plunging feed rate = current feed rate * **F**)
- **O**: **Mirror writing**
  - **0** (**No**): Engraving is not mirrored
  - **1** (**Yes**): Engraving is mirrored (mirror writing)



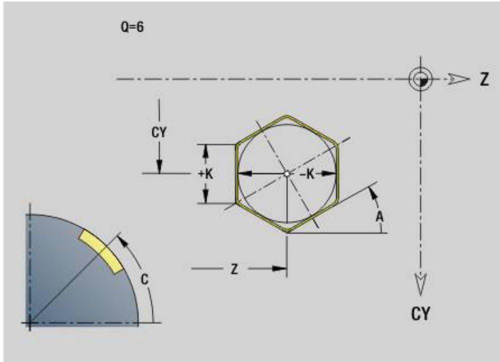


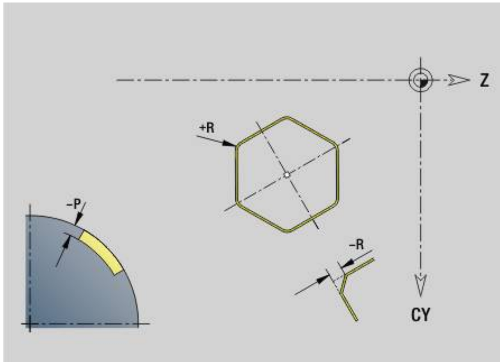
Polygon on lateral surface G317-Geo

G317 defines a polygon in a lateral-surface contour.

Parameters:

- **Z**: **Center**
- **CY**: **Center** as a linear dimension (reference: unrolled lateral surface, using the **Reference diameter**)
- **C**: **Center** (angle)
- **Q**: **Number edges**
- **A**: **Angle to Z axis** (default: 0°)
- **K**: **+edge lngth/-width a. flats**
  - **K** > 0: Edge length
  - **K** < 0: Width across (Inside diameter)
- **R**: **Chamf./round.** (default: 0)
  - **R** > 0: Radius of rounding arc
  - **R** < 0: Chamfer width
- **P**: **Depth** (default: **P** from **G308**)



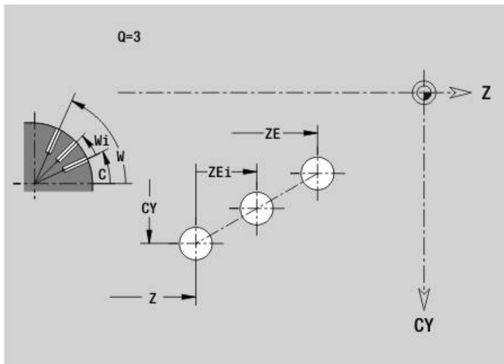


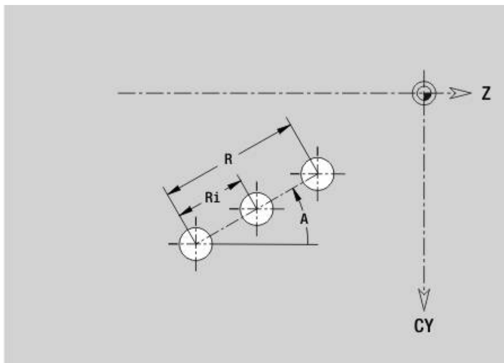
Linear pattern on lateral surface G411-Geo

G411 defines a linear hole or figure pattern on the lateral surface. **G411** is effective for the hole/figure defined in the following block (**G310**..**G315**, **G317**).

Parameters:

- **Q**: **Number** of figures
- **Z**: **Start point**
- **C**: **Start. angle**
- **CY**: **Start point** as a linear dimension (reference: unrolled lateral surface, using the **Reference diameter**)
- **ZE**: **Final point**
- **ZEi**: **Final point** – distance between two figures
- **W**: **Final angle**
- **Wi**: **Final angle** – **Angle** between two figures
- **A**: **Angle to Z axis** (default: 0°)
- **R**: **Length** – total length of pattern
- **Ri**: **Length** – **Incremental distance**





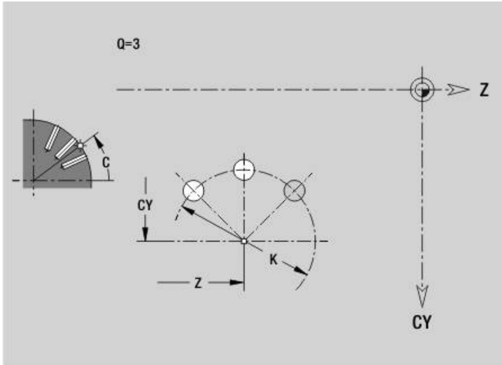
> - If you program **Q**, **Z** and **C** the holes/figures will be ordered in a regular manner along the circumference
> - Program the hole or figure in the following block without a center
> - The milling cycle calls the hole/figure in the following block—not the pattern definition

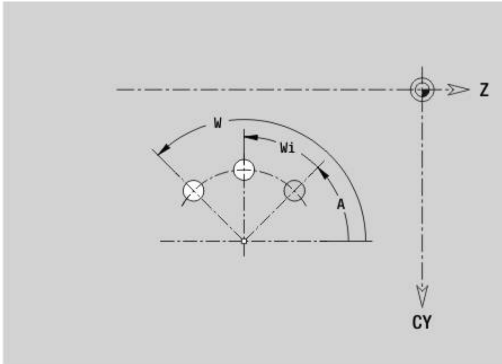
Circular pattern on lateral surface G412-Geo

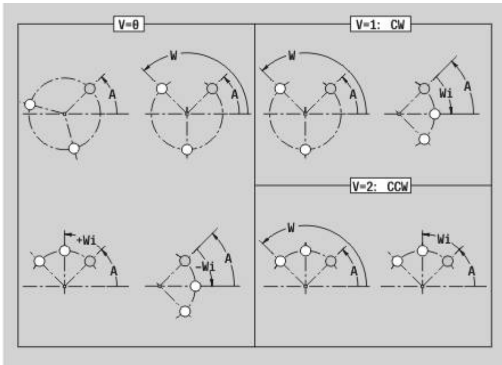
G412 defines a circular hole or figure pattern on the lateral surface. **G412** is effective for the hole/figure defined in the following block (**G310**..**G315**, **G317**).

Parameters:

- **Q**: **Number** of figures
- **K**: **Pattern diameter**
- **A**: **Start angle** – position of the first figure (reference: positive Z axis; (default: 0°)
- **W**: **Final angle** – position of the last figure (reference: positive Z axis; (default: 360°)
- **Wi**: **Final angle** – **Angle** between two figures
- **V**: **Direction** – orientation (default: 0)
  - **V** = 0, without **W**: Figures are arranged on a full circle
  - **V** = 0, with **W**: Figures are arranged on the longer circular arc
  - **V** = 0, with **W**: The algebraic sign of **Wi** defines the direction (**W** < 0: clockwise)
  - **V** = 1, with **W**: Clockwise
  - **V** = 1, with **W**: Clockwise (algebraic sign of **W** has no effect)
  - **V** = 2, with **W**: Counterclockwise
  - **V** = 2, with **W**: Counterclockwise (algebraic sign of **W** has no effect)
- **Z**: **Center** of pattern
- **C**: **Center** (angle)
- **H**: **0=Normal position** – position of the figures (default: 0)
  - **0**: Normal position – 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)







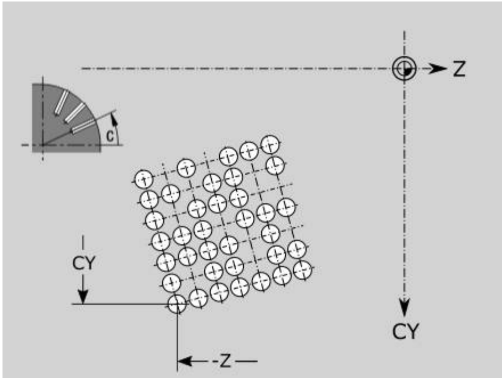
> **i** Programming notes:
> - Program the hole or figure in the following block without a center. Exception: circular slot
>   **Further information:** "Circular pattern with circular slots", Page 312
> - The drilling or milling cycle (**MACHINING** section) calls the hole or figure in the following block—not the pattern definition

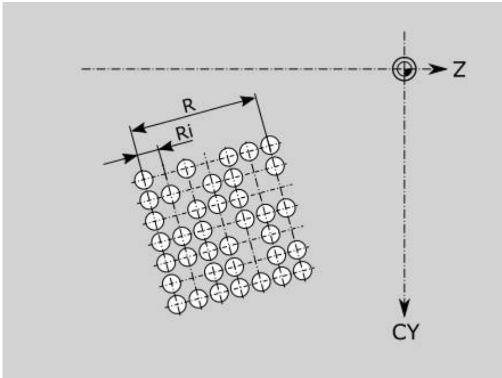
DataMatrix for lateral surface G415-Geo

G415 defines a pattern in DataMatrix code on the lateral surface. **G415** is effective for the hole or figure defined in the following block (**G310**, **G314**, **G315**, or **G317**).

Parameters:

- **ID**: **Text** to be converted into DataMatrix code
- **Z**: **Start point**
- **C**: **Start. angle**
- **CY**: **Start point** as a linear dimension (reference: unrolled lateral surface, using the **Reference diameter**)
- **A**: **Angle to Z axis** (default: 0°)
- **R**: **Length** – total length of pattern
- **Ri**: **Length** – distance to next hole or figure





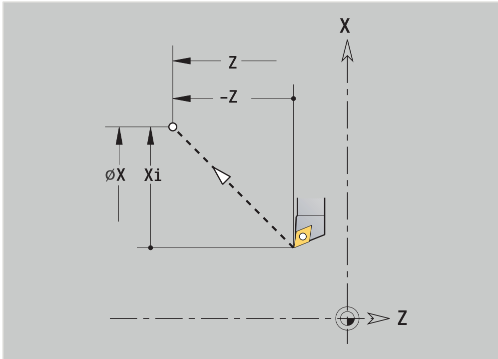
Programming notes

- If you do not enter the length, the control calculates the pattern so that the holes or figures touch each other.
- Program the hole or figure in the following block without a center
- In the **MACHINING** section, the drilling or milling cycle calls the hole or figure in the following block—not the pattern definition
- A maximum of 80 ASCII characters are allowed per DataMatrix code
- The G codes for rectangle and polygon are restricted to a square shape

6.9 Tool positioning

Rapid traverse G0

G0 moves at rapid traverse along the shortest path to the target point.



Parameters:

- **X**: **Diameter**
- **Z**: **Target point**

> Programming:
> - **X** and **Z** absolute, incremental or modal
>
> If more axes are available on your machine, additional input parameters will be displayed, e.g. parameter **B** for the B axis.

Rapid traverse in machine coordinates G701

G701 moves at rapid traverse along the shortest path to the target point.

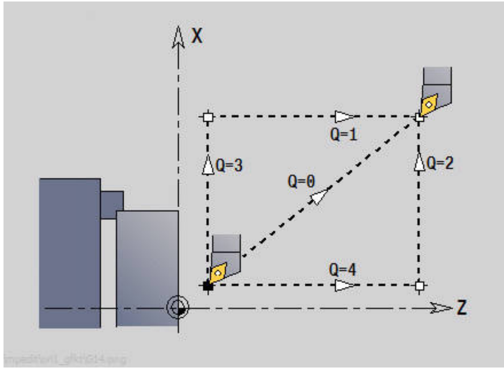
Parameters:

- **X**: **Diameter**
- **Z**: **Target point**

> **X** and **Z** refer to the machine datum and the slide reference point.
>
> If more axes are available on your machine, additional input parameters will be displayed, e.g. parameter **B** for the B axis.

Tool change point G14

G14 moves the tool at rapid traverse to the **Tool change point**. The coordinates for the tool change position can be defined in setup mode.



Parameters:

- **Q**: **Order** (default: 0)
  - **0: Simultaneously**
  - **1: First X, then Z**
  - **2: First Y, then Z, then X**
  - **3: Only X**
  - **4: Only Z**
  - **5: Only Y** (machine-dependent)
  - **6: Simultaneous w/ Y** (machine-dependent)
- **D**: **Number:** of the tool change position to be approached 0-2 (default: 0 = tool change position from parameters)

Example: G14

. . .	
N1 G14 Q0	Move to the tool change position
N2 T3 G95 F0.25 G96 S200 M3	
N3 G0 X0 Z2	
. . .	

Defining the Tool change point G140

G140 defines the position of the **Tool change point** specified in **D**. This position can be approached with **G14**.

Parameters:

- **D**: **Number:** of the tool change point 1-2
- **X**: **Diameter** – position of the tool change point
- **Z**: **Target point** – position of the tool change point

> If **X** or **Z** parameters are missing, the values from the tool change point parameter are entered.

Example: G140

. . .	
N1 G14 Q0	Tool change position from parameter
N2 T3 G95 F0.25 G96 S200 M3	
N3 G0 X40 Z10	
N5 G140 D1 X100 Z100	Set tool change pos.1
N6 G14 Q0 D1	Move to tool change pos. 1
N7 G140 D2 X150	Set tool change pos. 2, use Z from parameters
N8 G14 Q0 D2	Move to tool change pos. 2
. . .	

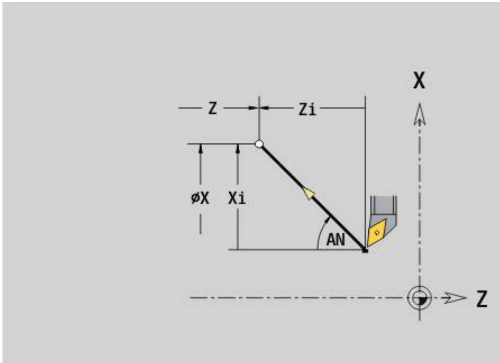
6.10 Linear and circular movements

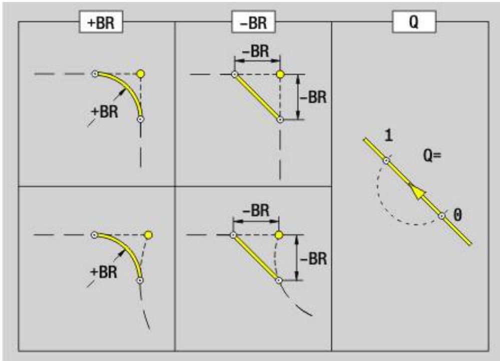
Linear movement G1

G1 moves the tool on a linear path at the feed rate to the end point.

Parameters:

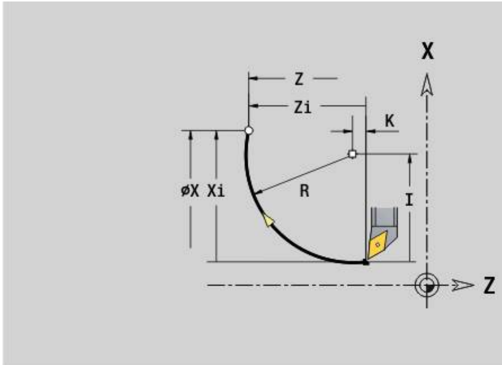
- **X**: **Diameter**
- **Z**: **Target point**
- **AN**: **Angle**
- **Q**: **Intersect. pt.** or **Final point** if the line segment intersects a circular arc (default: 0)
  - 0: Near point of intersection
  - 1: Far point of intersection
- **BR**: **Chamf./round.** – defines the transition to the next contour element
  When entering a **Chamf./round.**, program the theoretical end point.
  - No entry: Tangential transition
  - **BR** = 0: No tangential transition
  - **BR** > 0: Rounding radius
  - **BR** < 0: Width of chamfer
- **BE**: **Special feed rate factor** for **Chamf./round.** (default: 1)
  Special feed rate = active feed rate * **BE** (range: 0 < **BE** <= 1)

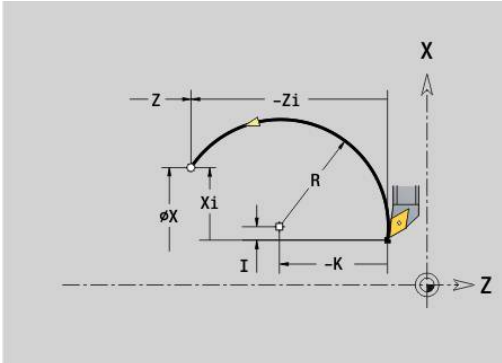




> **i** Programming:
> - **X** and **Z** absolute, incremental or modal
>
> If more axes are available on your machine, additional input parameters will be displayed, e.g. parameter **B** for the B axis.

Circular arc ccw G2/G3





G2 and **G3** move the tool in a circular arc at the feed rate to the end point. The center dimensioning is incremental.

Direction of rotation:

- **G2**: In clockwise direction
- **G3**: In counterclockwise direction

Parameters:

- **X**: **Diameter**
- **Z**: **Target point**
- **R**: **Radius** (0 < **R** <= 200000)
- **I**: **Center, incremental** (radius value)
- **K**: **Center, incremental**
- **Q**: **Intersect. pt.** or **Final point** if the circular arc intersects a line segment or another circular arc (default: 0)
  - 0: Near point of intersection
  - 1: Far point of intersection
- **BR**: **Chamf./round.** – defines the transition to the next contour element
  When entering a **Chamf./round.**, program the theoretical end point.
  - No entry: Tangential transition
  - **BR** = 0: No tangential transition
  - **BR** > 0: Rounding radius
  - **BR** < 0: Width of chamfer
- **BE**: **Special feed rate factor** for **Chamf./round.** (default: 1)
  Special feed rate = active feed rate * **BE** (range: 0 < **BE** <= 1)

> Programming:
> - **X** and **Z** absolute, incremental, modal or **?**

Example: G2, G3

```
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X0 Z2
N3 G42
N4 G1 Z0
N5 G1 X15 B-0.5 E0.05
N6 G1 Z-25 B0
N7 G2 X45 Z-32 R36 B2
N8 G1 A0
N9 G2 X80 Z-80 R20 B5
N10 G1 Z-95 B0
N11 G3 X80 Z-135 R40 B0
N12 G1 Z-140
N13 G1 X82 G40
...
```

Circular arc ccw G12/G13

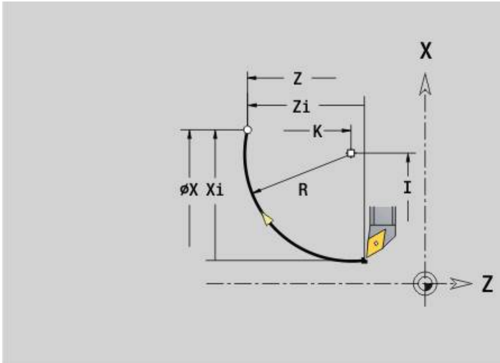
G12 and **G13** move the tool in a circular arc at the feed rate to the end point. The center dimensioning is absolute.

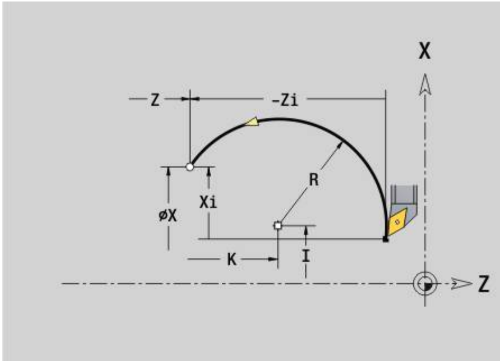
Direction of rotation:

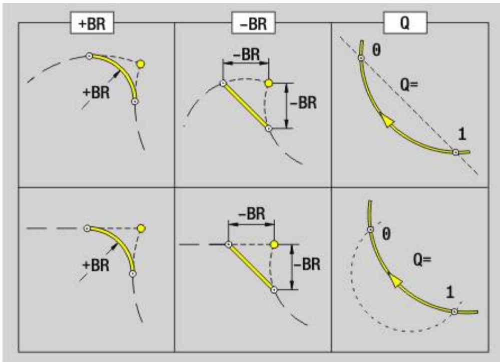
- **G12**: In clockwise direction
- **G13**: In counterclockwise direction

Parameters:

- **X**: **Diameter**
- **Z**: **Target point**
- **R**: **Radius** (0 < **R** <= 200000)
- **I**: **Center** absolute (radius value)
- **K**: **Center** absolute
- **Q**: **Intersect. pt.** or **Final point** if the circular arc intersects a line segment or another circular arc (default: 0)
  - 0: Near point of intersection
  - 1: Far point of intersection
- **BR**: **Chamf./round.** – defines the transition to the next contour element
  When entering a **Chamf./round.**, program the theoretical end point.
  - No entry: Tangential transition
  - **BR** = 0: No tangential transition
  - **BR** > 0: Rounding radius
  - **BR** < 0: Width of chamfer
- **BE**: **Special feed rate factor** for **Chamf./round.** (default: 1)
  Special feed rate = active feed rate * **BE** (range: 0 < **BE** <= 1)







> Programming:
> - **X** and **Z** absolute, incremental, modal or **?**

6.11 Feed rate, shaft speed

Speed limitation G26

The **Speed limitation** remains in effect until the end of the program or until a new value is programmed for **G26** or **Gx26**.

- **G26**: Main spindle
- **Gx26**: Spindle x (x: 1...3)

Parameters:

- **S**: Maximum **Speed**

> If **S** > absolute maximum speed (machine parameter), the parameter value will apply.

Example: G26

. . .	
N1 G14 Q0	
N1 G26 S2000	Maximum speed
N2 T3 G95 F0.25 G96 S200 M3	
N3 G0 X0 Z2	
. . .	

Reduce rapid traverse G48

The reduction of the rapid traverse rate remains in effect until the end of the program or until **G48** is programmed again without input values.

Parameters:

- **F**: **Max. feed** in mm/min for linear axes or °/min for rotary axes
- **D**: **Axis number**
  - **1: X**
  - **2: Y**
  - **3: Z**
  - **4: U**
  - **5: V**
  - **6: W**
  - **7: A**
  - **8: B**
  - **9: C**
- **A**: **Max. acceleration (in %)** (range: 0 % < **A** < 100 %)

  The parameter **A** allows you to reduce the acceleration ramp of the selected axis to the entered percentage value.

Interrupted feed G64

G64 interrupts the programmed feed for a short period of time. **G64** is a modal function.

Parameters:

- **E**: **Interval time** in seconds (range: 0.01 < E < 99.99)
- **F**: **Feed period** in seconds (range: 0.01 < E < 99.99)

Example: G64

. . .	
N1 T3 G95 F0.25 G96 S200 M3	
N2 G64 E0.1 F1	Interrupted feed on
N3 G0 X0 Z2	
N4 G42	
N5 G1 Z0	
N6 G1 X20 B-0.5	
N7 G1 Z-12	
N8 G1 Z-24 A20	
N9 G1 X48 B6	
N10 G1 Z-52 B8	
N11 G1 X80 B4 E0.08	
N12 G1 Z-60	
N13 G1 X82 G40	
N14 G64	Interrupted feed off
. . .	

Feed per tooth Gx93

Gx93 (x: spindle 1...3) defines the drive-dependent feed rate with respect to the number of teeth of the cutter.

Parameters:

- **F**: **Feed per tooth** in mm/tooth or inch/tooth

> The actual value display shows the feed rate in mm/rev.

Example: G193

. . .	
N1 M5	
N2 T1 G197 S1010 G193 F0.08 M104	
N3 M14	
N4 G152 C30	
N5 G110 C0	
N6 G0 X122 Z-50	
N7 G...	
N8 G...	
N9 M15	
. . .	

Constant feed G94 (feed per minute)

G94 defines the feed rate independent of drive.

Parameters:

- **F**: **Feed per min** in mm/min or inch/min

Example: G94

. . .	
N1 G14 Q0	
N2 T3 G94 F2000 G97 S1000 M3	
N3 G0 X100 Z2	
N4 G1 Z-50	
. . .	

Feed per revolution Gx95

Gx95 defines a drive-dependent feed rate.

- **G95**: Main spindle
- **Gx95**: Spindle x (x: 1...3)

Parameters:

- **F**: **Feed per rot.** in mm/rev or inch/rev

Example: G95, Gx95

```
. . .
N1 G14 Q0
N2 T3 G95 F0.25 G96 S200 M3
N3 G0 X0 Z2
N5 G1 Z0
N6 G1 X20 B-0.5
. . .
```

Constant cutting speed Gx96

The spindle speed is dependent on the X position of the tool tip or on the diameter of the drilling or milling tool.

- **G96**: Main spindle
- **Gx96**: Spindle x (x: 1...3)

Parameters:

- **S**: **Cutting speed** in m/min or ft/min

> If you call a drilling tool while a constant cutting speed is active, the control automatically calculates the spindle speed from the programmed cutting speed and activates it with **Gx97**. To prevent inadvertent rotation of the spindle, program the **spindle speed** first and **then T**.

Example: G96, G196

```
. . .
N1 T3 G195 F0.25 G196 S200 M3
N2 G0 X0 Z2
N3 G42
N4 G1 Z0
N5 G1 X20 B-0.5
N6 G1 Z-12
N7 G1 Z-24 A20
N8 G1 X48 B6
N9 G1 Z-52 B8
N10 G1 X80 B4 E0.08
N11 G1 Z-60
N12 G1 X82 G40
. . .
```

Spindle speed Gx97

Constant spindle speed.

- **G97**: Main spindle
- **Gx97**: Spindle x (x: 1...3)

Parameters:

- **S**: **Speed** in revolutions per minute

> **i** **G26**/**Gx26** limits the spindle speed.

Example: G97, G197

```
. . .
N1 G14 Q0
N2 T3 G95 F0.25 G97 S1000 M3
N3 G0 X0 Z2
N5 G1 Z0
N6 G1 X20 B-0.5
. . .
```

6.12 Tool-tip and cutter radius compensation

Fundamentals

Tool-tip radius compensation (TRC)

If **TRC** is not used, the theoretical tool tip is the reference point for the paths of traverse. This might lead to inaccuracies when the tool moves along non-paraxial paths of traverse. The **TRC** function corrects programmed paths of traverse. The **TRC** (**Q**=0) reduces the feed rate for circular arcs if the shifted radius < the original radius. The **TRC** corrects the special feed rate when a rounding arc is machined as transition to the next contour element. Reduced feed rate = feed rate * (shifted radius / original radius)

Milling cutter radius compensation (MCRC)

When the **MCRC** function is not active, the system defines the center of the cutter as the point of reference for the paths of traverse. With the **MCRC** function, the control accounts for the outside diameter of the tool when moving along the programmed paths of traverse. Recessing, area clearance and milling cycles already include **TRC** and **MCRC** calls. The **TRC** and **MCRC** must be switched off when these cycles are called.

Programming notes:

- If the tool radii are > than the contour radii, the **TRC**/**MCRC** might cause endless loops
  Recommendation: Use the finishing cycle **G890** or milling cycle **G840**
- Never program the **MCRC** during a perpendicular approach to the machining plane

Switch off TRC/MCRC G40

G40 is used to deactivate **TRC** and **MCRC**.

Please note:

- The **TRC** and **MCRC** remain in effect until a block with **G40** is reached
- The block containing **G40** or the block after **G40** only permits a linear path of traverse (**G14** is not permissible)

Example: G40

...	
N.. G0 X10 Z10	
N.. G41	Activate TRC to the left of the contour
N.. G0 Z20	Path of traverse: from X10/Z10 to X10+TRC/Z20+TRC
N.. G1 X20	The path of traverse is shifted by the TRC
N.. G40 G0 X30 Z30	Path of traverse from X20+TRC/Z20+TRC to X30/Z30
...	

Switch on TRC/MCRC G41/G42

G41 and **G42** switch on the **TRC** and **MCRC**.

- **G41**: compensation of the tool-tip/cutter radius to the **left** of the contour in traverse direction
- **G42**: compensation of the tool-tip/cutter radius to the **right** of the contour in traverse direction

Parameters:

- **Q**: **Level** (default: 0)
  - 0: TRC on the turning plane (XZ plane)
  - 1: MCRC on the front face (XC plane)
  - 2: MCRC on the lateral surface (ZC plane)
  - 3: MCRC on the front face (XY plane)
  - 4: MCRC on the lateral surface (YZ plane)
- **H**: **Out** (only with TRC – default: 0)
  - 0: Intersecting areas which are programmed in directly successive contour elements are not machined
  - 1: The complete contour is machined—even if certain areas are intersecting
- **O**: **Feed reduc. off** (default: 0)
  - **0: No** (feed rate reduction is active)
  - **1: Yes** (feed rate reduction is not active)

Please note:

- Program **G41**/**G42** in a separate NC block
- Program a straight line segment (**G0**/**G1**) after the block containing **G41**/**G42**
- The **TRC**/**MCRC** is taken into account from the next path of traverse.

Example: G40, G41, G42

. . .	
N1 T3 G95 F0.25 G96 S200 M3	
N2 G0 X0 Z2	
N3 G42	TRC on, to the right of the contour
N4 G1 Z0	
N5 G1 X20 B-0.5	
N6 G1 Z-12	
N7 G1 Z-24 A20	
N8 G1 X48 B6	
N9 G1 Z-52 B8	
N10 G1 X80 B4 E0.08	
N11 G1 Z-60	
N12 G1 X82 G4	TRC off
. . .	

6.13 Datum shifts

You can program several datum shifts in one NC program. The relationships of the coordinates (for blank/finished part, auxiliary contours) are not affected by the datum shifts.

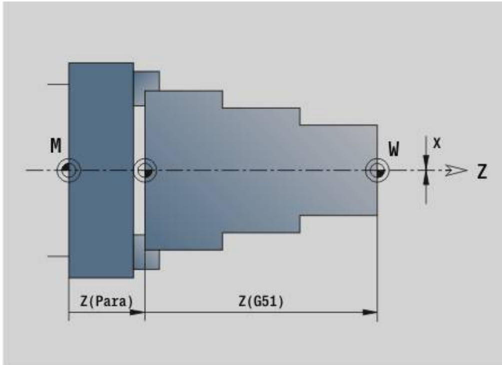
G920 temporarily deactivates datum shifts—**G980** reactivates them.

Overview of datum shifts

G51	■ Relative shift ■ Programmed shift ■ Reference: Previously defined workpiece datum	Page 344
G53/**G54**/**G55**	■ Relative shift ■ Shift defined in setup mode (offset) ■ Reference: Previously defined workpiece datum	Page 345
G56	■ Additive shift ■ Programmed shift ■ Reference: Workpiece datum defined at present	Page 345
G59	■ Absolute shift ■ Programmed shift ■ Reference: Machine datum	Page 346

Datum shift G51

G51 shifts the workpiece datum by the defined value in the selected axis. The **Displacement** is referenced to the workpiece datum defined in setup mode.



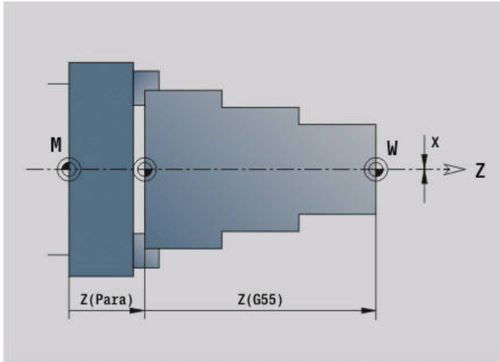
Parameters:

- **X**: **Shift** (radius value)
- **Y**: **Shift** (machine-dependent)
- **Z**: **Shift**
- **U**: **Shift** (machine-dependent)
- **V**: **Shift** (machine-dependent)
- **W**: **Shift** (machine-dependent)

Example: G51

. . .	
N1 T3 G95 F0.25 G96 S200 M3	
N2 G0 X62 Z5	
N3 G810 NS7 NE12 P5 I0.5 K0.2	
N4 G51 Z-28	Datum shift
N5 G0 X62 Z-15	
N6 G810 NS7 NE12 P5 I0.5 K0.2	
N7 G51 Z-56	Datum shift
. . .	

Datum shifts – G53/G54/G55



G53, **G54** and **G55** shift the workpiece datum by the offset values defined in setup mode.

The **Displacement** is referenced to the workpiece datum defined in setup mode, even if the datum is shifted several times with **G53**, **G54**, and **G55**.

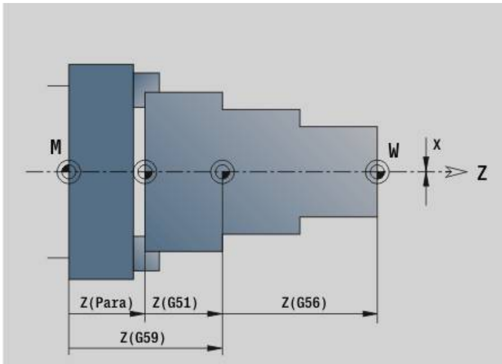
The **Displacement** is valid until you program another datum shift or until the end of the program.

Before using the **G53**, **G54**, or **G55 Displacement** , you need to define the datum shift values in setup mode.

Further information: User's Manual

> A shift in X is entered as a radius.

Additive datum shift G56



G56 shifts the workpiece datum by the defined value in the selected axis. The **Displacement** is referenced to the currently active workpiece datum.

Parameters:

- **X**: **Shift** (radius value)
- **Y**: **Shift** (machine-dependent)
- **Z**: **Shift**
- **U**: **Shift** (machine-dependent)
- **V**: **Shift** (machine-dependent)
- **W**: **Shift** (machine-dependent)

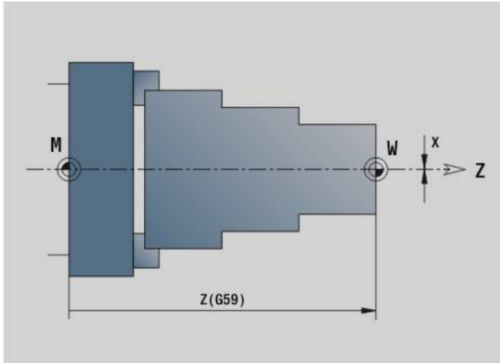
If you shift the workpiece datum more than once with **G56**, the **Displacement** is always added to the currently active workpiece datum.

Example: G56

. . .	
N1 T3 G95 F0.25 G96 S200 M3	
N2 G0 X62 Z5	
N3 G810 NS7 NE12 P5 I0.5 K0.2	
N4 G56 Z-28	Datum shift
N5 G0 X62 Z5	
N6 G810 NS7 NE12 P5 I0.5 K0.2	
N7 G56 Z-28	Datum shift
. . .	

Absolute datum shift G59

G59 sets the workpiece datum to the defined value in the selected axis. The new workpiece datum remains in effect to the end of the program.



Parameters:

- **X**: **Shift** (radius value)
- **Y**: **Shift** (machine-dependent)
- **Z**: **Shift**
- **U**: **Shift** (machine-dependent)
- **V**: **Shift** (machine-dependent)
- **W**: **Shift** (machine-dependent)

> **G59** cancels all previous datum shifts (with **G51**, **G56** or **G59**).

Example: G59

. . .	
N1 G59 Z256	Datum shift
N2 G14 Q0	
N3 T3 G95 F0.25 G96 S200 M3	
N4 G0 X62 Z2	
. . .	

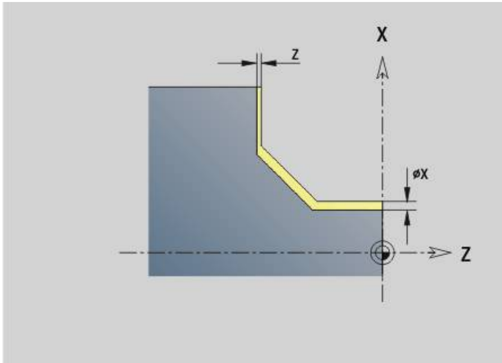
6.14 Oversizes

Switching off oversize G50

G50 switches off **Ov.size** defined with **G52** Geo for the following cycle. Program **G50** before the cycle.

To ensure compatibility, the **G52** code is also supported for switching off the oversizes. HEIDENHAIN recommends using **G50** for new NC programs.

Axis-parallel oversize G57



G57 defines different oversizes for X and Z. Program **G57** before the cycle call.

Parameters:

- **X**: **O-size X** (only positive values; diameter value)
- **Z**: **O-size Z** (only positive values)

G57 is effective differently in the following cycles:

- After cycle run, the oversizes are **deleted** with **G810**, **G820**, **G830**, **G835**, **G860**, **G869**, **G890**, **G891**, **G895**
- After cycle run the oversizes are **not deleted** with **G81**, **G82**, **G83**

> If the oversizes are programmed with **G57** and in the cycle itself, the cycle oversizes apply.

Example: G57

. . .	
N1 T3 G95 F0.25 G96 S200 M3	
N2 G0 X120 Z2	
N3 G57 X0.2 Z0.5	Paraxial oversize
N4 G810 NS7 NE12 P5	
. . .	

Contour-parallel oversize (equidistant) G58

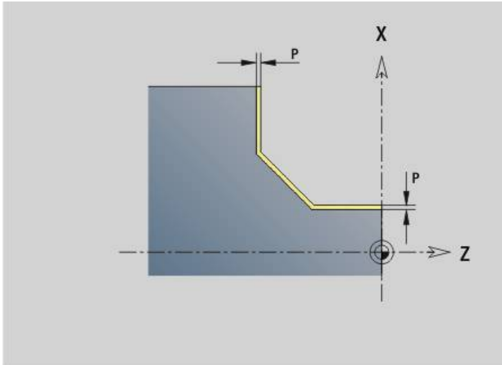
G58 defines a contour-parallel **Ov.size**. Program **G58** before the cycle call. A negative **Ov.size** is permitted during finishing with **G890**.

Parameters:

- **P**: **Allowance**

G58 is effective differently in the following cycles:

- After cycle run, the oversizes are **deleted** with **G810**, **G820**, **G830**, **G835**, **G860**, **G869**, **G890**
- After cycle run the oversizes are **not deleted** with **G83**



> If an oversize is programmed with **G58** and in the cycle, the oversize from the cycle is used.

Example: G58

...	
N1 T3 G95 F0.25 G96 S200 M3	
N2 G0 X120 Z2	
N3 G58 P2	Contour-parallel oversize
N4 G810 NS7 NE12 P5	
...	

6.15 Safety clearance

Safety clearance G47

G47 defines the **Safety clearance** for the following cycles:

- Turning cycles **G810**, **G820**, **G830**, **G835**, **G860**, **G869** and **G890**
- Drilling cycles **G71**, **G72** and **G74**
- Milling cycles **G840** to **G846**

Parameters:

- **P**: **Safety clearance**

G47 without parameters activates the parameter values defined in the **DefGlobG47P** (no. 602012) machine parameter.

> **G47** replaces the safety clearance set in the machining parameters or that set in **G147**.

Safety clearance G147

G147 defines the **Safety clearance** for the following cycles:

- Drilling cycles **G71**, **G72** and **G74**
- Milling cycles **G840** to **G846**

Parameters:

- **I**: **Safety clearance** to the milling plane (only for milling operations)
- **K**: **Safety clearance** in infeed direction (downfeed)

G147 without parameters activates the parameter values defined in the **DefGlobG147SCI** (no. 602014) and **DefGlobG147SCK** (no. 602014) machine parameters.

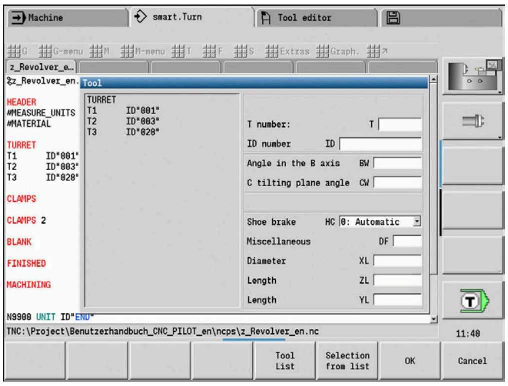
> **G147** replaces the safety clearance set in the machining parameters or that set in **G47**.

6.16 Tools, compensations

Inserting tools – T

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

The control displays the tool assignment defined in the **TURRET** section. You can enter the T number directly or select it from the tool list (switch with the **Tool List** soft key).

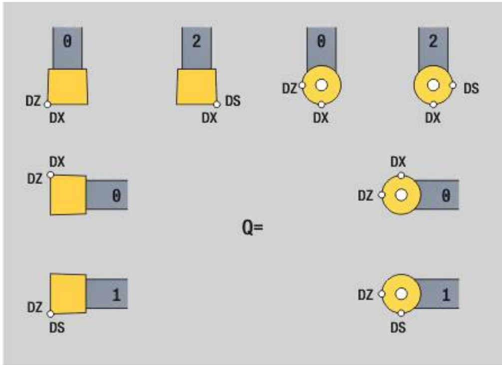


Correction of cut G148 (changing the cutter compensation)

G148 defines the values compensating for wear. **DX**, **DZ** become effective after program start and after a **T** command.

Parameters:

- **O**: **Selection** (default: 0)
  - **O** = 0: **DX**, **DZ** active – **DS** inactive
  - **O** = 1: **DS**, **DZ** active – **DX** inactive
  - **O** = 2: **DX**, **DS** active – **DZ** inactive



> The cycles **G860**, **G869**, **G879**, **G870** and **G890** automatically take the correct wear compensation into account.

Example: G148

. . .	
N1 T3 G95 F0.25 G96 S160 M3	
N2 G0 X62 Z2	
N3 G0 Z-29.8	
N4 G1 X50.4	
N5 G0 X62	
N6 G150	
N7 G1 Z-20.2	
N8 G1 X50.4	
N9 G0 X62	
N10 G151	Recessing finishing
N11 G148 O0	Change compensation
N12 G0 X62 Z-30	
N13 G1 X50	
N14 G0 X62	
N15 G150	
N16 G148 O2	
N17 G1 Z-20	
N18 G1 X50	
N19 G0 X62	
. . .	

Additive correction G149

The control manages 16 tool-independent compensation values. One **G149** followed by a **D** number activates the additive compensation function. **G149 D900** deactivates the additive compensation function. The compensation values are managed in **Program run** submode.

Further information: User's Manual

Parameters:

- **D**: **Corr. additiv** (default: 900)
  - **D** = 900: Deactivates the additive compensation
  - **D** = 901-916: Activates the additive compensation **D**

Programming:

- Program **G149** one block before the block containing the path of traverse to which the compensation is to apply, because the tool must have moved in the compensation direction by the compensation value before compensation becomes effective.
- Additive compensation remains in effect up to:
  - the next **G149 D900**
  - the next tool change
  - end of program

> The additive compensation is added to the tool compensation.

Example: G149

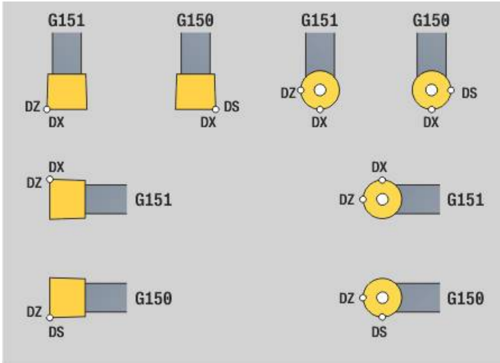
. . .	
N1 T3 G96 S200 G95 F0.4 M4	
N2 G0 X62 Z2	
N3 G89	
N4 G42	
N5 G0 X27 Z0	
N6 G1 X30 Z-1.5	
N7 G1 Z-25	
N8 G149 D901	Activate compensation
N9 G1 X40 BR-1	
N10 G1 Z-50	
N11 G149 D902	
N12 G1 X50 BR-1	
N13 G1 Z-75	
N14 G149 D900	Deactivate compensation
N15 G1 X60 B-1	
N16 G1 Z-80	
N17 G1 X62	
N18 G80	
. . .	

Compensation of tool tip G150/G151

G150/**G151** defines the tool reference point for recessing and button tools.

- **G150**: Reference point is on right tip
- **G151**: Reference point is on left tip

G150 and **G151** is effective from the block in which it is programmed and remains in effect up to the next tool change or program end.



- The displayed actual values always refer to the tool tip defined in the tool data
- If you use TRC, after **G150**/**G151** you must also adjust **G41**/**G42**

Example: G148

. . .	
N1 T3 G95 F0.25 G96 S160 M3	
N2 G0 X62 Z2	
N3 G0 Z-29.8	
N4 G1 X50.4	
N5 G0 X62	
N6 G150	
N7 G1 Z-20.2	
N8 G1 X50.4	
N9 G0 X62	
N10 G151	Recessing, finishing
N11 G148 O0	
N12 G0 X62 Z-30	
N13 G1 X50	
N14 G0 X62	
N15 G150	
N16 G148 O2	
N17 G1 Z-20	
N18 G1 X50	
N19 G0 X62	
. . .	

6.17 Contour-based turning cycles

Working with contour-based cycles

Possibilities of transferring the contour to be machined to the cycle:

- Transferring the contour reference in the **Contour start block no.** and the **Contour end block no.** The contour area is machined in the direction from **NS** to **NE**
- Transferring the contour reference via the name of the **Auxiliary contour** (**ID**). The complete **Auxiliary contour** is machined in the direction of contour definition
- Describing the contour with **G80** in the block directly after the cycle
  Further information: "Cycle end / simple contour G80", Page 390
- Describing the contour with **G0**, **G1**, **G2** and **G3** blocks directly after the cycle. The contour is concluded by **G80** without parameters

Possibilities of defining the workpiece blank for calculating the number of cutting passes:

- Defining a global workpiece blank in the **BLANK** program section. Contour follow-up of the workpiece blank is automatically active. The cycle uses the specified **Workpiece blank**
- If no global **Workpiece blank** has been defined, the cycle calculates an internal Workpiece blank based on the definition of parameter **RH**

Example: Contour-based cycles

. . .	
N1 G810 NS7 NE12 P3	Block reference
N2 ...	
N3 G810 ID"007" P3	Name of auxiliary contour
N4 ...	
N5 G810 ID"007" NS9 NE7 P3	Combination
N6 ...	
N7 G810 P3	Predefined contour description
N8 G80 XS60 ZS-2 XE90 ZE-50 AC10 WC10BS3 BE-2 RC5 EC0	
N9...	
N10 G810 P3	Direct contour description
N11 G0 X50 Z0	
N12 G1 Z-62 BR4	
N13 G1 X85 AN80 BR-2	
N14 G1 Zi-5	
N15 G80	
N16 ...	
. . .	

Finding the block references:

- Place cursor in **NS** or **NE** input field

Contour reference

- Press the **Contour reference** soft key
- Select the contour element:
  - Use the horizontal arrow keys to select the contour element
  - Use the vertical arrow keys to switch between contours (also face contours, etc.).

NS

- Switch between **NS** and **NE**:
  - Press the **NS** soft key
  - Press the **NE** soft key

Take over

- Press the **Take over** soft key to return to the dialog

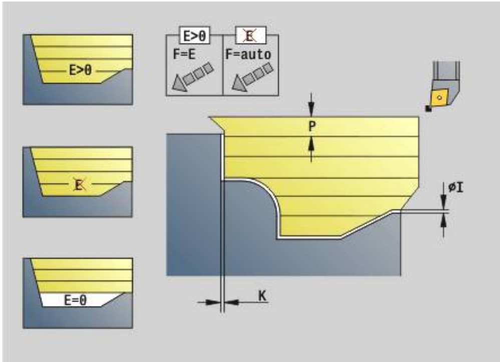
> The contour element can also be selected via touch gestures or the mouse.

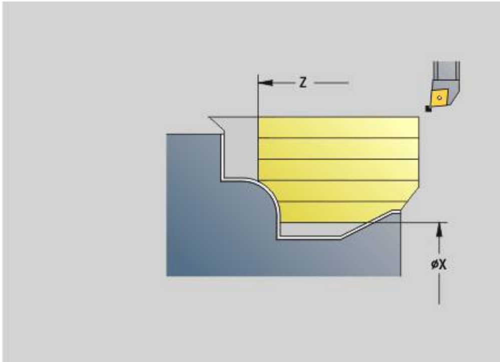
Cutting limits in X, Z

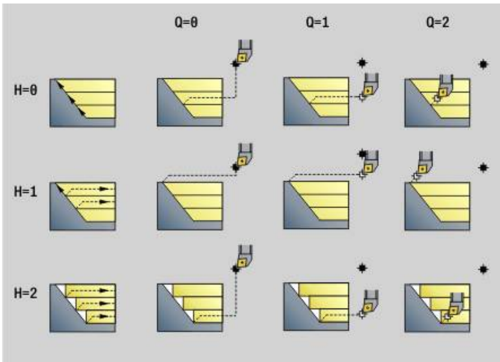
The tool position before the cycle call determines the effect of a cutting limit. The control machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called.

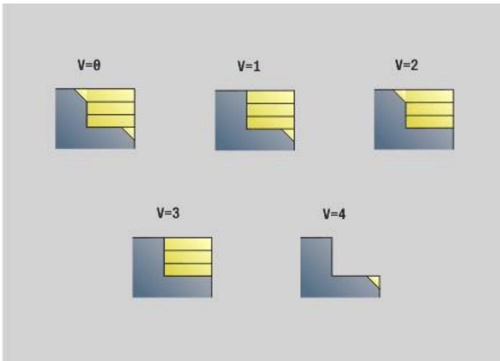
> A cutting limit restricts the contour area that can be machined; it does not apply to the paths for approach and departure.

Longitud. roughing G810









G810 machines the defined contour area. The reference to the contour to be machined can be transferred in the cycle parameters, or the contour can be defined directly after the cycle call.

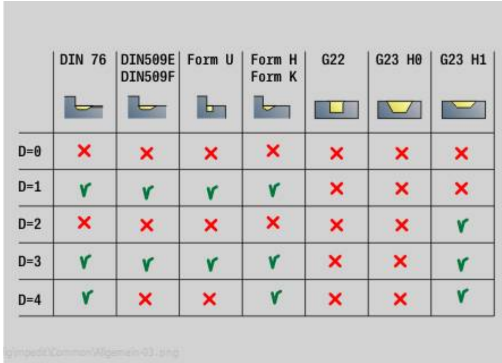
Further information: "Working with contour-based cycles", Page 354

The contour to be machined can contain various valleys. If required, the area to be machined is divided into several sections.

Parameters:

- **ID**: **Auxiliary contour** – ID number of the contour to be machined
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
  - **NE** not programmed: The contour element **NS** is machined in the direction of contour definition
  - **NS** = **NE** programmed: The contour element **NS** is machined opposite to the direction of contour definition.
- **P**: **Maximum infeed**
- **I**: **O-size X**
- **K**: **O-size Z**
- **E**: **Plunging behavior**
  - No input: Automatic feed-rate reduction
  - **E** = 0: No plunging
  - **E** > 0: Plunging feed rate in use
- **X**: **Cutting limit in X** (diameter value; default: no cutting limit)
- **Z**: **Cutting limit in Z** (default: no cutting limit)
- **A**: **Start angle** (reference: Z axis; default: parallel to Z axis)
- **W**: **Depart.angle** (reference: Z axis; default: orthogonal to Z axis)
- **H**: **Contour smoothing**
  - **0: With each cut**
  - **1: With the last cut**
  - **2: No smoothing**
- **Q**: **Kind of liber.** at end of cycle
  - **0: Back to beg., X before Z**
  - **1: Before finished contour**
  - **2: Retract by safety clear.**
- **V**: **Machine form elements** (default: 0)

  A chamfer/rounding arc is machined
  - **0: At beginning and end**
  - **1: At beginning**
  - **2: At end**
  - **3: No machining**
  - **4: Only chamfer/rounding** is machined—not the basic element (requirement: the contour section consists of a single element)
- **D**: **Omit elements** (see figure)
- **U**: **Cut line on horiz. element**
  - **0: No** (uniform cutting segmentation)
  - **1: Yes** (irregular cutting segmentation, if applicable)



	DIN 76	DIN509E DIN509F	Form U	Form H Form K	G22	G23 H0	G23 H1
D=0	✗	✗	✗	✗	✗	✗	✗
D=1	✓	✓	✓	✓	✗	✗	✗
D=2	✗	✗	✗	✗	✗	✗	✓
D=3	✓	✓	✓	✓	✗	✗	✓
D=4	✓	✗	✗	✓	✗	✗	✓

- **O**: **Hide undercutting**
  - **0: No**
  - **1: Yes**
- **B**: **Slide mov.diff.** – slide lead with 4-axis machining
  - **B** = 0: Both slides work on the same diameter—at double feed rate
  - **B** < 0: The slides work on different diameters at the same feed rate. The slide with larger number leads at a defined distance
  - **B** > 0: The slides work on different diameters at the same feed rate. The slide with the smaller number leads at a defined distance
- **RH**: **Wrkpc. blank contour** – evaluation only if no workpiece blank has been defined
  - **0:** (depends on defined parameters)
    - No parameters: workpiece blank from ICP contour and tool position
    - **XA** and **ZA**: workpiece blank from ICP contour and starting point of workpiece blank
    - **J**: workpiece blank from ICP contour and equidistant oversize
  - **1: From tool position** (workpiece blank from ICP contour and tool position)
  - **2: With blank start point** (workpiece blank from ICP contour and starting point of workpiece blank **XA** and **ZA**)
  - **3 Equidistant oversize** (workpiece blank from ICP contour and equidistant oversize **J**)
  - **4: Long.-transv. oversize** (workpiece blank from ICP contour, transverse oversize **XA** and longitudinal oversize **ZA**)
- **J**: **Workpiece blank oversize** (radius value – will only be evaluated if no workpiece blank has been defined)
- **XA**, **ZA**: **Start point blank** (Definition of the corner point of the workpiece blank contour—an evaluation will only take place if no workpiece blank has been defined beforehand)

The control uses the tool definition to distinguish between external and internal machining.

> - The tool radius compensation is active
> - A **G57** oversize enlarges the contour (also inside contours).
> - A **G58** oversize
>   - >0: Enlarges the contour
>   - <0: Is not offset
> - **G57**/**G58** oversizes are deleted after cycle end

Cycle run:

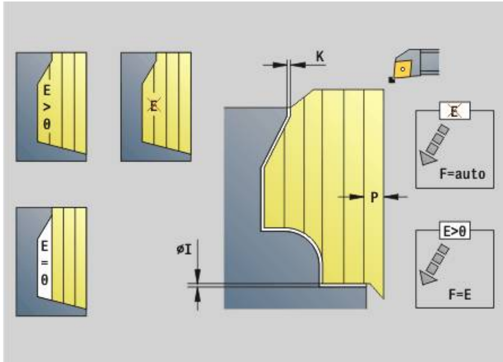
1 Calculates the areas to be machined and the cutting segmentation
2 Approaches workpiece for first pass from starting point, taking the safety clearance into account (first in Z direction, then in X)
3 Moves at feed rate to **Cutting limit in Z**
4 Depending on **H**:
   - **H** = 0: Machines the contour outline
   - **H** = 1 or 2: Retracts at 45°
5 Returns at rapid traverse and approaches for next pass
6 Repeats 3 to 5 until **Cutting limit in X** has been reached
7 If required, repeats 2 to 6 until all areas have been machined
8 If **H** = 1: Smooths the contour
9 Retracts as programmed in **Q**

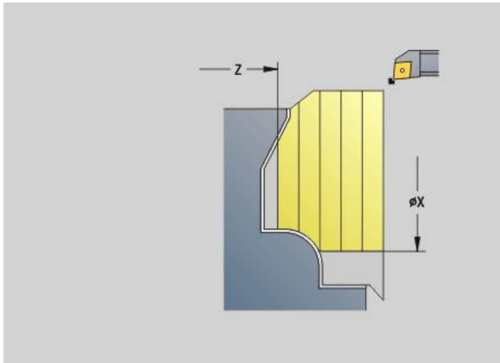
Application as 4-axis cycle

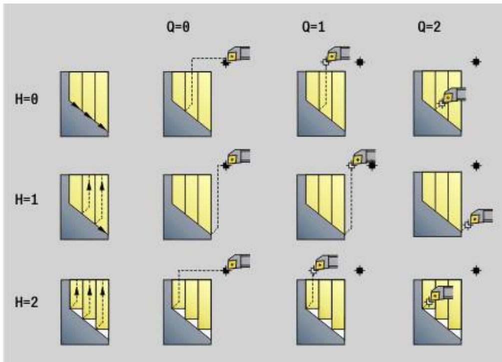
- Same diameter:
  - Both slides start simultaneously
- Differing diameters:
  - When the leading slide has reached **Slide mov.diff. B**, the second slide starts. This synchronization takes place with every cut
  - Each slide advances by the calculated depth of cut
  - If the number of cuts is uneven, the leading slide performs the last cut
  - If you are machining at a constant cutting speed, the cutting speed of the leading slide is used. The leading tool does not retract until the subsequent tool is ready for use

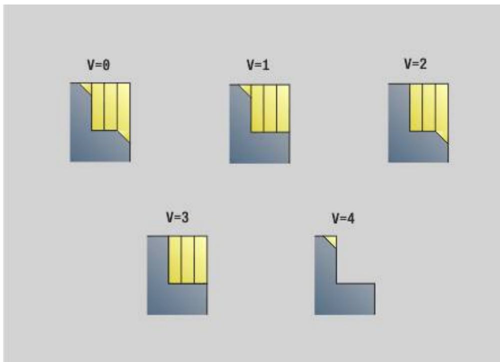
> - If you use 4-axis cycles, ensure that you use identical tools (e.g. tool type, cutting radius)
> - Undercuts are not machined in 4-axis cycles. Parameter **O** is hidden

Face roughing G820









G820 machines the defined contour area. The reference to the contour to be machined can be transferred in the cycle parameters, or the contour can be defined directly after the cycle call.

Further information: "Working with contour-based cycles", Page 354

The contour to be machined can contain various valleys. If required, the area to be machined is divided into several sections.

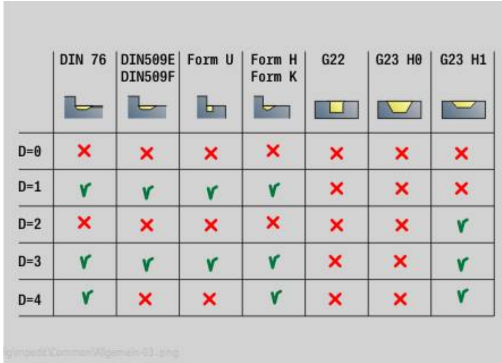
Parameters:

- **ID**: **Auxiliary contour** – ID number of the contour to be machined
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
  - **NE** not programmed: The contour element **NS** is machined in the direction of contour definition
  - **NS** = **NE** programmed: The contour element **NS** is machined opposite to the direction of contour definition.
- **P**: **Maximum infeed**
- **I**: **O-size X**
- **K**: **O-size Z**
- **E**: **Plunging behavior**
  - No input: Automatic feed-rate reduction
  - **E** = 0: No plunging
  - **E** > 0: Plunging feed rate in use
- **X**: **Cutting limit in X** (diameter value; default: no cutting limit)
- **Z**: **Cutting limit in Z** (default: no cutting limit)
- **A**: **Start angle** (reference: Z axis; default: orthogonal to Z axis)
- **W**: **Depart.angle** (reference: Z axis; default: parallel to Z axis)
- **H**: **Contour smoothing**
  - **0: With each cut**
  - **1: With the last cut**
  - **2: No smoothing**
- **Q**: **Kind of liber.** at end of cycle
  - **0: Back to beg., X before Z**
  - **1: Before finished contour**
  - **2: Retract by safety clear.**
- **V**: **Machine form elements** (default: 0)

  A chamfer/rounding arc is machined
  - **0: At beginning and end**
  - **1: At beginning**
  - **2: At end**
  - **3: No machining**
  - **4: Only chamfer/rounding** is machined—not the basic element (requirement: the contour section consists of a single element)
- **D**: **Omit elements** (see figure)
- **U**: **Cut line on horiz. element**
  - **0: No** (uniform cutting segmentation)
  - **1: Yes** (irregular cutting segmentation, if applicable)

- **O**: **Hide undercutting**
  - **0: No**
  - **1: Yes**
- **B**: **Slide mov.diff.** – slide lead with 4-axis machining
  - **B** = 0: Both slides work on the same diameter—at double feed rate
  - **B** < 0: The slides work on different diameters at the same feed rate. The slide with larger number leads at a defined distance
  - **B** > 0: The slides work on different diameters at the same feed rate. The slide with the smaller number leads at a defined distance
- **RH**: **Wrkpc. blank contour** – evaluation only if no workpiece blank has been defined
  - **0:** (depends on defined parameters)
    - No parameters: workpiece blank from ICP contour and tool position
    - **XA** and **ZA**: workpiece blank from ICP contour and starting point of workpiece blank
    - **J**: workpiece blank from ICP contour and equidistant oversize
  - **1: From tool position** (workpiece blank from ICP contour and tool position)
  - **2: With blank start point** (workpiece blank from ICP contour and starting point of workpiece blank **XA** and **ZA**)
  - **3 Equidistant oversize** (workpiece blank from ICP contour and equidistant oversize **J**)
  - **4: Long.-transv. oversize** (workpiece blank from ICP contour, transverse oversize **XA** and longitudinal oversize **ZA**)
- **J**: **Workpiece blank oversize** (radius value – will only be evaluated if no workpiece blank has been defined)
- **XA**, **ZA**: **Start point blank** (Definition of the corner point of the workpiece blank contour—an evaluation will only take place if no workpiece blank has been defined beforehand)

The control uses the tool definition to distinguish between external and internal machining.



	DIN 76	DIN509E DIN509F	Form U	Form H Form K	G22	G23 H0	G23 H1
D=0	✗	✗	✗	✗	✗	✗	✗
D=1	✓	✓	✓	✓	✗	✗	✗
D=2	✗	✗	✗	✗	✗	✗	✓
D=3	✓	✓	✓	✓	✗	✗	✓
D=4	✓	✗	✗	✓	✗	✗	✓

- The tool radius compensation is active
- A **G57** oversize enlarges the contour (also inside contours).
- A **G58** oversize
  - >0: Enlarges the contour
  - <0: Is not offset
- **G57**/**G58** oversizes are deleted after cycle end

Cycle run:

1. Calculates the areas to be machined and the cutting segmentation
2. Approaches workpiece for first pass from starting point, taking the safety clearance into account (first in X direction, then in Z)
3. Moves at feed rate to **Cutting limit in X**
4. Depending on **H**:
   - **H** = 0: Machines the contour outline
   - **H** = 1 or 2: Retracts at 45°
5. Returns at rapid traverse and approaches for next pass
6. Repeats 3 to 5 until **Cutting limit in Z** has been reached
7. If required, repeats 2 to 6 until all areas have been machined
8. If **H** = 1: Smooths the contour
9. Retracts as programmed in **Q**

Application as 4-axis cycle

- Same diameter:
  - Both slides start simultaneously
- Differing diameters:
  - When the leading slide has reached **Slide mov.diff. B**, the second slide starts. This synchronization takes place with every cut
  - Each slide advances by the calculated depth of cut
  - If the number of cuts is uneven, the leading slide performs the last cut
  - If you are machining at a constant cutting speed, the cutting speed of the leading slide is used. The leading tool does not retract until the subsequent tool is ready for use

> - If you use 4-axis cycles, ensure that you use identical tools (e.g. tool type, cutting radius)
> - Undercuts are not machined in 4-axis cycles. Parameter **O** is hidden

Contour-parallel roughing G830

G830 machines the contour area defined in **ID**, or by **NS**, **NE**, parallel to the contour.

Further information: "Working with contour-based cycles", Page 354

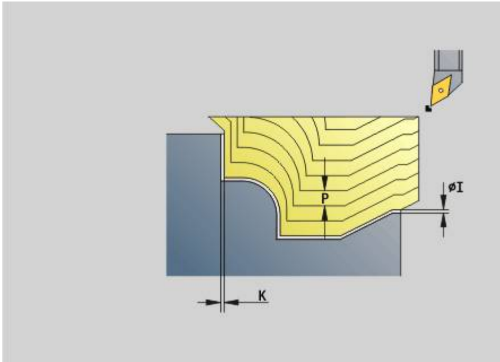
The contour to be machined can contain various valleys. If required, the area to be machined is divided into several sections.

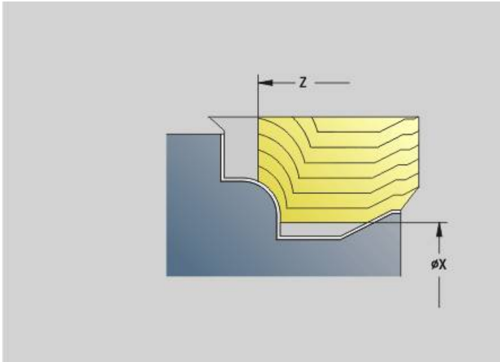
Parameters:

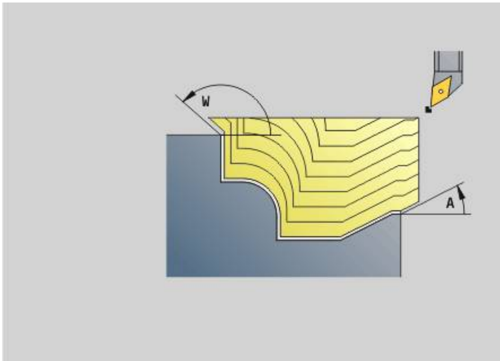
- **ID**: **Auxiliary contour** – ID number of the contour to be machined
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
  - **NE** not programmed: The contour element **NS** is machined in the direction of contour definition
  - **NS** = **NE** programmed: The contour element **NS** is machined opposite to the direction of contour definition.
- **P**: **Maximum infeed**
- **I**: **O-size X**
- **K**: **O-size Z**
- **X**: **Cutting limit in X** (diameter value; default: no cutting limit)
- **Z**: **Cutting limit in Z** (default: no cutting limit)
- **A**: **Start angle** (reference: Z axis; default: parallel to Z axis, or with facing tools: parallel to X axis)
- **W**: **Depart.angle** (reference: Z axis; default: orthogonal to Z axis, or with facing tools: orthogonal to X axis)
- **Q**: **Kind of liber.** at end of cycle
  - **0: Back to beg., X before Z**
  - **1: Before finished contour**
  - **2: Retract by safety clear.**
- **V**: **Machine form elements** (default: 0)

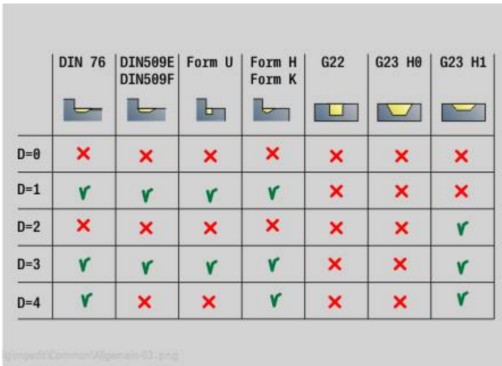
  A chamfer/rounding arc is machined
  - **0: At beginning and end**
  - **1: At beginning**
  - **2: At end**
  - **3: No machining**
  - **4: Only chamfer/rounding** is machined—not the basic element (requirement: the contour section consists of a single element)
- **D**: **Omit elements** (see figure)
- **B**: **Contour calculation**

  B: **Contour calculation**
  - **0: Automatic**
  - **1: Tool left (G41)**
  - **2: Tool right(G42)**

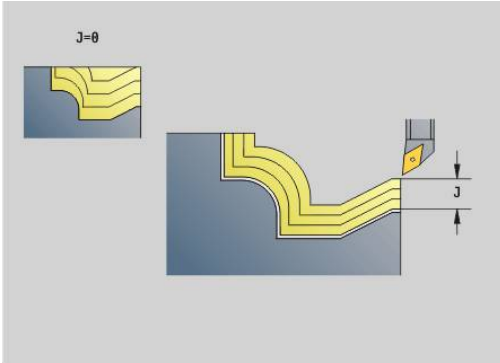


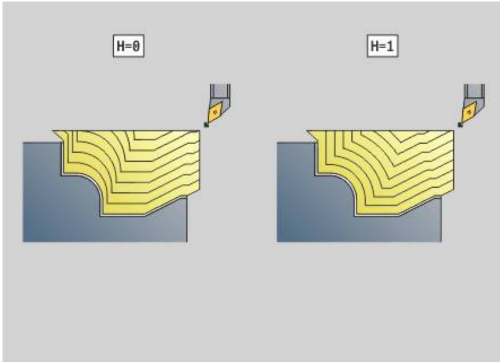






	DIN 76	DIN509E DIN509F	Form U	Form H Form K	G22	G23 H0	G23 H1
D=0	✗	✗	✗	✗	✗	✗	✗
D=1	✓	✓	✓	✓	✗	✗	✗
D=2	✗	✗	✗	✗	✗	✗	✓
D=3	✓	✓	✓	✓	✗	✗	✓
D=4	✓	✗	✗	✓	✗	✗	✓





- **H**: **Type of cut lines**
  - **0: Constant mach.depth** – contour is shifted by a constant infeed value (paraxial)
  - **1: Equidistant cut lines** – cutting lines run at a constant distance from the contour (contour-parallel). The contour is scaled.
- **RH**: **Wrkpc. blank contour** – evaluation only if no workpiece blank has been defined
  - **0:** (depends on defined parameters)
    - No parameters: workpiece blank from ICP contour and tool position
    - **XA** and **ZA**: workpiece blank from ICP contour and starting point of workpiece blank
    - **J**: workpiece blank from ICP contour and equidistant oversize
  - **1: From tool position** (workpiece blank from ICP contour and tool position)
  - **2: With blank start point** (workpiece blank from ICP contour and starting point of workpiece blank **XA** and **ZA**)
  - **3 Equidistant oversize** (workpiece blank from ICP contour and equidistant oversize **J**)
  - **4: Long.-transv. oversize** (workpiece blank from ICP contour, transverse oversize **XA** and longitudinal oversize **ZA**)
- **J**: **Workpiece blank oversize** (radius value – will only be evaluated if no workpiece blank has been defined)
- **XA**, **ZA**: **Start point blank** (Definition of the corner point of the workpiece blank contour—an evaluation will only take place if no workpiece blank has been defined beforehand)

The control uses the tool definition to distinguish between external and internal machining.

> - The tool radius compensation is active
> - A **G57** oversize enlarges the contour (also inside contours).
> - A **G58** oversize
>   - >0: Enlarges the contour
>   - <0: Is not offset
> - **G57**/**G58** oversizes are deleted after cycle end

Cycle run:

1. Calculates the areas to be machined and the cutting segmentation
2. Approaches workpiece for first pass from starting point, taking the safety clearance into account
3. Executes the first cut (roughing)
4. Returns at rapid traverse and approaches for next pass
5. Repeats 3 to 4 until the complete area has been machined
6. If required, repeats 2 to 5 until all areas have been machined
7. Retracts as programmed in **Q**

Contour cycle, bidirectional G835 (contour-parallel with neutral tool)

G835 machines the contour area defined in **ID**, or by **NS**, **NE**, parallel to the contour and bidirectionally.

Further information: "Working with contour-based cycles", Page 354

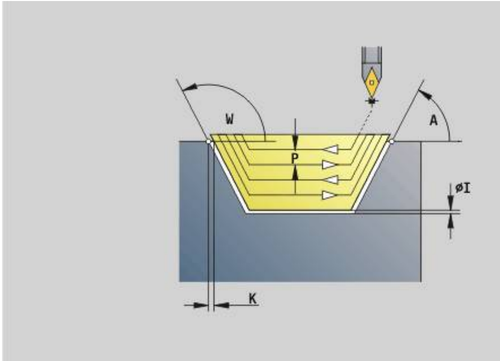
The contour to be machined can contain various valleys. If required, the area to be machined is divided into several sections.

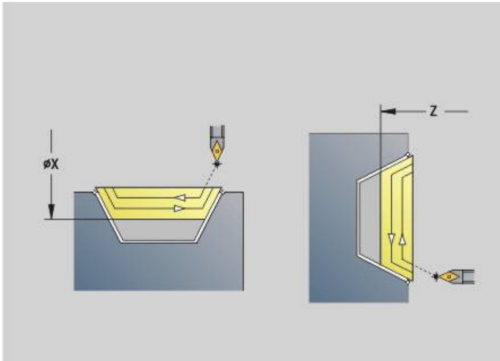
Parameters:

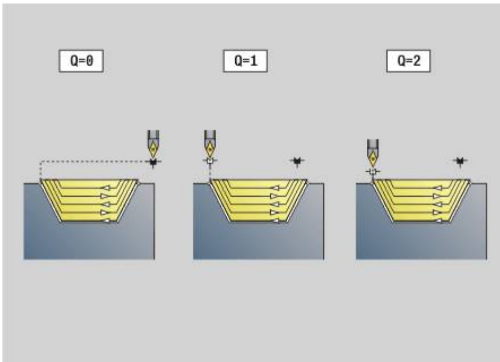
- **ID**: **Auxiliary contour** – ID number of the contour to be machined
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
  - **NE** not programmed: The contour element **NS** is machined in the direction of contour definition
  - **NS** = **NE** programmed: The contour element **NS** is machined opposite to the direction of contour definition.
- **P**: **Maximum infeed**
- **I**: **O-size X**
- **K**: **O-size Z**
- **X**: **Cutting limit in X** (diameter value; default: no cutting limit)
- **Z**: **Cutting limit in Z** (default: no cutting limit)
- **A**: **Start angle** (reference: Z axis; default: parallel to Z axis, or with facing tools: parallel to X axis)
- **W**: **Depart.angle** (reference: Z axis; default: orthogonal to Z axis, or with facing tools: orthogonal to X axis)
- **Q**: **Kind of liber.** at end of cycle
  - **0: Back to beg., X before Z**
  - **1: Before finished contour**
  - **2: Retract by safety clear.**
- **V**: **Machine form elements** (default: 0)

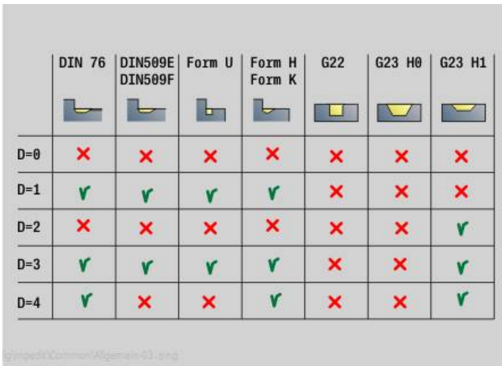
  A chamfer/rounding arc is machined
  - **0: At beginning and end**
  - **1: At beginning**
  - **2: At end**
  - **3: No machining**
  - **4: Only chamfer/rounding** is machined—not the basic element (requirement: the contour section consists of a single element)
- **B**: **Contour calculation**

  B: **Contour calculation**
  - **0: Automatic**
  - **1: Tool left (G41)**
  - **2: Tool right(G42)**
- **D**: **Omit elements** (see figure)



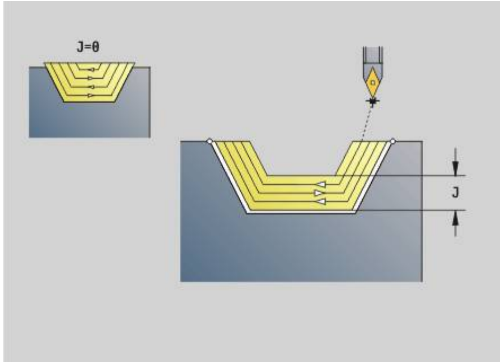


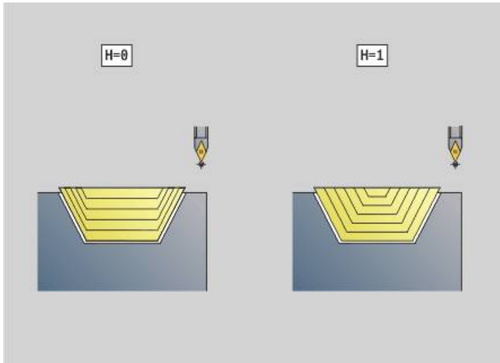




	DIN 76	DIN509E DIN509F	Form U	Form H Form K	G22	G23 H0	G23 H1
D=0	✗	✗	✗	✗	✗	✗	✗
D=1	✓	✓	✓	✓	✗	✗	✗
D=2	✗	✗	✗	✗	✗	✗	✓
D=3	✓	✓	✓	✓	✗	✗	✓
D=4	✓	✗	✗	✓	✗	✗	✓

- **H**: **Type of cut lines**
  - **0: Constant mach.depth** – contour is shifted by a constant infeed value (paraxial)
  - **1: Equidistant cut lines** – cutting lines run at a constant distance from the contour (contour-parallel). The contour is scaled.
- **RH**: **Wrkpc. blank contour** – evaluation only if no workpiece blank has been defined
  - **0:** (depends on defined parameters)
    - No parameters: workpiece blank from ICP contour and tool position
    - **XA** and **ZA**: workpiece blank from ICP contour and starting point of workpiece blank
    - **J**: workpiece blank from ICP contour and equidistant oversize
  - **1: From tool position** (workpiece blank from ICP contour and tool position)
  - **2: With blank start point** (workpiece blank from ICP contour and starting point of workpiece blank **XA** and **ZA**)
  - **3 Equidistant oversize** (workpiece blank from ICP contour and equidistant oversize **J**)
  - **4: Long.-transv. oversize** (workpiece blank from ICP contour, transverse oversize **XA** and longitudinal oversize **ZA**)
- **J**: **Workpiece blank oversize** (radius value – will only be evaluated if no workpiece blank has been defined)
- **XA**, **ZA**: **Start point blank** (Definition of the corner point of the workpiece blank contour—an evaluation will only take place if no workpiece blank has been defined beforehand)





The control uses the tool definition to distinguish between external and internal machining.

> - The tool radius compensation is active
> - A **G57** oversize enlarges the contour (also inside contours).
> - A **G58** oversize
>   - >0: Enlarges the contour
>   - <0: Is not offset
> - **G57**/**G58** oversizes are deleted after cycle end

Cycle run:

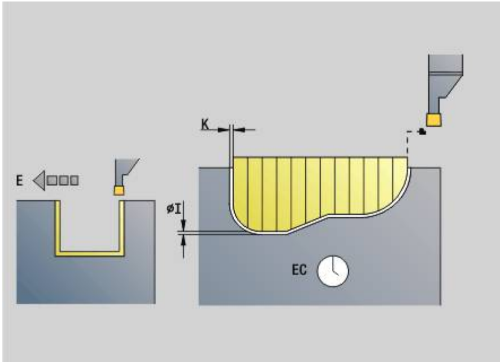
1. Calculates the areas to be machined and the cutting segmentation
2. Approaches workpiece for first pass from starting point, taking the safety clearance into account
3. Executes the first cut (roughing)
4. Approaches for the next pass and executes the next cut (roughing) in the opposite direction
5. Repeats 3 to 4 until the complete area has been machined
6. If required, repeats 2 to 5 until all areas have been machined
7. Retracts as programmed in **Q**

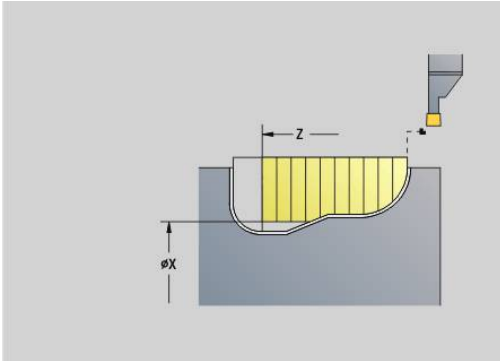
Recessing G860

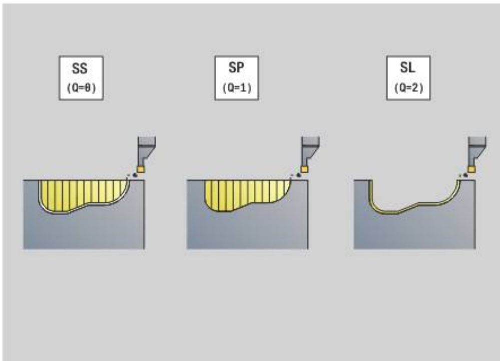
G860 machines the defined contour area. The reference to the contour to be machined can be transferred in the cycle parameters, or the contour can be defined directly after the cycle call.

Further information: "Working with contour-based cycles", Page 354

The contour to be machined can contain various valleys. If required, the area to be machined is divided into several sections.





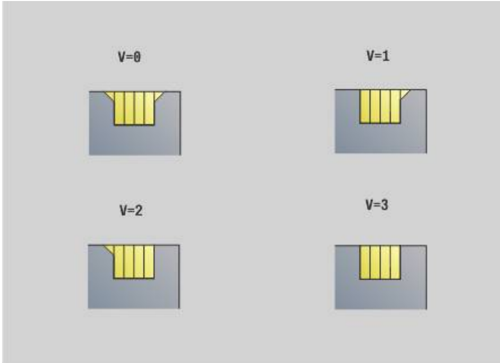


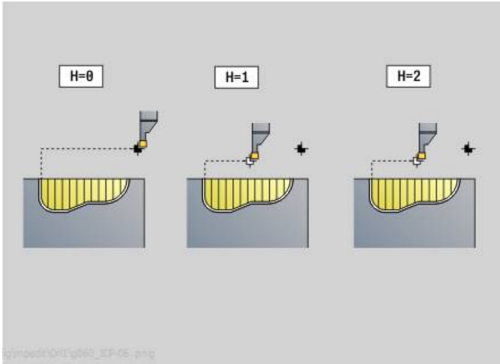
Parameters:

- **ID**: **Auxiliary contour** – ID number of the contour to be machined
- **NS**: **Starting block no. of contour** – beginning of contour section
  - Beginning of contour section
  - Reference to a **G22**/**G23**-Geo recess
- **NE**: **Contour end block no.** – end of contour section
  - **NE** not programmed: The contour element **NS** is machined in the direction of contour definition
  - **NS** = **NE** programmed: The contour element **NS** is machined opposite to the direction of contour definition.
- **I**: **O-size X**
- **K**: **O-size Z**
- **Q**: **Roughing/Finish** – procedure (default: 0)
  - **0: Roughing and finishing**
  - **1: Only roughing**
  - **2: Only finishing**
  - **3: multi.plunge + finishing** – rough grooving is performed by means of full-section cuts—the ridges in between are machined centrally relative to the recessing tool
  - **4: only multiple plunging**
- **X**: **Cutting limit in X** (diameter value; default: no cutting limit)
- **Z**: **Cutting limit in Z** (default: no cutting limit)
- **V**: **Machine form elements** (default: 0)

  A chamfer/rounding arc is machined
  - **0: At beginning and end**
  - **1: At beginning**
  - **2: At end**
  - **3: No machining**
- **E**: **Finishing feed**
- **EW**: **Recessing feed**
- **EC**: **Delay**
- **D**: **Rev. on recessing floor**

- **H**: **Kind of liber.** at end of cycle
  - **0: Back to start point**
    - Axial recess: First Z, then X direction
    - Radial recess: First X, then Z direction
  - **1: Before finished contour**
  - **2: Stops at safety clear.**
- **B**: **Cut. width**
- **P**: **Cut depth** – infeed depth for a single cut
- **O**: **End of rough cut**
  - **0: Lift-up at rapid**
  - **1: Half recessng width 45°**
- **U**: **End of finishing cut**
  - **0: Value from glob. param.**
  - **1: Parting horizntl. elem.**
  - **2: Complete horzntl. elem.**





The control uses the tool definition to distinguish between external and internal machining, or between radial and axial recesses.

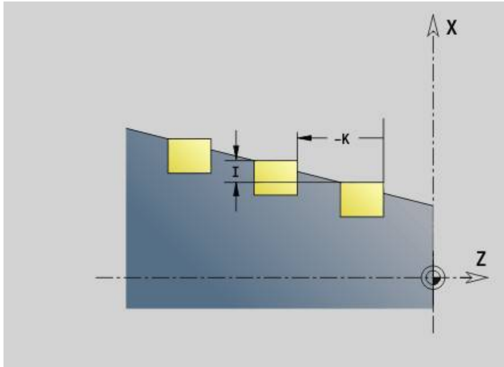
Recessing repeats can be programmed with **G741** before the cycle call.

> - The tool radius compensation is active
> - A **G57** oversize enlarges the contour (also inside contours).
> - A **G58** oversize
>   - >0: Enlarges the contour
>   - <0: Is not offset
> - **G57**/**G58** oversizes are deleted after cycle end

Cycle run:

1. Calculates the areas to be machined and the cutting segmentation
2. Approaches workpiece for first pass from starting point, taking the safety clearance into account
   - Radial recess: First Z, then X direction
   - Axial recess: First X, then Z direction
3. Executes first cut (roughing)
4. Returns at rapid traverse and approaches for next pass
5. Repeats 3 to 4 until the complete area has been machined
6. If required, repeats 2 to 5 until all areas have been machined
7. If **Q** = 0: Finish-machines the contour

Repeating recessing cycle G740

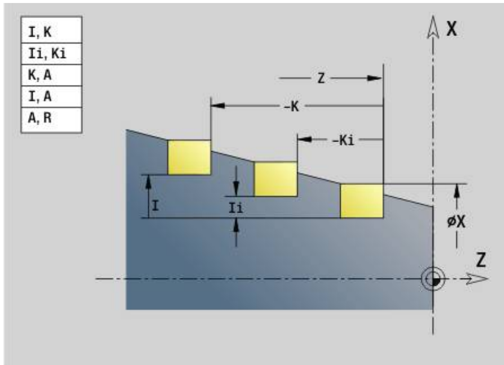


G740 is programmed before **G860** to repeat the recessing contour defined in Cycle **G860**.

Parameters:

- **X**: **Start pt. X** – shifts the starting point of the recessing contour defined by **G860** to this coordinate
- **Z**: **Start pt. Z** – shifts the starting point of the recessing contour defined by **G860** to this coordinate
- **I**: **Length** – distance between the starting points of the individual recessing contours (in X)
- **K**: **Length** – distance between the starting points of the individual recessing contours (in Z)
- **Q**: **Number** of recessing contours

Repeating recessing cycle G741



G741 is programmed before **G860** to repeat the recessing contour defined in Cycle **G860**.

Parameters:

- **X**: **Start pt. X** – shifts the starting point of the recessing contour defined by **G860** to this coordinate
- **Z**: **Start pt. Z** – shifts the starting point of the recessing contour defined by **G860** to this coordinate
- **I**: **Length** – distance between the first and last recessing contour (in X)
- **Ii**: **Length** – distance between the recessing contours (in X)
- **K**: **Length** – distance between the first and last recessing contour (in Z)
- **Ki**: **Length** – distance between the recessing contours (in Z)
- **Q**: **Number** of recessing contours
- **A**: **Angle** at which the recessing contours are arranged
- **R**: **Length** – distance between first and last recessing contour
- **Ri**: **Length** – distance between the recessing contours
- **O**: **Flow**
  - 0: Roughen all recesses then finish all recesses (default: previous behavior)
  - 1: Machine each recess completely before machining the next one

Example: Attributes in contour description G149

```
. . .
AUXILIARY CONTOUR ID"Recess"
N 47 G0 X50 Z0
N 48 G1 Z-5
N 49 G1 X45
N 54 G1 Z-15
N 56 G1 Z-17
MACHINING
N 162 T4
N 163 G96 S150 G95 F0.2 M3
N 165 G0 X120 Z100
N 166 G47 P2
N 167 G741 K-50 Q3 A180 O0
N 168 G860 I0.5 K0.2 E0.15 Q0 H0
N 172 G0 X50 Z0
N 173 G1 X40
N 174 G1 Z-9
N 175 G1 X50
N 169 G80
N 170 G14 Q0
. . .
```

The following parameter combinations are allowed:

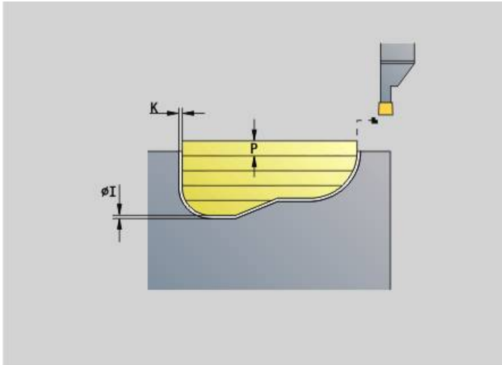
- **I**, **K**
- **Ii**, **Ki**
- **I**, **A**
- **K**, **A**
- **A**, **R**

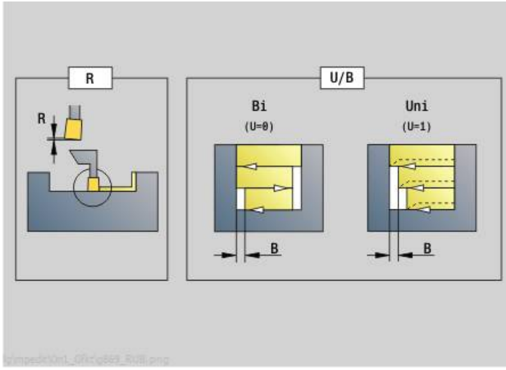
Recess turning cycle G869

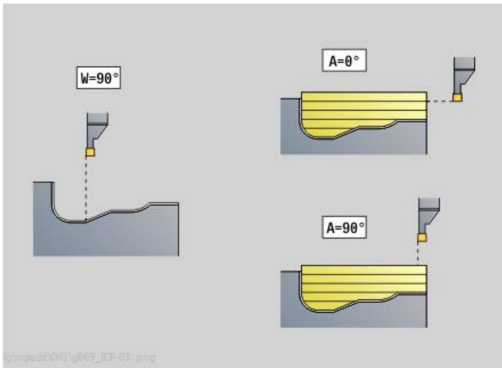
G869 machines the defined contour area. The reference to the contour to be machined can be transferred in the cycle parameters, or the contour can be defined directly after the cycle call.

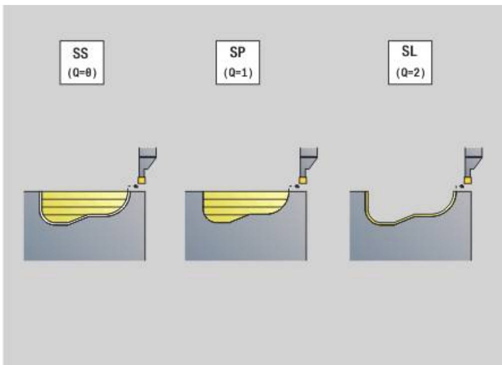
Further information: "Working with contour-based cycles", Page 354

The workpiece is machined by alternate recessing and roughing movements. The machining process requires a minimum of retraction and infeed movements. The contour to be machined can contain various valleys. If required, the area to be machined is divided into several sections.



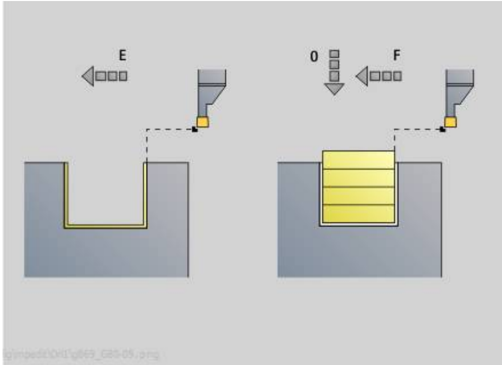






Parameters:

- **ID**: **Auxiliary contour** – ID number of the contour to be machined
- **NS**: **Starting block no. of contour** – beginning of contour section
  - Beginning of contour section
  - Reference to a **G22**/**G23**-Geo recess
- **NE**: **Contour end block no.** – end of contour section
  - **NE** not programmed: The contour element **NS** is machined in the direction of contour definition
  - **NS** = **NE** programmed: The contour element **NS** is machined opposite to the direction of contour definition.
- **P**: **Maximum infeed**
- **R**: **Depth compens.** for finishing (default: 0)
- **I**: **O-size X**
- **K**: **O-size Z**
- **X**: **Cutting limit in X** (diameter value; default: no cutting limit)
- **Z**: **Cutting limit in Z** (default: no cutting limit)
- **A**: **Start angle** (default: opposite to recessing direction)
- **W**: **Depart.angle** (default: opposite to recessing direction)
- **Q**: **Roughing/Finish** – procedure (default: 0)
  - **0: Roughing and finishing**
  - **1: Only roughing**
  - **2: Only finishing**
- **U**: **Unidir. turning** (default: 0)
  - **0: Bidirectional**
  - **1: Unidirectional**
- **H**: **Kind of liber.** at end of cycle
  - **0: Back to start point**
    - Axial recess: First Z, then X direction
    - Radial recess: First X, then Z direction
  - **1: Before finished contour**
  - **2: Stops at safety clear.**



- **V**: **Machine form elements** (default: 0)
  A chamfer/rounding arc is machined
  - **0: At beginning and end**
  - **1: At beginning**
  - **2: At end**
  - **3: No machining**
- **O**: **Recessing feed** (default: active feed rate)
- **E**: **Finishing feed**
- **B**: **Offset width** (default: 0)
- **XA**, **ZA**: **Start point blank** (Definition of the corner point of the workpiece blank contour—an evaluation will only take place if no workpiece blank has been defined beforehand)
  - **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

The control uses the tool definition to distinguish between radial and axial recesses.

Program at least one contour reference (e.g.: **NS** or **NS**, **NE**) and **P**.

Depth compens. R: Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with the turning depth compensation factor. The value is usually determined empirically.

Offset width B: After the second infeed movement, during the transition from turning to recessing, the length to be machined is reduced by **Offset width B**. For each other transition at this edge, the path is reduced by **B**—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 control reduces the programmed offset width. After precutting (rough grooving), the remaining material is machined with a single cut.

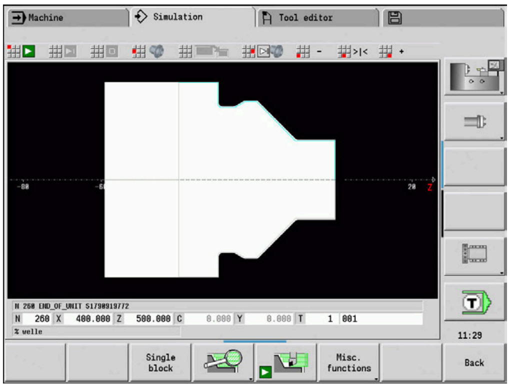
- The tool radius compensation is active
- A **G57** oversize enlarges the contour (also inside contours).
- A **G58** oversize
  - >0: Enlarges the contour
  - <0: Is not offset
- **G57**/**G58** oversizes are deleted after cycle end

Cycle run (with **Q** = 0 or 1):

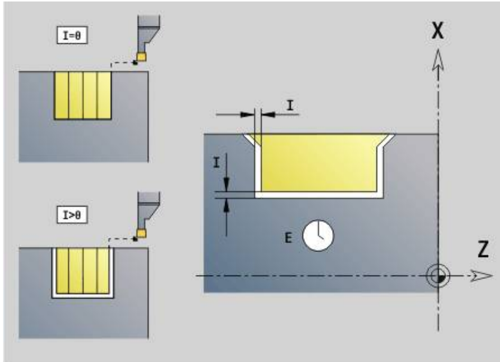
1 Calculates the areas to be machined and the cutting segmentation
2 Approaches workpiece for first pass from starting point, taking the safety clearance into account
   - Radial recess: First Z, then X direction
   - Axial recess: First X, then Z direction
3 Executes first cut (recessing)
4 Machines perpendicularly to recessing direction (turning)
5 Repeats 3 to 4 until the complete area has been machined
6 If required, repeats 2 to 5 until all areas have been machined
7 If **Q** = 0: Finish-machines the contour

Machining information

- Transition from turning to recessing: Before the transition from turning to recessing, the control retracts the tool by 0.1 mm. Thus, a tilted cutting edge will be adjusted for the recessing operation, independent of **Offset width B**
- Inside radii and chamfers: Depending on the recessing width and the radii of rounding arcs, single cuts preventing a fluid transition from recessing to turning are executed before the rounding is machined. This prevents damage to the tool.
- Edges: Edges are recessed. This prevents residual rings.



Recessing cycle G870



G870 generates a recess defined by **G22**-Geo. The control uses the tool definition to distinguish between external and internal machining, or between radial and axial recesses.

Parameters:

- **ID**: **Auxiliary contour** – ID number of the contour to be machined
- **NS**: **Starting block no. of contour** – references **G22**-Geo
- **I**: **Ov.size** for precutting (rough grooving) (default: 0)
  - **I** = 0: Recess is made in one work step
  - **I** > 0: The first operation is roughing, the second finishing
- **E**: **Delay** (default: time for one spindle revolution)
  - If **I** = 0: For every recess
  - If **I** > 0: Only for finishing

Calculation of cutting segmentation: Maximum offset = 0.8 * cutting width

> - The tool radius compensation is active
> - An oversize is not taken into account

Cycle run:

1. Calculates the number of cutting passes
2. Approaches workpiece from starting point for first pass
   - Radial recess: First Z, then X direction
   - Axial recess: First X, then Z direction
3. Executes the first cut according to **I**
4. Returns at rapid traverse and approaches for next pass
5. If **I** = 0: Dwells for time **E**
6. Repeats 3 to 4 until the complete recess has been machined
7. If **I** > 0: Finish-machines the contour

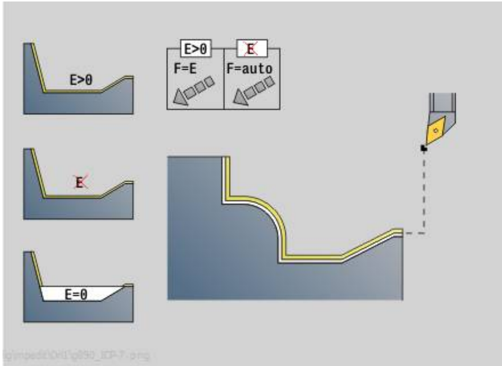
Contour finishing G890

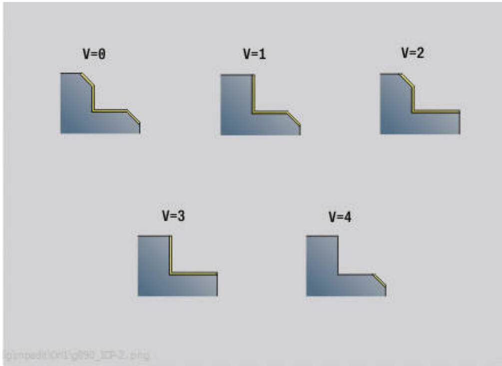
G890 finishes the defined contour area in one pass. The reference to the contour to be machined can be transferred in the cycle parameters, or the contour can be defined directly after the cycle call.

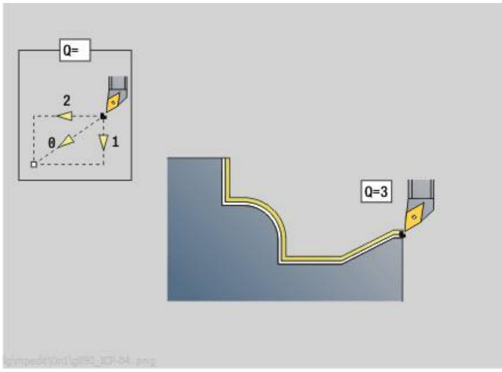
Further information: "Working with contour-based cycles", Page 354

The contour to be machined can contain various valleys. If required, the area to be machined is divided into several sections.

> In Machine Parameter 602322, you define whether the control is to check the usable length of the cutting edge during finishing. For recessing and button tools, the length of the cutting edge is not checked.







Parameters:

- **ID**: **Auxiliary contour** – ID number of the contour to be machined
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
  - **NE** not programmed: The contour element **NS** is machined in the direction of contour definition
  - **NS** = **NE** programmed: The contour element **NS** is machined opposite to the direction of contour definition.
- **E**: **Plunging behavior**
  - No input: Automatic feed-rate reduction
  - **E** = 0: No plunging
  - **E** > 0: Plunging feed rate in use
- **V**: **Machine form elements** (default: 0)

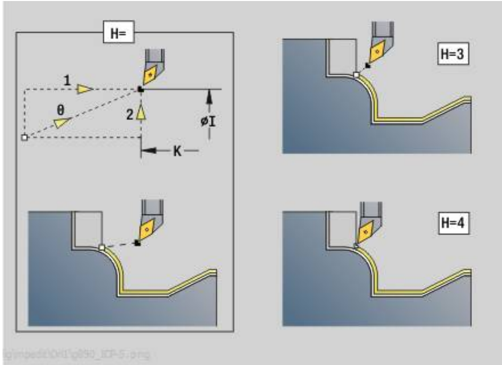
  A chamfer/rounding arc is machined
  - **0: At beginning and end**
  - **1: At beginning**
  - **2: At end**
  - **3: No machining**
  - **4: Only chamfer/rounding** is machined—not the basic element (requirement: the contour section consists of a single element)

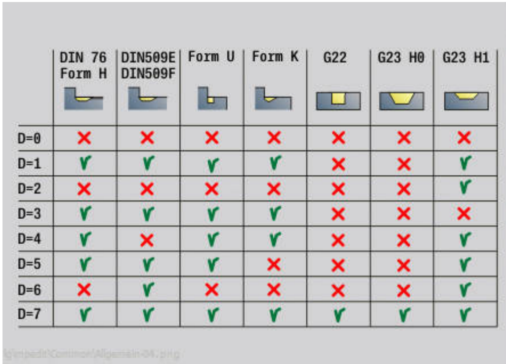
- **Q**: **Type of approach** (default: 0)
  - **0: Automatic** – the control checks:
    - Diagonal approach
    - First X, then Z direction
    - Equidistant around the workpiece blank, taking the safety clearance into account
    - Omission of the first contour elements if the starting position is inaccessible
  - **1: First X, then Z**
  - **2: First Z, then X**
  - **3: No approach** – tool is located near the starting point of the contour area
  - **4: Finish rem. mat.**
- **H**: **Type of retraction** – tool backs off at an angle below 45° opposite the machining direction and moves to the position **I**, **K** (default: 3)
  - **0: Same time, to I+K**
  - **1: First X then Z, to I+K**
  - **2: First Z then X, to I+K**
  - **3: Retract by safety clear.**
  - **4: No retraction motion** (tool remains on the end coordinate)
  - **5: Diagonal to start pos.**
  - **6: X then Z to start pos.**
  - **7: Z then X to start pos.**
- **X**: **Cutting limit in X** (diameter value; default: no cutting limit)
- **Z**: **Cutting limit in Z** (default: no cutting limit)
- **D**: **Omit elements** (see figure)

Codes for omitting recesses and undercuts

G call	Function	D code
G22	Recess for sealing ring	512
G22	Recess for circlip	1.024
G23 H0	General recess	256
G23 H1	Relief turn	2.048
G25 H4	Undercut type U	32.768
G25 H5	Undercut type E	65.536
G25 H6	Undercut type F	131.072
G25 H7	Undercut type G	262.144
G25 H8	Undercut type H	524.288
G25 H9	Undercut type K	1.048.576

Add the codes if you want to hide several elements





	DIN 76 Form H	DIN509E DIN509F	Form U	Form K	G22	G23 H0	G23 H1
D=0	✗	✗	✗	✗	✗	✗	✗
D=1	✓	✓	✓	✓	✗	✗	✓
D=2	✗	✗	✗	✗	✗	✗	✓
D=3	✓	✓	✓	✓	✗	✗	✗
D=4	✓	✗	✓	✓	✗	✗	✓
D=5	✓	✓	✓	✗	✗	✗	✓
D=6	✗	✓	✗	✗	✗	✗	✓
D=7	✓	✓	✓	✓	✓	✓	✓

- **I**: **Final point** that is approached at the end of the cycle (diameter value)
- **K**: **Final point** that is approached at the end of the cycle
- **O**: **Feed reduc. off** for circular elements (default: 0)
  - **0: No** (feed rate reduction is active)
  - **1: Yes** (feed rate reduction is not active)
- **U**: **Type of cycle** – required for generating the contour from the **G80** parameters (default: 0)
  - 0: standard contour (longitudinal or transverse), recessing contour or ICP contour
  - 1: Linear path without/with return
  - 2: Circular arc CW, without/with return
  - 3: Circular arc CCW, without/with return
  - 4: Chamfer without/with return
  - 5: Rounding arc without/with return
- **B**: **TRC/MRC switch on** – type of tool radius compensation
  - **0: Automatic**
  - **1: Tool left (G41)**
  - **2: Tool right(G42)**
  - **3: Automatic w/o tool compens.**
  - **4: W/o tool compens. left (G41)**
  - **5: W/o tool compens. left (G42)**
- **HR**: **Main machining direction**
  - **0: auto**
  - **1: +Z**
  - **2: +X**
  - **3: -Z**
  - **4: -X**

The control uses the tool definition to distinguish between external and internal machining.

Undercuts are machined if they are programmed and if tool geometry permits.

Feed rate reduction

For chamfers and rounding arcs:

- Feed rate is programmed with **G95**-Geo: No automatic feed rate reduction
- Feed rate is not programmed with **G95**-Geo: Automatic feed rate reduction. Each chamfer/rounding is therefore machined with at least three revolutions.
- For chamfers/rounding arcs which, as a result of their size, are machined with at least three revolutions, the feed rate is not reduced automatically.

For circular elements:

- For small circular elements, the feed rate is decreased until every element is machined with at least four spindle revolutions. You can switch this feed rate reduction off with **O**.
- The tool radius compensation (**TRC**) results under certain conditions to a feed rate reduction for circular elements. You can switch this feed rate reduction off with **O.**
  Further information: "Fundamentals", Page 341

- A **G57** oversize enlarges the contour (also inside contours).
- A **G58** oversize
  - >0: Enlarges the contour
  - <0: Reduces the contour
- **G57**/**G58** oversizes are deleted after cycle end

Simultaneous roughing G895 (option 54)

G891 enables three-axis simultaneous roughing of the defined contour area in several steps. This allows you to machine complex contours with only one tool.

During machining the cycle continuously adjusts the inclination of the tool based on the following criteria:

- Optimum inclination angle relative to the contour
- Prevention of collisions between the workpiece and the tool holder

In order to enable realistic collision monitoring through the cycle, you need to assign the corresponding tool holder to the tool being used.

The real holder must be within the defined holder dimensions.

In addition to the holder, the machine tool builder can also describe another component of the tilting axis as a collision object (e.g., the B-axis head). If this description is available as a 2-D view in the plane of rotation, then this object will be displayed in the 2-D simulation and will automatically be included in collision monitoring.

NOTICE

Danger of collision!

Collision monitoring is performed only in the two-dimensional machining plane XZ. The cycle does not check whether an area in the Y coordinate of the cutting edge, tool holder, or tilting body will lead to a collision.

- Perform a first run of the NC program in **Single block** mode
- Limit the machining area

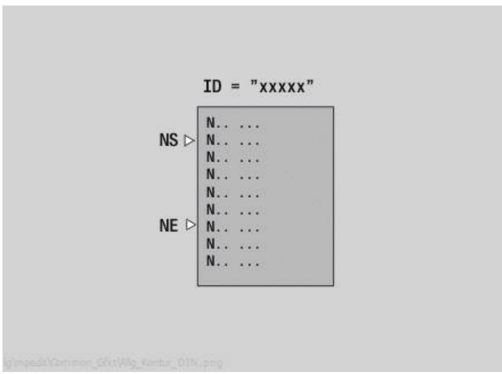
If the geometry of the cutting edge or collision monitoring requires an interruption of the cut, the tool is withdrawn and then re-positioned. The cycle uses turning tools and button tools.

Further information: "Working with contour-based cycles", Page 354

The control calculates the inclinations of the tilting axis from the minimum inclination angle, the maximum inclination angle, and the clearance angles (**IC**, **JC**, **KC**, and **RC**).

Parameters:

- **ID**: **Auxiliary contour** – ID number of the contour to be machined
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
  - **NE** not programmed: The contour element **NS** is machined in the direction of contour definition
  - **NS** = **NE** programmed: The contour element **NS** is machined opposite to the direction of contour definition.



- **V**: **Machine form elements** (default: 0)
  A chamfer/rounding arc is machined
  - **0: At beginning and end**
  - **1: At beginning**
  - **2: At end**
  - **3: No machining**
  - **4: Only chamfer/rounding** is machined—not the basic element (requirement: the contour section consists of a single element)
- **D**: **Omit elements** (see figure)

Codes for omitting recesses and undercuts

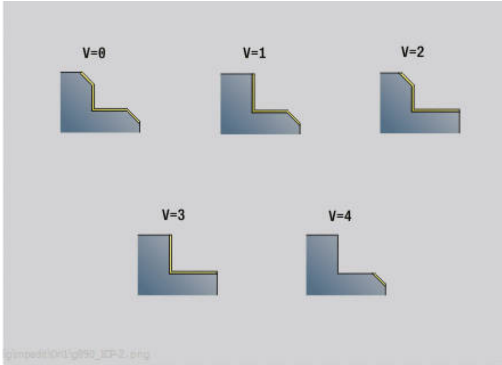
G call	Function	D code
G22	Recess for sealing ring	512
G22	Recess for circlip	1.024
G23 H0	General recess	256
G23 H1	Relief turn	2.048
G25 H4	Undercut type U	32.768
G25 H5	Undercut type E	65.536
G25 H6	Undercut type F	131.072
G25 H7	Undercut type G	262.144
G25 H8	Undercut type H	524.288
G25 H9	Undercut type K	1.048.576

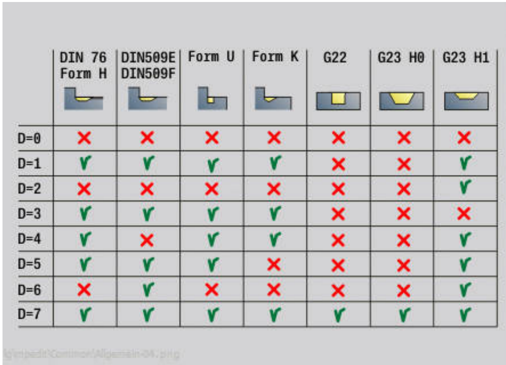
In order to omit multiple elements, you add the D codes from the table or use the D values from the graphic.

Example for omitting the undercut types **E** and **F**:

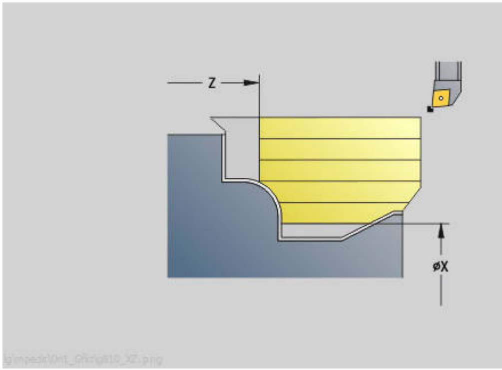
65.536 + 131.072 = 196.608

- **X**: **Cutting limit in X** (diameter value; default: no cutting limit)
- **Z**: **Cutting limit in Z** (default: no cutting limit)
- **A**: **Start angle** (reference: Z axis; default: parallel to Z axis)
  The line that is formed from the starting point of the contour area to be machined by using the **Start angle** must not intersect the contour of the finished part at any point.
- **W**: **Depart.angle** (reference: Z axis; default: orthogonal to Z axis)
  The line that is formed to the end point of the contour area to be machined by using the **Depart.angle** must not intersect the contour of the finished part at any point.

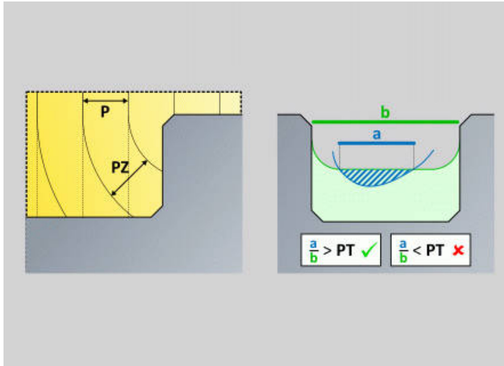




	DIN 76 Form H	DIN509E DIN509F	Form U	Form K	G22	G23 H0	G23 H1
D=0	✗	✗	✗	✗	✗	✗	✗
D=1	✓	✓	✓	✓	✗	✗	✓
D=2	✗	✗	✗	✗	✗	✗	✓
D=3	✓	✓	✓	✓	✗	✗	✗
D=4	✓	✗	✓	✓	✗	✗	✓
D=5	✓	✓	✓	✗	✗	✗	✓
D=6	✗	✓	✗	✗	✗	✗	✓
D=7	✓	✓	✓	✓	✓	✓	✓



Material removal:



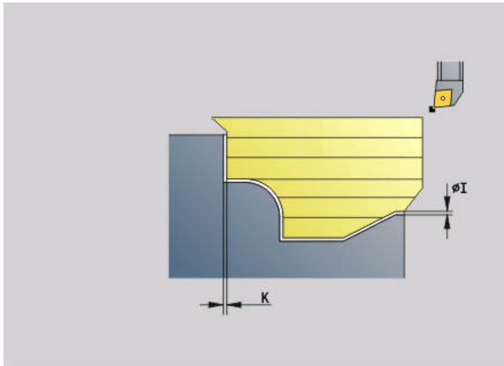
- **P**: **Desired infeed** – basis for calculating the infeed
- **PZ**: **Maximum infeed**

  The **Desired infeed P** is allowed to be exceeded up to the value specified in parameter **PZ** for a short time (e.g., for machining a corner). If the infeed exceeds the value specified in parameter **PZ**, then the control divides the corresponding area into several cuts.

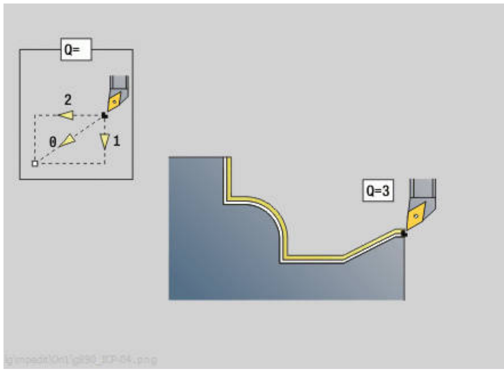
  No input: **PZ** corresponds to two thirds of the length of the cutting edge
- **PT**: **Minimum removal** – maintaining the infeed **P** in %

  If the material that can be removed by a cut in the cycle is less percent of the desired infeed specified in **P** than has been defined in **PT**, then the control skips this cut.

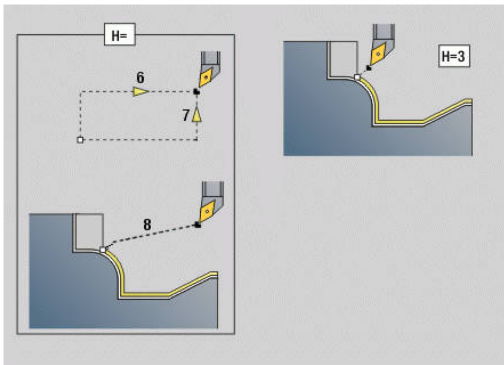
  If small values are defined in **PT** (e.g., **2 %**), then minimum cuts that remove much less material than the desired infeed **P** are performed at locations that are not easily accessible. The definition of larger values, such as **15 %**, saves machining time because contour areas that are not easily accessible are not machined completely.



- **I**: **O-size X**
- **K**: **O-size Z**



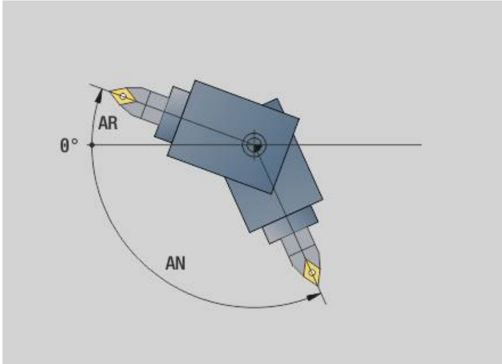
- **Q**: **Type of approach** (default: 0)
  - **0: automatic (with B)** – the control checks:
    - Diagonal approach
    - First X, then Z direction
    - Equidistant around the workpiece blank, taking the safety clearance into account
    - Omission of the first contour elements if the starting position is inaccessible
  - **1: First X, then Z**
  - **2: First Z, then X**
  - **3: No approach** – tool is located near the starting point of the contour area



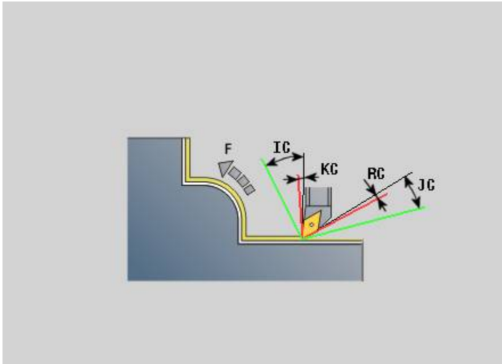
- **H**: **Kind of liber.**
  - **3: Retract by safety clear.**
  - **6: X then Z to start pos.**
  - **7: Z then X to start pos.**
  - **8: with B-axis motion to start pos.**

Dynamics:

- **AR**: **Minimum angle of incidence** – minimum permissible angle of the tilting axis (range: –359.999° < **AR** < 359.999°)
- **AN**: **Maximum angle of incidence** – maximum permissible angle of the tilting axis (range: –359.999° < **AN** < 359.999°)



Clearance angle (reference: finished part contour)



- **U** : **Use of soft clearance angles** – defines how the soft clearance angles **IC** and **JC** are used

  The parameter **Use of soft clearance angles** allows you to adapt the motion dynamics of the cycle. Based on the defined clearance angles, **U** influences the positioning of the tilting axis depending on the selected setting.

  The parameter **U Use of soft clearance angles** provides the following settings options:

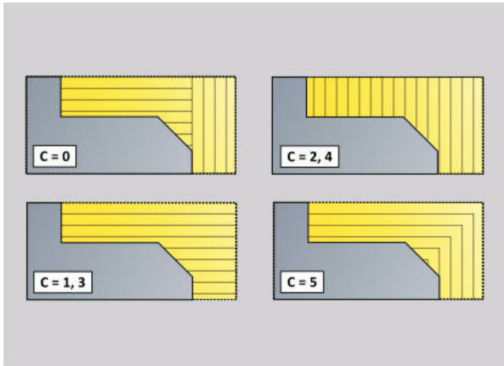
  - **0: very hard**
  - **1: hard**
  - **2: medium**
  - **3: soft**
  - **4: very soft**

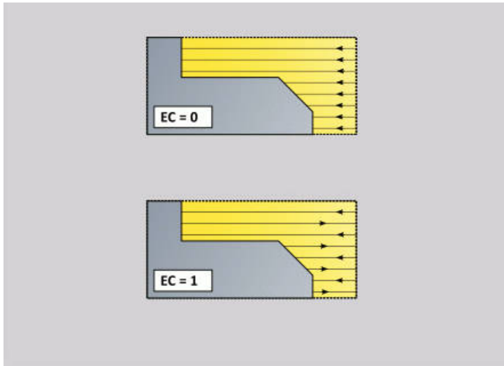
  The setting **0: very hard** leads to large compensating movements of the tilting axis because the angular tolerance of the tool is smaller. In contrast, with the setting **4: very soft**, the tilting axis performs small compensating movements because the angular tolerance of the tool is greater.
- **IC**: **Primary clearance angle - soft**: Desired clearance area in front of the cutting edge
- **JC** : **Secondary clearance angle - soft**: Desired clearance area behind the cutting edge
- **KC**: **Primary clearance angle - hard**: Safe clearance area in front of the cutting edge
- **RC**: **Primary clearance angle - hard**: Safe clearance area behind the cutting edge

> The hard clearance angles that you have defined must be complied with during machining (do not go below the defined values). If the hard clearance angles defined for machining a contour cannot be complied with, the control will output an error message.
>
> In addition to the hard clearance angles, you can use the soft clearance angles to define a desired angle range for machining. The control takes into account the soft clearance angles for path calculation. During machining the hard clearance angles are complied with where possible. The soft clearance angles do not have to be complied with during machining.

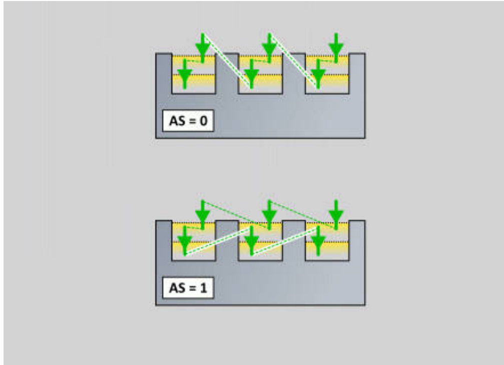
Strategy:

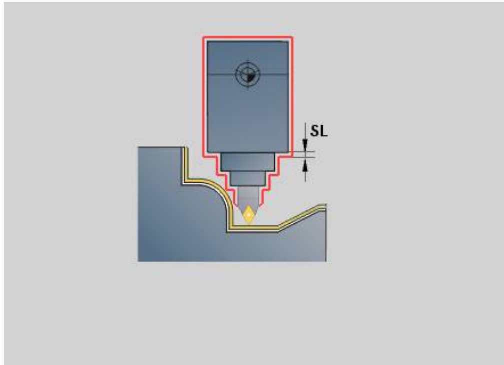
- **C**: **Cutting strategy** – type of cut lines (cutting paths)
  - **0: automatic** – the control automatically combines transverse and longitudinal turning
  - **1: longitudinal (outside)**
  - **2: transverse (front)**
  - **3: longitudinal (inside)**
  - **4: transverse (back)**
  - **5: parallel to blank**
- **EC**: **Cutting direction**
  - **0: Unidirectional** – every cut is performed in the direction of contour definition
  - **1: Bidirectional** – the cuts are performed along the optimum cutting path with respect to the machining time, and can be performed in the direction or opposite to the direction of contour definition
- **AS**: **Strategy for sequence** – machining sequence for separate pockets
  - **0: transverse (preferred)** – the machining sequence is chosen so that the center of gravity of the workpiece is always as close to the chucking equipment as possible
  - **1: longitudinal (preferred)** – the machining sequence is chosen so that the moment of inertia of the workpiece is as low as possible
- **SL**: **Oversize of tool holder** – oversize for calculating collisions between the workpiece and the tool holder
- **E**: **Fmax in compensating motion** – speed limit during the compensation movement of the linear axes
- **EW**: **Plunging feed rate** – feed rate for plunging into the material in mm/min

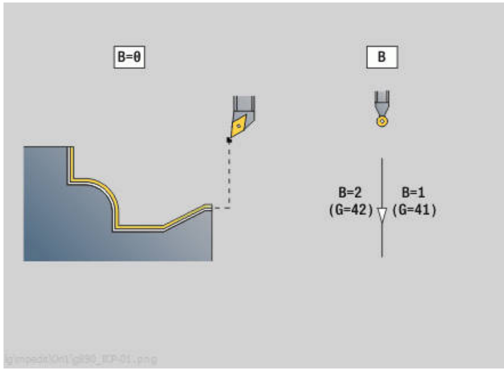




- **B**: Contour calculation
  - **0: Automatic**
  - **1: Tool left (G41)**
  - **2: Tool right(G42)**







Simultaneous finishing G891 (option 54)

G891 finishes the defined contour area simultaneously in three axes in one pass.

During machining, the cycle continuously adjusts the tool angle of inclination based on the following criteria:

- Optimum inclination angle relative to the contour
- Avoidance of collisions between the workpiece and collision bodies

This enables flexible machining of complex contours with a single tool.

> In order to enable realistic collision monitoring through the cycle, you need to assign the corresponding tool holder to the tool being used.
>
> The real holder must be within the defined holder dimensions.
>
> In addition to the holder, the machine tool builder can also describe another component of the tilting axis as a collision object (e.g., the B-axis head). If this description is available as a 2-D view in the plane of rotation, then this object will be displayed in the 2-D simulation and will automatically be included in collision monitoring.

NOTICE

Danger of collision!

Collision monitoring is performed only in the two-dimensional machining plane X-Z. The cycle does not check whether an area in the Y coordinate of the cutting edge, tool holder, or tilting body will lead to a collision.

- Verify the NC program in **Single Block**
- Limit the machining area

If the geometry of the cutting edge or collision monitoring requires an interruption of the cut, the machining operation is stopped and then resumed. The cycle uses turning tools and button tools. Recessing tools are not permitted in this cycle.

Further information: "Working with contour-based cycles", Page 354

The control calculates the inclinations of the tilting axis from the values entered for the minimum inclination angle, the maximum inclination angle, and the clearance angles (**IC**, **JC**, **KC**, and **RC**).

> In machine parameter **checkCuttingLength** (no. 602322), you can define whether the control is to check the usable length of the cutting edge during finishing. For button tools, the default setting does not check the length of the cutting edge.

Parameters:

- **ID**: **Auxiliary contour** – ID number of the contour to be machined
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
  - **NE** not programmed: The contour element **NS** is machined in the direction of contour definition
  - **NS** = **NE** programmed: The contour element **NS** is machined opposite to the direction of contour definition.
- **V**: **Machine form elements** (default: 0)

  A chamfer/rounding arc is machined
  - **0: At beginning and end**
  - **1: At beginning**
  - **2: At end**
  - **3: No machining**
  - **4: Only chamfer/rounding** is machined—not the basic element (requirement: the contour section consists of a single element)
- **D**: **Omit elements** (see figure)

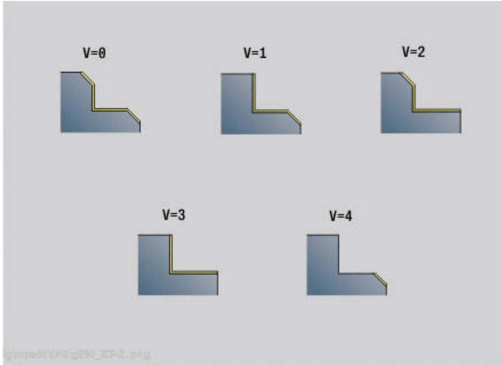
  Codes for omitting recesses and undercuts

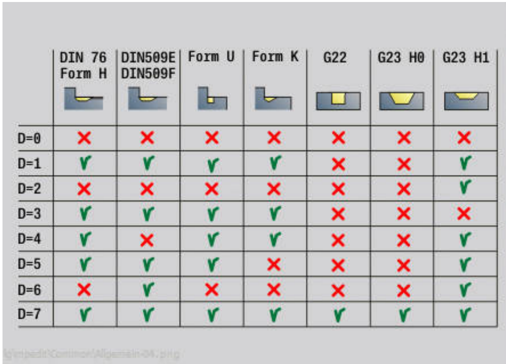
G call	Function	D code
G22	Recess for sealing ring	512
G22	Recess for circlip	1,024
G23 H0	General recess	256
G23 H1	Relief turn	2,048
G25 H4	Undercut type U	32,768
G25 H5	Undercut type E	65,536
G25 H6	Undercut type F	131,072
G25 H7	Undercut type G	262,144
G25 H8	Undercut type H	524,288
G25 H9	Undercut type K	1,048,576

  In order to omit multiple elements, you add the D codes from the table or use the D values from the graphic.

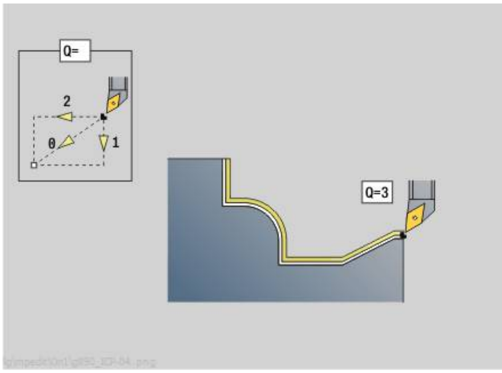
  Example for omitting the undercut types **E** and **F**:

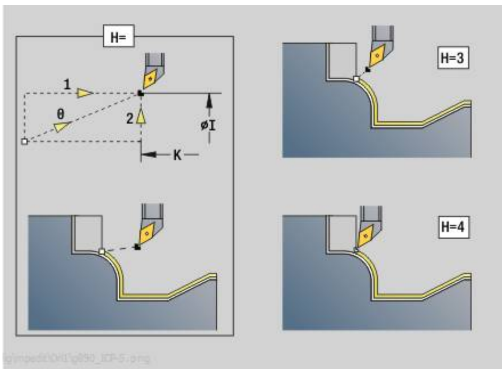
  65.536 + 131.072 = 196.608
- **Q**: **Type of approach** (default: 0)
  - **0: automatic (with B)** – the control checks:
    - Diagonal approach
    - First X, then Z direction
    - Equidistant around the workpiece blank, taking the safety clearance into account
    - Omission of the first contour elements if the starting position is inaccessible
  - **1: First X, then Z**
  - **2: First Z, then X**
  - **3: No approach** – tool is located near the starting point of the contour area



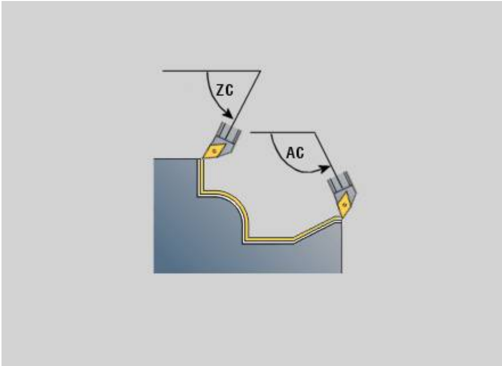


	DIN 76 Form H	DIN509E DIN509F	Form U	Form K	G22	G23 H0	G23 H1
D=0	✗	✗	✗	✗	✗	✗	✗
D=1	✓	✓	✓	✓	✗	✗	✓
D=2	✗	✗	✗	✗	✗	✗	✓
D=3	✓	✓	✓	✓	✗	✗	✗
D=4	✓	✗	✓	✓	✗	✗	✓
D=5	✓	✓	✓	✗	✗	✗	✓
D=6	✗	✓	✗	✗	✗	✗	✓
D=7	✓	✓	✓	✓	✓	✓	✓



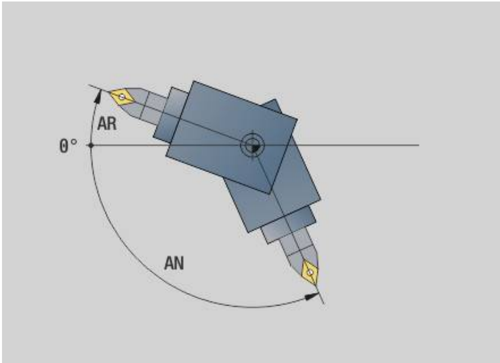


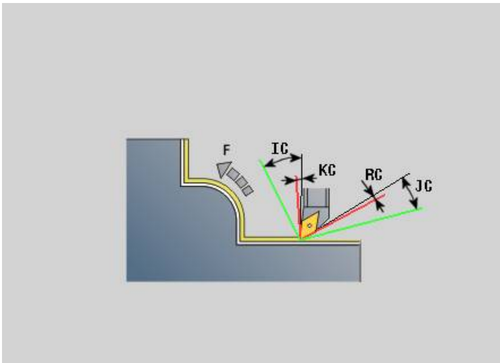
- **H** : **Kind of liber.**
  - **3: Retract by safety clear.**
  - **4: No retraction motion** (tool remains on the end coordinate)
  - **5: Diagonal to start pos.**
  - **6: X then Z to start pos.**
  - **7: Z then X to start pos.**
  - **8: with B-axis motion to start pos.**
- **AC**: **B angle at starting point** – inclined tilt angle at the beginning of the contour (range: 0° < **AC** < 360°)
- **ZC**: **B angle at end point** – inclined tilt angle at the end of the contour (range: 0° < **ZC** < 360°)

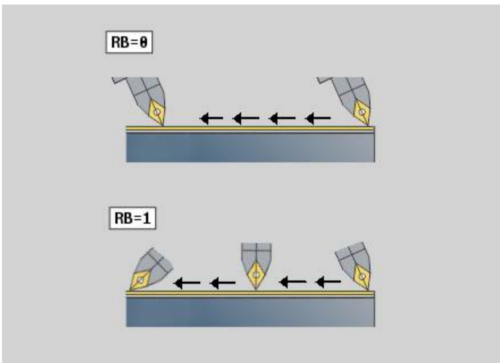


Dynamics:

- **AR**: **Minimum angle of incidence** – minimum permissible angle of the tilting axis (range: –359.999° < **AR** < 359.999°)
- **AN**: **Maximum angle of incidence** – maximum permissible angle of the tilting axis (range: –359.999° < **AN** < 359.999°)
- **U** : **Use of soft clearance angles** – defines how the soft clearance angles **IC** and **JC** are used
  - **0: very hard**
  - **1: hard**
  - **2: medium**
  - **3: soft**
  - **4: very soft**
- **RB** : **Roll over** – uniform wear of cutting edge by controlling the inclination angle
  - **0: No**
  - **1: Yes**







Clearance angle: (reference: finished part contour)

- **IC**: **Primary clearance angle - soft**: Desired clearance area in front of the cutting edge
- **JC** : **Secondary clearance angle - soft**: Desired clearance area behind the cutting edge
- **KC**: **Primary clearance angle - hard**: Safe clearance area in front of the cutting edge
- **RC**: **Primary clearance angle - hard**: Safe clearance area behind the cutting edge

> If you have defined hard clearance angles, they must be complied with during machining (do not go below the defined values). If the hard clearance angles defined for machining a contour cannot be complied with, the control will output an error message.
>
> In addition to the hard clearance angles, you can use the soft clearance angles to define a desired angle range for machining. The control takes into account the soft clearance angles during path calculation and preferably performs machining within the defined angle range. The soft clearance angles do not have to be complied with during machining.

- **O**: **Feed reduc. off** (default: 0)
  - **0: No** (feed rate reduction is active)
  - **1: Yes** (feed rate reduction is not active)

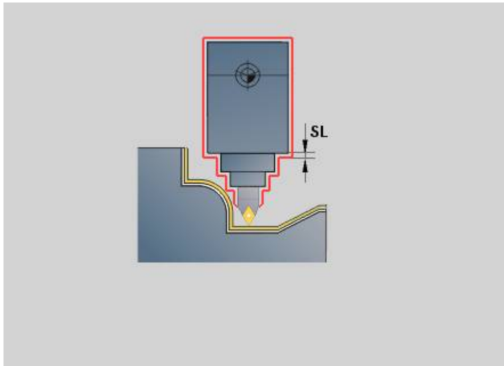
> If contour elements cannot be machined at the programmed feed rate due to their size, then the control reduces the feed rate during machining even if feed-rate reduction is not active. This ensures dimensionally accurate production of contours.
>
> With active feed-rate reduction, you only need a minimum number of spindle revolutions for machining a contour element.
>
> Machine parameter **fmur** (no. 602321) enables you to define the minimum number of spindle revolutions for machining a contour element.

- **B**: **Contour calculation**

  B: **Contour calculation**
  - **0: Automatic**
  - **1: Tool left (G41)**
  - **2: Tool right(G42)**
- **X**: **Cutting limit in X** (diameter value; default: no cutting limit)
- **Z**: **Cutting limit in Z** (default: no cutting limit)
- **A**: **Start angle** (reference: Z axis; default: parallel to Z axis)

  The line that is formed from the starting point of the contour area to be machined by using the **Start angle** must not intersect the contour of the finished part at any point.
- **W**: **Depart.angle** (reference: Z axis; default: orthogonal to Z axis)

  The line that is formed to the end point of the contour area to be machined by using the **Depart.angle** must not intersect the contour of the finished part at any point.
- **SL**: **Oversize of tool holder** – oversize for calculating collisions between the workpiece and the tool holder
- **E**: **Fmax in compensating motion** – speed limit during the compensation movement of the linear axes



- A **G57** oversize enlarges the contour (also inside contours).
- A **G58** oversize
  - >0: Enlarges the contour
  - <0: Reduces the contour
- **G57**/**G58** oversizes are deleted after cycle end

Measuring path G809

Cycle **G809** performs a cylindrical measuring cut with the length defined in the cycle, moves to the breakpoint for measuring and stops the program. After the program was stopped, you can manually measure the workpiece.

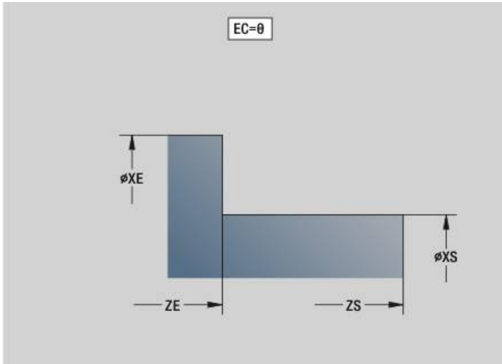
Parameters:

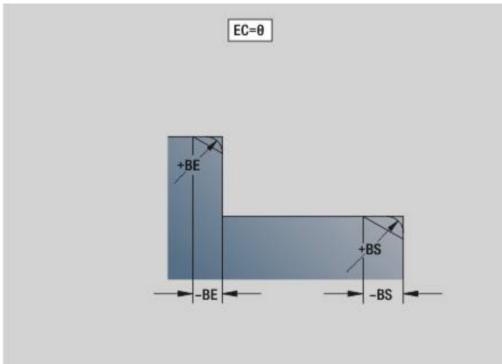
- **X**: **Start point** X
- **Z**: **Start point** Z
- **R**: **Length of measuring cut**
- **P**: **Measuring cut oversize**
- **I**: **Breakpoint Xi for measuring** – incremental distance to starting point of measurement
- **K**: **Breakpoint Zi for measuring** – incremental distance to starting point of measurement
- **ZS**: **Start point blank** – collision-free approach for inside machining
- **XE**: **Departure position X**
- **D**: **Additive correction** (numbers: 1 to 16)
- **V**: **Measuring cut counter** – number of workpieces after which a measurement is performed
- **Q**: **Mach. direction** (default: 0)
  - **0: -Z**
  - **1: +Z**
- **EC**: **Machining location**
  - **1: Outside**
  - **-1: Inside**
- **WE**: **Type of approach**
  - **0: Simultaneously**
  - **1: First X, then Z**
  - **2: First Z, then X**
- **O**: **Approach angle**
  If an approach angle is entered, the cycle positions the tool over the starting point taking into account the safety clearance, and from there plunges at the specified angle to the diameter to be measured.

6.18 Contour definitions in the machining section

Cycle end / simple contour G80

By programming **G80** (with parameters), a turning contour consisting of more than one element can be defined in one NC block. **G80** (without parameters) ends a contour definition directly after a cycle.





Parameters:

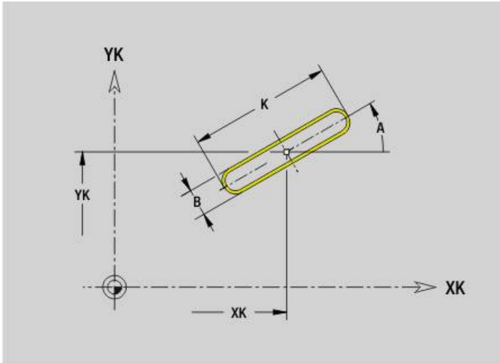
- **XS**: **Start point** of contour in X (diameter value)
- **ZS**: **Start point** of contour in Z
- **XE**: **Final point** of contour in X (diameter value)
- **ZE**: **Final point** of contour in Z
- **AC**: **Angle** of first element (range: 0° <= **AC** < 90°)
- **WC**: **Angle** of second element (range: 0° <= **WC** < 90°)
- **BS**: **-Chamfer/+radius at start**
- **WS**: **Angle for chamfer**
- **BE**: **-Chamfer/+radius at end**
- **WE**: **Angle for chamfer** at contour end
- **RC**: **Radius**
- **IC**: **Chamfer width**
- **KC**: **Chamfer width**
- **JC**: **Execution**
  - 0: Simple contour
  - 1: Expanded contour
- **EC**: **Type of contour**
  - 0: Rising contour
  - 1: Plunging contour
- **HC**: **1: Transverse** – contour direction for finishing
  - 0: Longitudinal
  - 1: Transverse

The control uses **IC** and **KC** internally to represent the chamfer/rounding cycles.

Example: G80

N1 T3 G95 F0.25 G96 S200 M3	
N2 G0 X120 Z2	
N3 G810 P3	
N4 G80 XS60 ZS-2 XE90 ZE-50 BS3 BE-2 RC5	
N5 ...	
N6 G0 X85 Z2	
N7 G810 P5	
N8 G0 X0 Z0	
N9 G1 X20	
N10 G1 Z-40	
N11 G80	

Linear slot on front/rear face G301

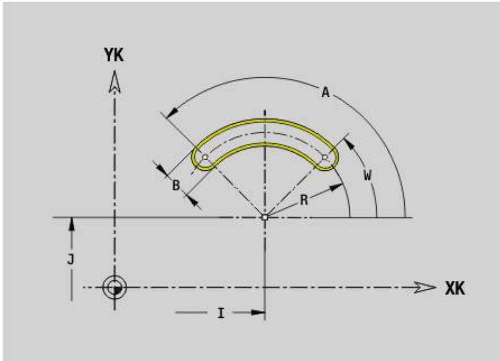


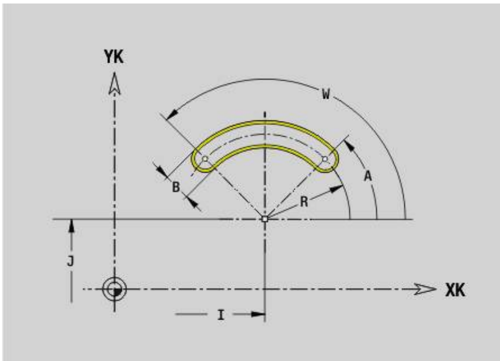
G301 defines a linear slot in a contour on the front or rear face. Program this figure in conjunction with **G840**, **G845** or **G846**.

Parameters:

- **XK**: **Center** (in Cartesian coordinates)
- **YK**: **Center** (in Cartesian coordinates)
- **X**: **Diameter** – **Center** (in polar coordinates)
- **C**: **Angle** – **Center** (in polar coordinates)
- **A**: **Angle** to XK axis (default: 0°)
- **K**: **Length**
- **B**: **Width**
- **P**: **Depth/Height** – depth for pockets, height for islands
  - **P** < 0: Pocket
  - **P** > 0: Island

Circular slot on front/rear face G302/G303





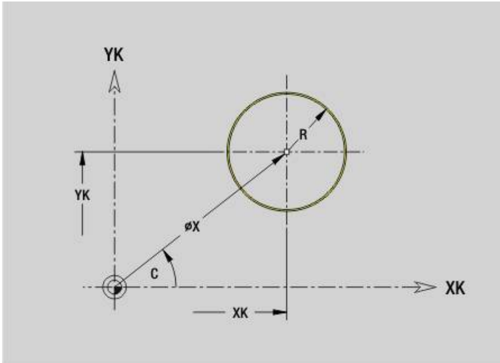
G302 and **G303** define a circular slot in a contour on the front face/ rear face. Program this figure in conjunction with **G840**, **G845** or **G846**.

- **G302**: Circular slot clockwise
- **G303**: Circular slot counterclockwise

Parameters:

- **I**: **Center** (in Cartesian coordinates)
- **J**: **Center** (in Cartesian coordinates)
- **X**: **Diameter** – **Center** (in polar coordinates)
- **C**: **Angle** – **Center** (in polar coordinates)
- **A**: **Angle** to XK axis (default: 0°)
- **W**: **Final angle** to XK axis (default: 0)
- **B**: **Width**
- **P**: **Depth/Height** – depth for pockets, height for islands
  - **P** < 0: Pocket
  - **P** > 0: Island

Full circle on front/rear face G304

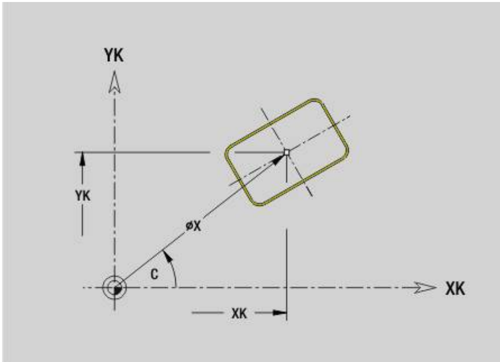


G304 defines a full circle in a contour on the front face/rear face. Program this figure in conjunction with **G840**, **G845** or **G846**.

Parameters:

- **XK**: **Center** (in Cartesian coordinates)
- **YK**: **Center** (in Cartesian coordinates)
- **X**: **Diameter** – **Center** (in polar coordinates)
- **C**: **Angle** – **Center** (in polar coordinates)
- **R**: **Radius**
- **P**: **Depth/Height** – depth for pockets, height for islands
  - **P** < 0: Pocket
  - **P** > 0: Island

Rectangle on front/rear face G305

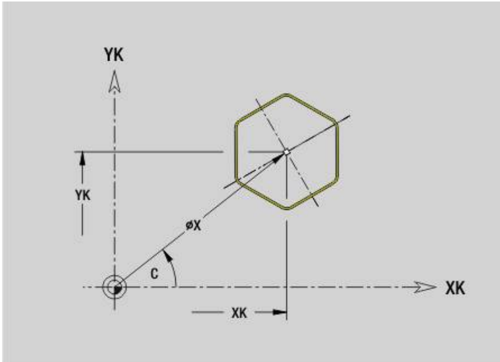


G305 defines a rectangle in a contour on the front face/rear face. Program this figure in conjunction with **G840**, **G845** or **G846**.

Parameters:

- **XK**: **Center** (in Cartesian coordinates)
- **YK**: **Center** (in Cartesian coordinates)
- **X**: **Diameter** – **Center** (in polar coordinates)
- **C**: **Angle** – **Center** (in polar coordinates)
- **A**: **Angle** to XK axis (default: 0°)
- **K**: **Length**
- **B**: **Height** of rectangle
- **R**: **Chamf./round.** (default: 0)
  - **R** > 0: Radius of rounding arc
  - **R** < 0: Chamfer width
- **P**: **Depth/Height** – depth for pockets, height for islands
  - **P** < 0: Pocket
  - **P** > 0: Island

Polygon on front/rear face G307

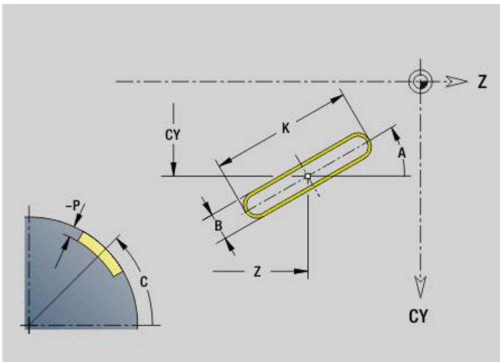


G307 defines a polygon in a contour on the front face/rear face. Program this figure in conjunction with **G840**, **G845** or **G846**.

Parameters:

- **XK**: **Center** (in Cartesian coordinates)
- **YK**: **Center** (in Cartesian coordinates)
- **X**: **Diameter** – **Center** (in polar coordinates)
- **C**: **Angle** – **Center** (in polar coordinates)
- **A**: **Angle** to XK axis (default: 0°)
- **Q**: **Number edges**
- **K**: **+edge lngth/-width a. flats**
  - **K** > 0: Edge length
  - **K** < 0: Width across (Inside diameter)
- **R**: **Chamf./round.** (default: 0)
  - **R** > 0: Radius of rounding arc
  - **R** < 0: Chamfer width
- **P**: **Depth/Height** – depth for pockets, height for islands
  - **P** < 0: Pocket
  - **P** > 0: Island

Linear slot on lateral surface G311

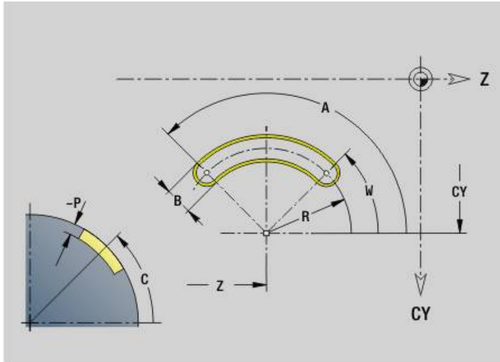


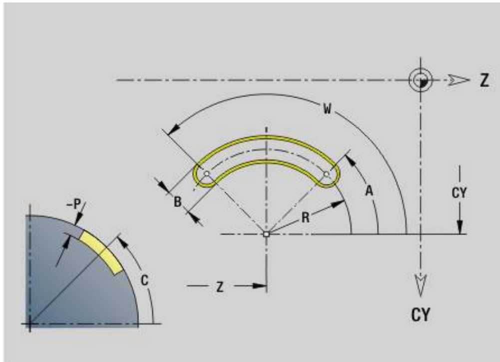
G311 defines a linear slot in a lateral-surface contour. Program this figure in conjunction with **G840**, **G845** or **G846**.

Parameters:

- **Z**: **Center**
- **CY**: **Center** as a linear dimension (reference: unrolled lateral surface, using the **Reference diameter**)
- **C**: **Center** (angle)
- **A**: **Angle to Z axis** (default: 0°)
- **K**: **Length**
- **B**: **Width**
- **P**: **Depth**

Circular slot on lateral surface G312/G313



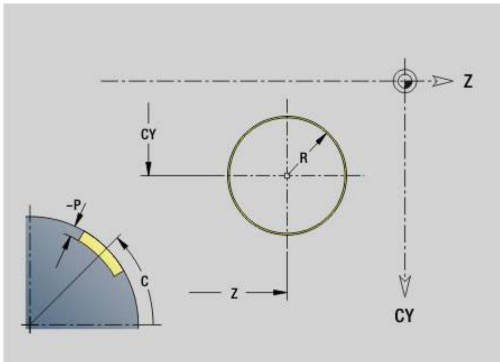


G312 and **G313** define a circular slot in a lateral-surface contour. Program this figure in conjunction with **G840**, **G845** or **G846**.

Parameters:

- **Z**: **Center**
- **CY**: **Center** as a linear dimension (reference: unrolled lateral surface, using the **Reference diameter**)
- **C**: **Center** (angle)
- **R**: **Radius**
- **A**: **Start angle**
- **W**: **End angle** (reference: Z axis)
- **B**: **Width**
- **P**: **Depth**

Full circle on lateral surface G314



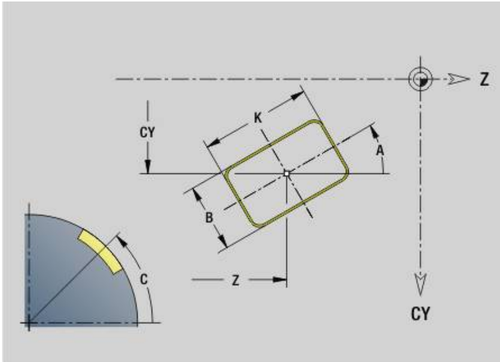
G314 defines a full circle in a lateral-surface contour. Program this figure in conjunction with **G840**, **G845** or **G846**.

Parameters:

- **Z**: **Center**
- **CY**: **Center** as a linear dimension (reference: unrolled lateral surface, using the **Reference diameter**)
- **C**: **Center** (angle)
- **R**: **Radius**
- **P**: **Depth**

Rectangle, surface G315

G315 defines a rectangle in a lateral-surface contour. Program this figure in conjunction with **G840**, **G845** or **G846**.

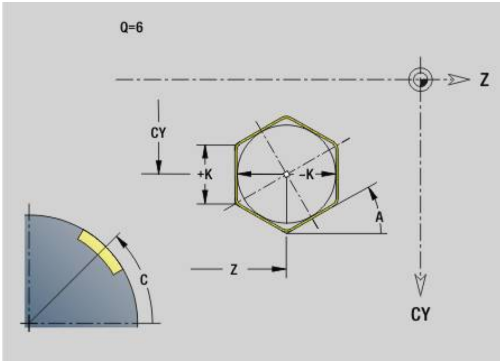


Parameters:

- **Z**: **Center**
- **CY**: **Center** as a linear dimension (reference: unrolled lateral surface, using the **Reference diameter**)
- **C**: **Center** (angle)
- **A**: **Angle to Z axis** (default: 0°)
- **K**: **Length** of rectangle
- **B**: **Height** of rectangle
- **R**: **Chamf./round.** (default: 0)
  - **R** > 0: Radius of rounding arc
  - **R** < 0: Chamfer width
- **P**: **Depth**

Eccentric polygon on lateral surface G317

G317 defines a polygon in a lateral-surface contour. Program this figure in conjunction with **G840**, **G845** or **G846**.



Parameters:

- **Z**: **Center**
- **CY**: **Center** as a linear dimension (reference: unrolled lateral surface, using the **Reference diameter**)
- **C**: **Center** (angle)
- **Q**: **Number edges**
- **A**: **Angle to Z axis** (default: 0°)
- **K**: **+edge lngth/-width a. flats**
  - **K** > 0: Edge length
  - **K** < 0: Width across (Inside diameter)
- **R**: **Chamf./round.** (default: 0)
  - **R** > 0: Radius of rounding arc
  - **R** < 0: Chamfer width
- **P**: **Depth**

6.19 Thread cycles

Overview of thread cycles

- **G31** machines single threads, successions of threads and multi-start threads defined with **G24**-Geo, **G34**-Geo, or **G37**-Geo (**FINISHED PART**). **G31** can also machine a threading contour defined directly after the cycle call and concluded by **G80**
  Further information: "Universal tapping cycle G31", Page 398
- **G32** cuts a simple thread in any desired direction and position
  Further information: "Single thread cycle G32", Page 403
- **G33** conducts a single thread cut. The direction of the single thread cut is as desired
  Further information: "Thread single path G33", Page 405
- **G35** cuts a simple cylindrical metric ISO thread without run-out
  Further information: "Metric ISO thread G35", Page 407
- **G352** cuts a tapered API thread
  Further information: "Tapered API thread G352", Page 408

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

Refer to your machine manual.

This function must be set up by your machine manufacturer.

Remember that position changes resulting from handwheel superimposition are no longer effective after the cycle end or the **Last cut** function!

Parameter V: Type of infeed

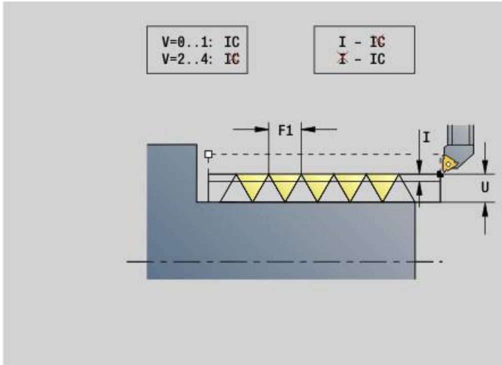
With the **V** parameter you define the type of infeed for thread cutting cycles.

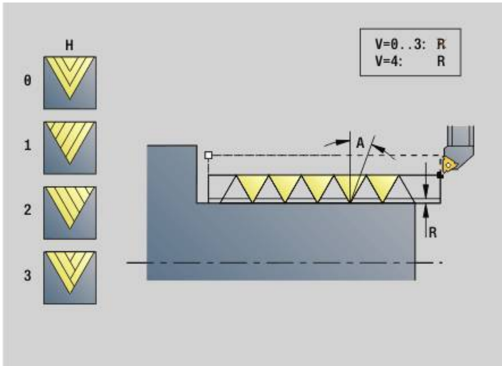
The following infeed types are available:

- **0: Const. mach. X-section** – the control reduces the cutting depth with each infeed so that the chip cross section, and therefore the removal rate, remain constant
- **1: Const. infeed** – with each infeed the control uses the same cutting depth without exceeding the **Max. approach I**
- **2: EPL with distrib. of cuts** – the control calculates the cutting depth for constant infeed from the **Thread pitch F1** and **Shaft speed constant S**. If the multiple of the cutting depth does not correspond to the **Thread depth**, the control uses the **Rem. cut depth** for the first infeed. The control proportions the remaining cuts, i.e. it divides the last cutting depth into four partial cuts. The first cut is half the calculated cutting depth, the second is a quarter, and the third and fourth each are an eighth
- **3: EPL w/o distrib. of cuts** – the control calculates the cutting depth for constant infeed from the **Thread pitch F1** and constant shaft speed **S**. If the multiple of the cutting depth does not correspond to the **Thread depth**, the control uses the **Rem. cut depth** for the first infeed. All subsequent infeeds are constant and correspond to the calculated cutting depth
- **4: MANUALplus 4110** – the control executes the first infeed with the **Max. approach I**. To determine the subsequent cutting depths, the control uses the formula **gt = 2 * I * SQRT** current no. of cuts, where **gt** is the absolute depth. Since the cutting depth is reduced with each infeed because the current number of cuts is increased accordingly by the value **1**, and if the cutting depth falls below the value given in **Rem. cut depth R**, the control uses the value defined here as the new constant cutting depth! If the multiple of the cutting depth does not correspond to the **Thread depth**, the control executes the last cut at the final depth
- **5: Constant infeed (4290)** – with each infeed the control uses the same cutting depth where the cutting depth corresponds to the **Max. approach I**. If the multiple of the cutting depth does not correspond to the **Thread depth**, the control uses the remaining cutting depth for the first infeed.
- **6: Const. w/ distrib. (4290)** – with each infeed, the control uses the same cutting depth where the cutting depth corresponds to the **Max. approach I**. If the multiple of the cutting depth does not correspond to the **Thread depth**, the control uses the **Rem. cut depth** for the first infeed. The control proportions the remaining cuts, i.e. it divides the last cutting depth into four partial cuts. The first cut is half the calculated cutting depth, the second is a quarter, and the third and fourth each are an eighth

Universal tapping cycle G31

G31 machines single threads, successions of threads and multi-start threads defined with **G24**, **G34** or **G37**-Geo. **G31** can also machine a threading contour defined directly after the cycle call and concluded by **G80**.





Parameters:

- **ID**: **Auxiliary contour** – ID number of the contour to be machined
- **NS**: **Contour start block no.** – reference to basic element **G1**-Geo (for successions of threads: block number of the first basic element)
- **NE**: **Contour end block no.** – reference to basic element **G1**-Geo (for successions of threads: block number of the last basic element)
- **O**: **Char.start/end** – machine the form element
  - **0: No machining**
  - **1: At beginning**
  - **2: At end**
  - **3: At beginning and end**
  - **4: Only chamfer/rounding** (requirement: the contour section consists of a single element)
- **J**: **Thread orientation** – reference orientation
  - **From 1st contour element**
  - **0: Longitudinal**
  - **1: Plane**
- **I**: **Max. approach**

  No input and **V** = 0 (constant chip cross section): **I** = 1/3 * **F**
- **IC**: **Number of cuts** – The infeed is calculated from **IC** and **U**

  Usable with:
  - **V** = 0: Constant chip cross section
  - **V** = 1: Constant infeed
- **B**: **Run-in lgth**

  (No input: The run-in length is determined from the contour)
  If this is not possible, the value is calculated from the kinematic parameters. The thread contour is extended by the value **B**.
- **P**: **Overflow length**

  No input: The run-out length is determined from the contour. If this is not possible, the value is calculated. The thread contour is extended by the value **P**.
- **A**: **Approach ang.** (range: –60° < **A** < 60°; default: 30°)

- **V**: **Type of infeed**
  - **0: Const. mach. X-section**
  - **1: Const. infeed**
  - **2: EPL with distrib. of cuts**
  - **3: EPL w/o distrib. of cuts**
  - **4: MANUALplus 4110**
  - **5: Constant infeed (4290)**
  - **6: Const. w/ distrib. (4290)**
- **H**: **Type of offset** for smoothing the thread flanks (default: 0)
  - **0: Without offset**
  - **1: From left**
  - **2: From right**
  - **3: Alternating left/right**
- **R**: **Remaining cut depth (V=4)**
- **C**: **Start angle**
- **BD**: **Outside=0 / Inside=1** – external/internal thread (no meaning for closed contours)
  - 0: External thread
  - 1: Internal thread
- **F**: **Thread pitch**
- **U**: **Thread depth**
- **K**: **Thread runout length**
  - **K** > 0 Run-out
  - **K** < 0 Run-in
- **D**: **No.gears**
- **Q**: **Number no-load.**
- **E**: **Variable gr.** (default: 0)
  Increases/decreases the pitch per revolution by **E**.

> If a thread has been defined with **G24**, **G34** or **G37**-Geo, the parameters **F**, **U**, **K** and **D** are not relevant.

Slop.length B: The slide requires a run-in distance at the thread starting point in order to accelerate to the programmed contouring feed rate before starting the actual thread.

Overflow length P: The slide needs overtravel at the end of the thread to decelerate. Remember that the paraxial line **P** needs overtravel, even with an oblique thread run-out.

You can calculate the minimum **Slop.length** and **Overflow length** using the following equation:

- **Slop.length**: **B** = 0.75 * (**F** * **S**)² / **a** * 0.66 + 0.15
- **Overflow length**: **P** = 0.75 * (**F** * **S**)² / **a** * 0.66 + 0.15
  - **F**: **Thread pitch** in mm/revolution
  - **S**: **Spindle speed** in revolutions/second
  - **a**: **Acceleration** in mm/s² (see axis data)

Determination of external/internal thread:

- **G31** with contour reference—closed contour: External or internal thread is defined by the contour. **BD** has no meaning
- **G31** with contour reference—open contour: External or internal thread is defined by **BD**. If **BD** is not programmed, the contour is used to determine whether a thread is external or internal.
- If the thread contour is programmed directly after the cycle, **BD** determines whether the thread is an internal or external thread. If **BD** is not programmed, the algebraic sign of **U** is evaluated (as in the MANUALplus 4110).
  - **U** > 0: Internal thread
  - **U** < 0: External thread

Start angle C: At the end of the **Slop.length B**, the spindle is at the **Start angle C** position. Therefore, if the thread is to start exactly at the **Start angle**, position the tool by the **Slop.length** (run-in length)—or by the **Slop.length** plus a multiple of the thread pitch—in front of the beginning of the thread.

The individual thread cuts are calculated from the **Thread depth**, **Max. approach I**, and **Type of infeed V**.

- **NC stop** – the control retracts the tool from the thread groove and then stops all tool movements
  Lift-off distance in the **threadLiftOff** machine parameter (no. 601804)
- Feed rate override is not effective

NOTICE

Danger of collision!

The control does not check for collisions between the **Overflow length P** and the workpiece contour (e.g., contour of the finished part). Danger of collision during machining!

- ▶ Test **Overflow length P** in **Simulation** submode using the graphic

Example: G31

. . .	
FINISHED PART	
N 2 G0 X16 Z0	
N 3 G52 P2 H1	
N 4 G95 F0.8	
N 5 G1 Z-18	
N 6 G25 H7 I1.15 K5.2 R0.8 W30 BF0 BP0	
N 7 G37 Q12 F2 P0.8 A30W30	
N 8 G1 X20 BR-1 BF0 BP0	
N 9 G1 Z-23.8759 BR0	
N 10 G52 G95	
N 11 G3 Z-41.6241 I-14.5 BR0	
N 12 G1 Z-45	

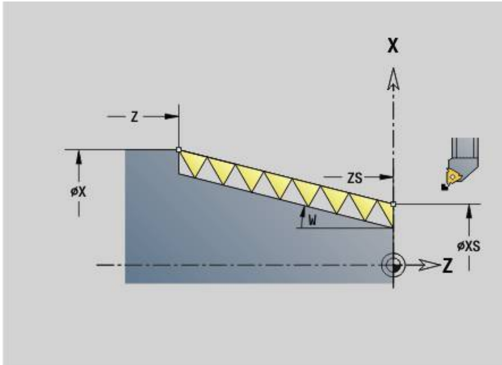
N 13 G1 X30 BR2	
N 14 G1 Z-50 BR0	
N 15 G2 X36 Z-71 I12 BR5	
N 16 G1 X40 Z-80	
N 17 G1 Z-99	
N 18 G1 Z-100	Thread
N 19 G1 X50	
N 20 G1 Z-120	
N 21 G1 X0	Thread
N 22 G1 Z0N 23 G1 X16 BR-1.5	
. . .	
AUXILIARY CONTOUR ID"Thread"	
N 24 G0 X20 Z0	
N 25 G1 Z-30	
N 26 G1 X30 Z-60	
N 27 G1 Z-100	
MACHINING	
N 32 G14 Q0 M108	
N 33 T9 G97 S1000 M3	
N 34 G47 P2	
N 35 G31 NS16 NE17 J0 IC5 B5 P0 V0 H1BD0 F2 K10	
N 36 G0 X110 Z20	
N 38 G47 M109	
	G80 contours can be inside or outside
N 43 G31 IC4 B4 P4 A30 V0 H2 C30 BD0 F6U3 K-10 Q2	
N 44 G0 X80 Z0	
N 45 G1 Z-20	
N 46 G1 X100 Z-40	
N 47 G1 Z-60	
N 48 G80	
	External thread regardless of the value defined in **BD**
N 49 G0 X50 Z-30	
N 50 G31 NS16 NE17 O0 IC2 B4 P0 A30 V0H1 C30 BD1 F2 U1 K10	
N 51 G0 Z10 X50	
	Auxiliary contours can be inside or outside if they are not closed
N 52 G0 X50 Z-30	
N 53 G31 ID"Thread" O0 IC2 B4 P0 A30 V0H1 C30 BD1 F2 U1 K10	
N 60 G0 Z10 X50	

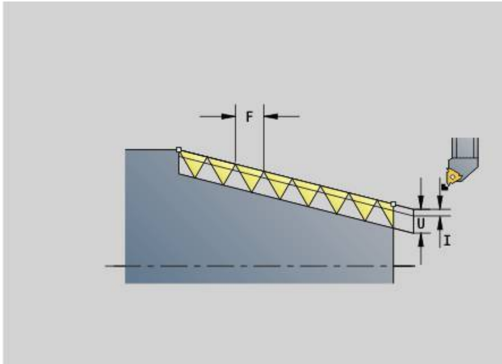
Cycle run:

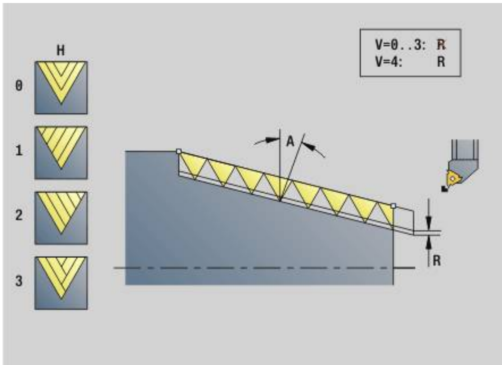
1 Calculates the number of cutting passes
2 Moves diagonally to the internal starting point at rapid traverse. This point lies in front of the thread starting point by the **Slop.length B**. If **H** = 1 (or 2, 3), the current offset is taken into account for calculating the internal starting point. The internal starting point is calculated on the basis of the tool tip
3 Accelerates to feed rate (line **B**)
4 Executes a thread cut
5 Decelerates (line **P**)
6 Retracts to safety clearance, returns at rapid traverse, and approaches for next pass. For multiple threads, the same rate of cut is used for each thread turn, before the next infeed motion is executed
7 Repeats 3 to 6 until the complete thread has been cut
8 Executes air cuts
9 Returns to starting point

Single thread cycle G32

G32 cuts a single thread in any desired direction and position (longitudinal, tapered or transverse thread; internal or external thread).







NOTICE

Danger of collision!

If you modify the angle of infeed or the thread depth, the control shifts the position of the thread in an axial direction. In this case, the tool no longer hits any existing thread grooves and the thread flanks are destroyed. Danger of collision during reworking!

- ▶ Be sure to compensate only the tool, not the thread parameters

Parameters:

- **X**: **Final point** (diameter value)
- **Z**: **Final point**
- **XS**: **Starting diameter**
- **ZS**: **Starting position Z**
- **BD**: **Outside=0 / Inside=1** – external/internal thread
  - 0: External thread
  - 1: Internal thread
- **F**: **Thread pitch**
- **U**: **Thread depth** (default: no input)
  - Outside thread: **U** = 0.6134 * **F1**
  - Inside thread: **U** = –0.5413 * **F1**
- **I**: **Max. approach**
- **IC**: **Number of cuts** – The infeed is calculated from **IC** and **U**
  Usable with:
  - **V** = 0: Constant chip cross section
  - **V** = 1: Constant infeed
- **V**: **Type of infeed**
  - **0: Const. mach. X-section**
  - **1: Const. infeed**
  - **2: EPL with distrib. of cuts**
  - **3: EPL w/o distrib. of cuts**
  - **4: MANUALplus 4110**
  - **5: Constant infeed (4290)**
  - **6: Const. w/ distrib. (4290)**
- **H**: **Type of offset** for smoothing the thread flanks (default: 0)
  - **0: Without offset**
  - **1: From left**
  - **2: From right**
  - **3: Alternating left/right**
- **WE**: **Lift off method with K=0** (default: 0)
  - **0: G0 at end**
  - **1: Lift-off in thread**

- **K**: **Thread runout length** at thread end point (default: 0)
- **W**: **Taper angle** (range: –45° < **W** < 45°)
  Position of the taper thread with respect to longitudinal or transverse axis:
  - **W** > 0: Rising contour (in machining direction)
  - **W** < 0: Falling contour
- **C**: **Start angle**
- **A**: **Approach ang.** (range: –60° < **A** < 60°; default: 30°)
- **R**: **Remaining cut depth** (default: 0)
  - **0**: The last cut is divided into four partial cuts: 1/2, 1/4, 1/8 and 1/8
  - **1**: Without distribution of remaining cuts
- **E**: **Variable gr.** (default: 0)
  Increases/decreases the pitch per revolution by **E**. (has currently no effect)
- **Q**: **Number no-load.**
- **D**: **No.gears**
- **J**: **Thread orientation** – reference orientation
  - **0: Longitudinal**
  - **1: Transverse**

The cycle calculates the thread from the **Final point** of the thread, the **Thread depth**, and the current tool position.

First infeed = Remainder of the division of thread depth/cutting depth.

Transverse thread: Use **G31** with contour definition for cutting transverse threads.

- **NC stop** – the control retracts the tool from the thread groove and then stops all tool movements
  Lift-off distance in the **threadLiftOff** machine parameter (no. 601804)
- Feed rate override is not effective

Example: G32

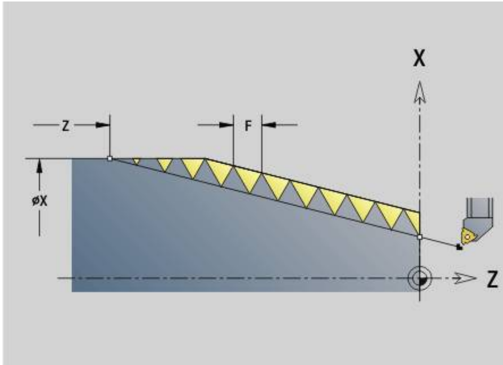
. . .	
N1 T4 G97 S800 M3	
N2 G0 X16 Z4	
N3 G32 X16 Z-29 F1.5	Thread
. . .	

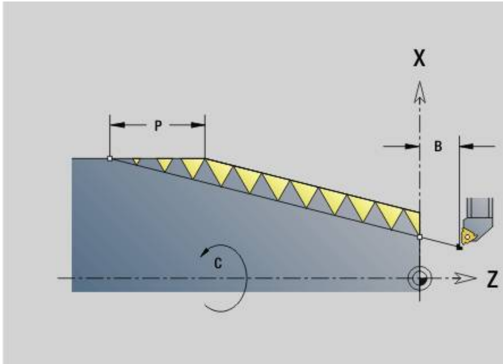
Cycle run:

1. Calculates the number of cutting passes
2. Executes a thread cut
3. Returns at rapid traverse and approaches for next pass
4. Repeats 2 to 3 until the complete thread has been cut
5. Executes air cuts
6. Returns to starting point

Thread single path G33

G33 conducts a single thread cut. The direction of the single thread path is as desired (longitudinal, tapered or transverse threads; internal or external threads). You can make successive threads by programming **G33** several times in succession.





Position the tool in front of the thread by the **Slop.length B** if the slide must accelerate to the feed rate. And remember the **Overflow length P** before the **Final point** of the thread if the slide needs to decelerate.

Parameters:

- **X**: **Final point** (diameter value)
- **Z**: **Final point**
- **F**: **Feed per rot.** (thread pitch)
- **B**: **Run-in lgth**
- **P**: **Overflow length**
- **C**: **Start angle**
- **H**: **Ref. direction** for the thread pitch (default: 0)
  - 0: Feed rate on the Z axis (for longitudinal and taper threads up to a max. angle of +45°/–45° to the Z axis)
  - 1: Feed rate on the X axis (for transverse and taper threads up to a max. angle of +45°/–45° to the X axis)
  - 3: Contouring feed rate
- **E**: **Variable gr.** (default: 0)
  Increases/decreases the pitch per revolution by **E**. (has currently no effect)
- **I**: **Retraction distance** X – retraction path for cycle stop in the thread (incremental value)
- **K**: **Retraction distance** Z – retraction path for cycle stop in the thread (incremental value)

Slop.length B: The slide requires a run-in length to the thread starting point in order to accelerate to the programmed contouring feed rate before starting the actual thread. Default: **cfgAxisProperties/SafetyDist**

Overflow length P: The slide needs an overtravel at the end of the thread to decelerate. Remember that the paraxial line **P** needs overtravel, even with an oblique thread run-out.

- **P** = 0: Start of a successive thread
- **P** > 0: End of a successive thread

Start angle C: At the end of the **Slop.length B**, the spindle is at the **Start angle C** position.

- **NC stop** – the control retracts the tool from the thread groove and then stops all tool movements
  Lift-off distance in the **threadLiftOff** machine parameter (no. 601804)
- Feed rate override is not effective
- Create thread with **G95** (feed rate per revolution)

Example: G33

. . .	
N1 T5 G97 S1100 G95 F0.5 M3	
N2 G0 X101.84 Z5	
N3 G33 X120 Z-80 F1.5 P0	Thread single path
N4 G33 X140 Z-122.5 F1.5	
N5 G0 X144	
. . .	

Cycle run:

1 Accelerates to feed rate (line **B**)
2 Moves at feed rate to the **Final point** of the thread – **Overflow length P**
3 Decelerates (line **P**) and remains at the **Final point** of the thread

Activating handwheel during G33

With the **G923** code, you can activate the handwheel in order to make compensations during a thread cut. In the **G923** code, you define limits within which traverse with the handwheel is possible.

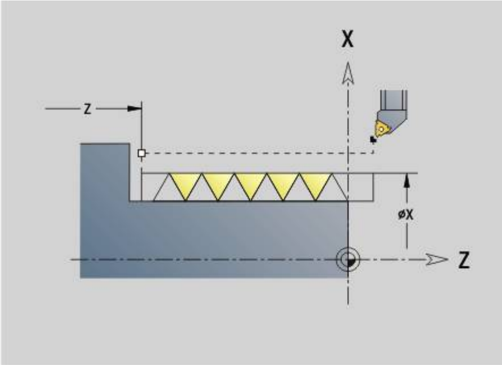
Parameters:

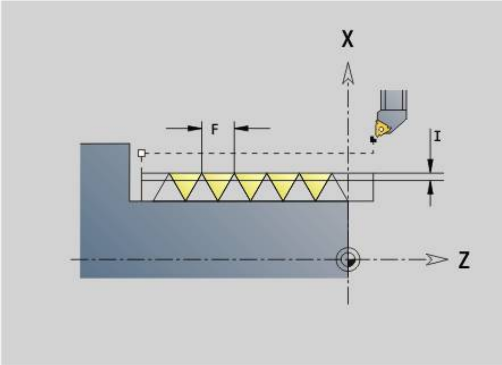
- **X**: **Max. positive offset** – limit in +X
- **Z**: **Max. positive offset** – limit in +Z
- **U**: **Max. negative offset** – limit in –X
- **W**: **Max. negative offset** – limit in –Z
- **H**: **Ref. direction**
  - **H** = 0: Longitudinal thread
  - **H** = 1: Transverse thread
- **Q**: **Sort of thread**
  - **Q** = 1: Right-hand thread
  - **Q** = 2: Left-hand thread

Metric ISO thread G35

G35 cuts a longitudinal thread (internal or external thread). The thread starts at the current tool position and ends at the **Final point X**, **Z**.

From the tool position relative to the **Final point** of the thread, the control automatically determines whether an internal or external thread is to be cut.





Parameters:

- **X**: **Final point** (diameter value)
- **Z**: **Final point**
- **F**: **Thread pitch**
- **I**: **Max. approach**

  No input: **I** is calculated from the thread pitch and the thread depth
- **Q**: **Number no-load.**
- **V**: **Type of infeed**
  - **0: Const. mach. X-section**
  - **1: Const. infeed**
  - **2: EPL with distrib. of cuts**
  - **3: EPL w/o distrib. of cuts**
  - **4: MANUALplus 4110**
  - **5: Constant infeed (4290)**
  - **6: Const. w/ distrib. (4290)**

> - **NC stop** – the control retracts the tool from the thread groove and then stops all tool movements
>   Lift-off distance in the **threadLiftOff** machine parameter (no. 601804)
> - When programming an internal thread, it is advisable to preset the **Thread pitch F** since the diameter of the longitudinal element is not the thread diameter.
>   If you have the control calculate the thread pitch automatically, slight deviations may occur.

Example: G35

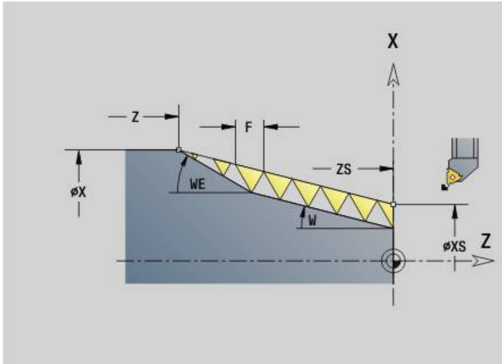
%35.nc	
N1 T5 G97 S1500 M3	
N2 G0 X16 Z4	
N3 G35 X16 Z-29 F1.5	
END	

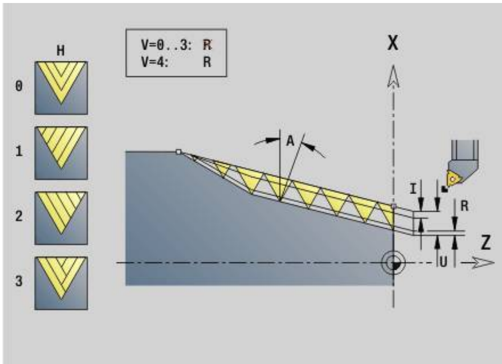
Cycle run:

1 Calculates the number of cutting passes
2 Executes a thread cut
3 Returns at rapid traverse and approaches for next pass
4 Repeats 2 to 3 until the complete thread has been cut
5 Executes air cuts
6 Returns to starting point

Tapered API thread G352

G352 cuts a tapered single or multi-start **API thread**. The **Thread depth** decreases at the thread run-out.





Parameters:

- X: **Final point** (diameter value)
- Z: **Final point**
- XS: **Starting diameter**
- ZS: **Starting position Z**
- F: **Thread pitch**
- U: **Thread depth**
  - **U** > 0: Internal thread
  - **U** <= 0: External thread (lateral surface or front face)
  - **U** = +999 or –999: Thread depth is calculated
- I: **Max. approach**
- V: **Type of infeed**
  - **0: Const. mach. X-section**
  - **1: Const. infeed**
  - **2: EPL with distrib. of cuts**
  - **3: EPL w/o distrib. of cuts**
  - **4: MANUALplus 4110**
  - **5: Constant infeed (4290)**
  - **6: Const. w/ distrib. (4290)**
- H: **Type of offset** for smoothing the thread flanks (default: 0)
  - **0: Without offset**
  - **1: From left**
  - **2: From right**
  - **3: Alternating left/right**
- A: **Approach ang.** (range: –60° < **A** < 60°; default: 30°)
  - **A** < 0: Infeed on left thread flank
  - **A** > 0: Infeed on right thread flank
- R: **Remaining cut depth (V=4)**
- W: **Taper angle** (range: –45° < **W** < 45°)
- WE: **Run-out angle** (range: 0° < **WE** < 90°)
- D: **No.gears**
- Q: **Number no-load.**
- C: **Start angle**

Internal or external threads: See algebraic sign of **U**

Number of cutting passes: The first cut is performed at the cutting depth defined for **I** and is reduced with each cut until the tool reaches the remaining cutting depth **R**.

Handwheel superposition (provided that your machine is equipped accordingly): The superposition is limited to the following range:

- X direction: Depending on the current cutting depth—without exceeding the starting and end points of the thread
- Z direction: Maximal 1 thread groove—without exceeding the starting and end points of the thread

Definition of taper angle:

- **XS/ZS**, **X/Z**
- **XS/ZS**, **Z**, **W**
- **ZS**, **X/Z**, **W**

> - **NC stop** – the control retracts the tool from the thread groove and then stops all tool movements
>   Lift-off distance in the **threadLiftOff** machine parameter (no. 601804)
> - When programming an internal thread, it is advisable to preset the **Thread pitch F** since the diameter of the longitudinal element is not the thread diameter.
>   If you have the control calculate the thread pitch automatically, slight deviations may occur.

Example: G352

%352.nc	
N1 T5 G97 S1500 M3	
N2 G0 X13 Z4	
N3 G352 X16 Z-28 XS13 ZS0 F1.5 U-999WE12	
END	

Cycle run:

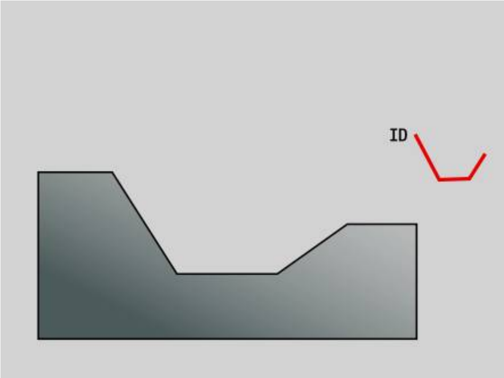
1. Calculates the number of cutting passes
2. Executes a thread cut
3. Returns at rapid traverse and approaches for next pass
4. Repeats 2 to 3 until the complete thread has been cut
5. Executes air cuts
6. Returns to starting point

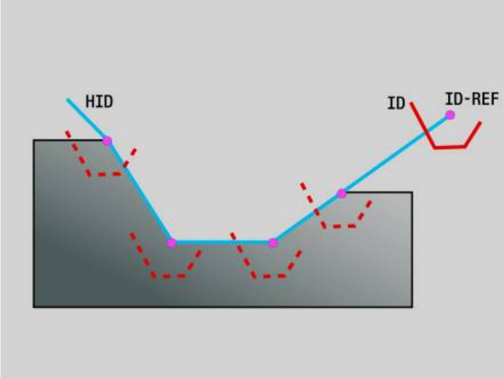
Contour thread G38

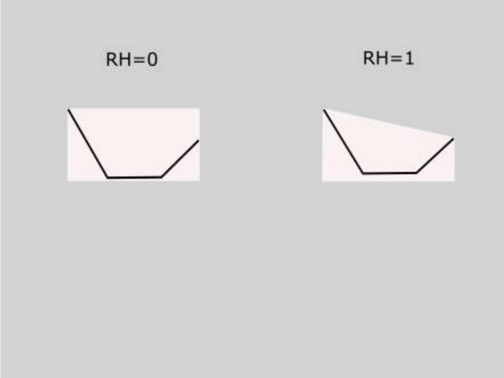
Cycle **G38** machines a thread whose form does not correspond to the tool form. Use a recessing or button tool for machining.

Describe the contour of the thread profile as an **Auxiliary contour** that you call in the **ID** parameter. The position of the **Auxiliary contour** must correspond to the start position of the thread cuts. You can select the entire **Auxiliary contour** or just segments in the cycle.

If you wish, you can describe the thread path in another **Auxiliary contour** and call it in the **HID** parameter. This **Auxiliary contour** may contain approach and departure paths, but no circular arcs or roundings.







Parameters:

- **ID**: **Thread profile** – ID number of the contour to be machined, defining the thread profile
- **NS**: **Starting block no. of contour** – beginning of contour section
- **NE**: **Contour end block no.** – end of contour section
- **HID**: **Thread path** – ID number of the contour to be machined, defining the thread path
- **RH**: **Type of workpiece blank**
  - **0**: The workpiece blank corresponds to a paraxial envelope around the **AUXILIARY CONTOUR** of the thread profile (**ID**).
  - **1**:The workpiece blank corresponds to the thread profile **AUXILIARY CONTOUR** that is closed on the shortest path (**ID**).
- **Q**: **Roughing/Finish** – process variants
  - **0: Roughing**: The contour is roughed out line by line at maximum infeed **I** and **K**. A programmed oversize (**G58** or **G57**) is taken into account.
  - **1: Finishing**: The turn of the thread is created in individual cuts along the contour. Define the distances between the individual thread cuts on the contour with **I** and **K.**
- **X**: **Final point** (diameter value)
- **Z**: **Final point**
- **H**: **Kind of liber.** – Sequence for approaching the departing position (**XE** and **ZE**) after every machining step
- **XE**: **Departure position X**
- **ZE**: **Departure position Z**
- **F**: **Thread pitch**
- **I**: **Max. approach**
  - If **Q** = 0: Plunging depth during roughing
  - If **Q** = 1: Distance between the finishing cuts on circular arcs
- **K**: **Max. approach**
  - If **Q** = 0: Offset width for roughing
  - If **Q** = 1: Distance between the finishing cuts on a straight line
- **J**: **Thread runout length**
- **C**: **Start angle**

- **O**: **Type of infeed** – For checking the proportioning of cuts in the simulation
  - **0: Rapid traverse**
  - **1: Feed rate**

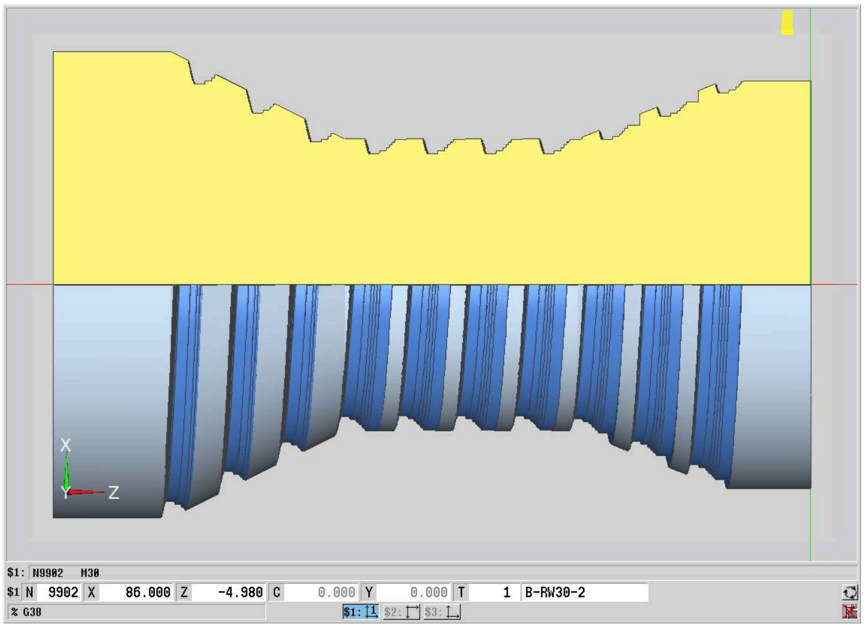
Notes:

- Program the contour of the thread profile (**ID**) at the position at which the tool is to start the thread grooves.
- Use the start point of the thread path (**HID**) to define the profile reference point (**ID-REF**). The profile reference point (**ID-REF**) serves as a leading point of the thread profile (**ID**) along the thread path (**HID**). During machining, the distance between the profile reference point (**ID-REF**) and the thread profile (**ID**) remains unchanged.
- If you shift the position of the thread profile (**ID**), you also have to shift the start point (**ID-REF**) of the thread path (**HID**). Otherwise, there will be an incorrect result.
- Depending on the desired result, the thread path (**HID**) may differ from the finished part contour.
- If no thread path is programmed as an **Auxiliary contour** (**HID**), the parameters **X**, **Z** and **J** define the thread path. As soon as you program the thread path as an **Auxiliary contour** (**HID**), the parameters **X**, **Z** and **J** have no effect anymore.
- By defining the value **1** in the parameter **RH: Type of workpiece blank**, you can avoid unnecessary air cuts on taper threads.

Example: G38

...	
FINISHED PART	
N 1 G0 X0 Z0	
N 2 G1 X70	
N 3 G1 Z-15	
N 4 G1 X50 Z-40	
N 5 G1 Z-80	
N 6 G1 X80 Z-110	
N 7 G1 Z-130	
...	
AUXILIARY CONTOUR ID"profile"	
N 9 G0 X80 Z0	
N 10 G1 X76 Z-2 BR0.4	
N 11 G1 Z-3 BR0.4	
N 12 G1 X75	
N 13 G1 Z-5 BR0.5	
N 14 G1 X83 Z-6	
...	
AUXILIARY CONTOUR ID"path"	
N 15 G0 X80 Z-3	
N 16 G1 X70 Z-15	
N 17 G1 X50 Z-40	

N 18 G1 Z-80	
N 19 G1 X80 Z-110	
N 20 G1 X90 Z-114	
...	
MACHINING	
N 21 G14	
$1 N 22 T1 ID"B-RW30-2" BW60 CW0	
N 23 G97 S500 M4 G95 F0.2	
N 24 G0 X120 Z10	
N 25 G38 ID"profile" HID"path" RH1 H1 XE120 ZE-12 F10 I0.5 K0.8 C0 O1	
N 26 G38 ID"profile" HID"path" RH1 Q1 H1 XE120 ZE-12 F10 I0.2 K0.1 C0 O1	
. . .	

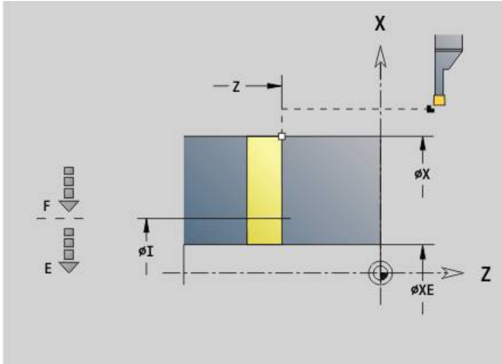


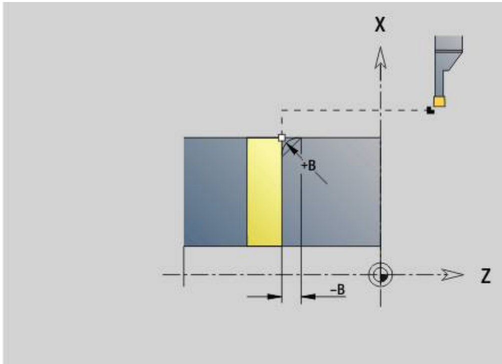
6.20 Parting cycle

Cut-off cycle G859

Cycle **G859** parts the workpiece. A **Chamf./round.** (optional) can be machined on the outside diameter. At the end of cycle, the tool retracts along the plane surface and returns to the starting point.

You can define a feed rate reduction, which becomes effective as soon as the position **I** is reached.





Parameters:

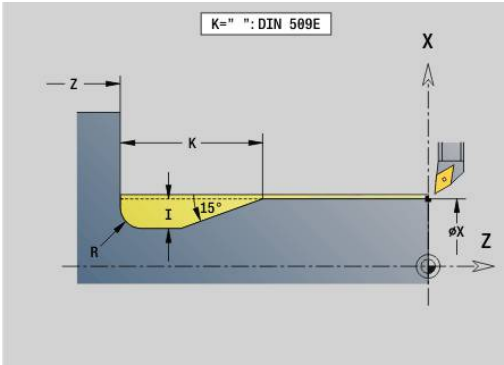
- **X**: **Cut-off diam.**
- **Z**: **Cut-off posit.**
- **XE**: **Inner diameter (pipe)**
- **B**: **-B cham./+B round.**
  - **B** > 0: Radius of rounding
  - **B** < 0: Width of chamfer
- **D**: **Speed limitation** – maximum speed during parting
- **I**: **Diam. feed reduction** – Limit diameter above which the workpiece will be machined with a reduced feed rate
  - **I** is defined: The control switches to feed rate after this position
  - **I** is not defined: No feed rate reduction
- **E**: **Reduced feed**
- **SD**: **Speed limit from I up**
- **U**: **Collector active diameter** (machine-dependent)
- **K**: **Retraction distance** after parting – Lift off the tool laterally from the plane surface before retraction

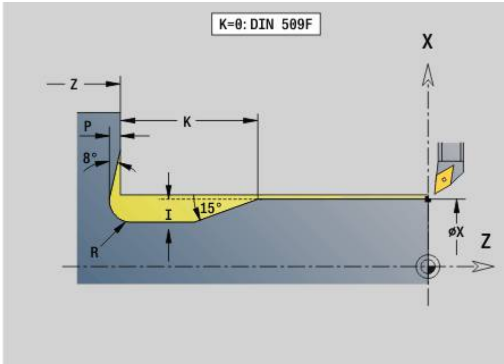
Example: G859

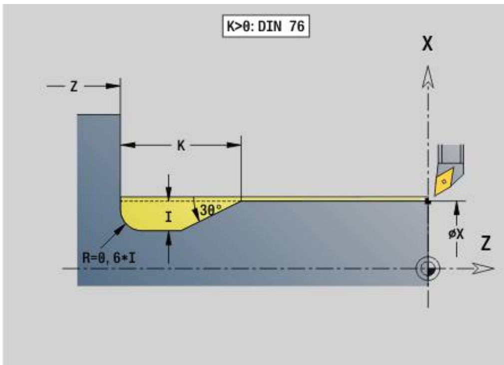
%859.nc	
N1 T3 G95 F0.23 G96 S248 M3	
N2 G0 X60 Z-28	
N3 G859 X50 Z-30 I10 XE8 E0.11 B1	
END	

6.21 Undercut cycles

Undercut cycle G85







G85 machines undercuts according to DIN 509 E, DIN 509 F, and DIN 76 (thread undercut).

Parameters:

- X: **Diameter**
- Z: **Target point**
- I: **Grind. o./Depth** (radius value)
  - DIN 509 E, F: Finishing oversize (default: 0)
  - DIN 76: Undercut depth
- K: **Undercut lgth.** and type of undercut
  - **K** no input: DIN 509 E
  - **K** = 0: DIN 509 F
  - **K** > 0: Undercut width for DIN 76
- E: **Reduc. Feed** for machining the undercut (default: active feed rate)

G85 machines the adjoining cylinder if you position the tool to **Destinat. point X** in front of the cylinder.

The undercut rounding arcs are executed with the radius 0.6 * **I**.

Parameters for Undercut DIN 509 E

Diameter	I	K	R
<= 18	0.25	2	0.6
> 18 – 80	0.35	2.5	0.6
> 80	0.45	4	1

Parameters for Undercut DIN 509 F

Diameter	I	K	R	P
<= 18	0.25	2	0.6	0.1
> 18 – 80	0.35	2.5	0.6	0.2
> 80	0.45	4	1	0.3

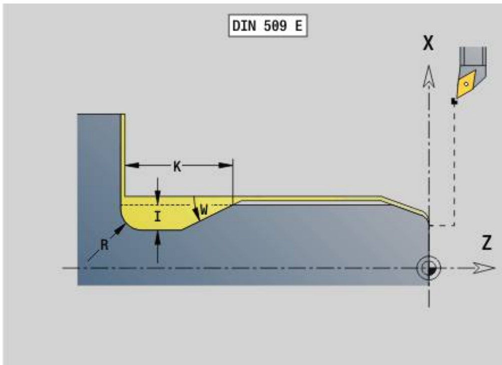
- **I** = **Undercut depth**
- **K** = **Undercut length**
- **R** = **Undercut radius**
- **P** = **Surf. depth**
- **Undercut angle** according to **Undercut DIN 509 E** and **Undercut DIN 509 F**: 15°
- **Face angle** according to **Undercut DIN 509 F**: 8°

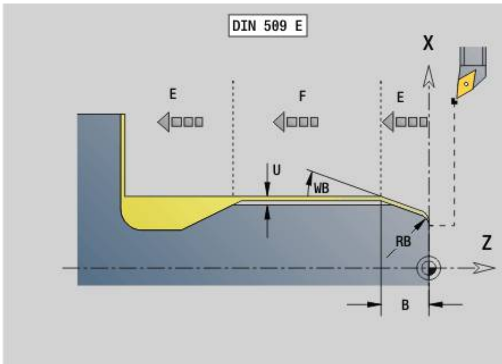
> - Tool radius compensation is inactive
> - Oversizes are not offset

Example: G85

. . .	
N1 T21 G95 F0.23 G96 S248 M3	
N2 G0 X62 Z2	
N3 G85 X60 Z-30 I0.3	
N4 G1 X80	
N5 G85 X80 Z-40 K0	
N6 G1 X100	
N7 G85 X100 Z-60 I1.2 K6 E0.11	
N8 G1 X110	
. . .	

Undercut DIN 509 E with cylinder machining G851





G851 machines the adjoining cylinder, the undercut, and the plane surface. It also machines a cylinder start chamfer if you enter at least one of the parameters **Cut-in length** or **Cut-in radius**.

Parameters:

- **I**: **Undercut depth** (default: value from standard table)
- **K**: **Undercut length** (default: standard table)
- **W**: **Undercut angle** (default: value from standard table)
- **R**: **Undercut radius** (default: standard table)
- **B**: **Cut-in length** (no input: no chamfer machined at start of cylinder)
- **RB**: **1st cut radius** (no input: 1st cut radius is not machined)
- **WB**: **Cut-in angle** (default: 45°)
- **E**: **Reduc. Feed** for machining the undercut (default: active feed rate)
- **H**: **Type of departure**
  - **0: To starting point**
  - **1: Plane surface end**
- **U**: **Grind. overs.** for the area of the cylinder (default: 0)

Parameters that are not programmed are automatically calculated by the control due to the cylinder diameter from the standard table.

Further information: "Undercut cycle G85", Page 414

Blocks following the cycle call

N.. G851 I.. K.. W..	Cycle call
N.. G0 X.. Z..	Corner point of cylinder start chamfer
N.. G1 Z..	Undercut corner
N.. G1 X..	End point on plane surface
N.. G80	End of contour definition

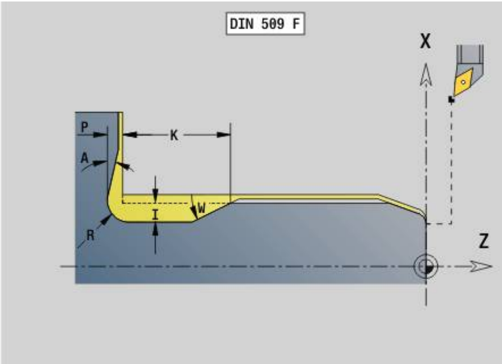
- Undercuts can only be executed in orthogonal, paraxial contour corners along the longitudinal axis
- Cutter radius compensation is active
- Oversizes are not offset

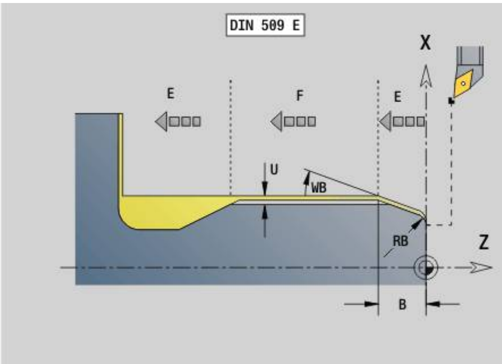
Example: G851

```
%851.nc
N1 T2 G95 F0.23 G96 S248 M3
N2 G0 X60 Z2
N3 G851 I3 K15 W30 R2 B5 RB2 WB30 E0.2 H1
N4 G0 X50 Z0
N5 G1 Z-30
N6 G1 X60
N7 G80
END
```

Undercut DIN 509 F with cylinder machining G852

G852 machines the adjoining cylinder, the undercut, and the plane surface. It also machines a cylinder start chamfer if you enter at least one of the parameters **Cut-in length** or **Cut-in radius**.





Parameters:

- **I**: **Undercut depth** (default: value from standard table)
- **K**: **Undercut length** (default: standard table)
- **W**: **Undercut angle** (default: value from standard table)
- **R**: **Undercut radius** (default: standard table)
- **P**: **Trav.dpth** (default: value from standard table)
- **A**: **Face angle** (default: standard table)
- **B**: **Cut-in length** (no input: no chamfer machined at start of cylinder)
- **RB**: **1st cut radius** (no input: 1st cut radius is not machined)
- **WB**: **Cut-in angle** (default: 45°)
- **E**: **Reduc. Feed** for machining the undercut (default: active feed rate)
- **H**: **Type of departure**
  - **0: To starting point**
  - **1: Plane surface end**
- **U**: **Grind. overs.** for the area of the cylinder (default: 0)

Parameters that are not programmed are automatically calculated by the control based on the cylinder diameter from the standard table.

Further information: "Undercut cycle G85", Page 414

Blocks following the cycle call

N.. G852 I.. K.. W..	Cycle call
N.. G0 X.. Z..	Corner point of cylinder start chamfer
N.. G1 Z..	Undercut corner
N.. G1 X..	End point on plane surface
N.. G80	End of contour definition

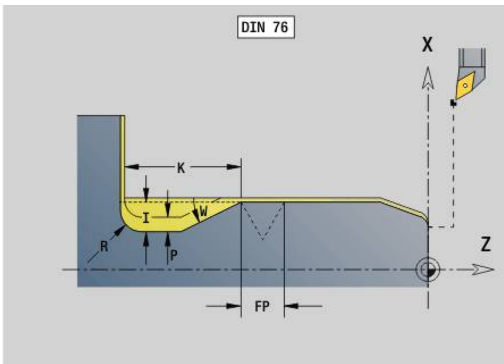
- Undercuts can only be executed in orthogonal, paraxial contour corners along the longitudinal axis
- Cutter radius compensation is active
- Oversizes are not offset

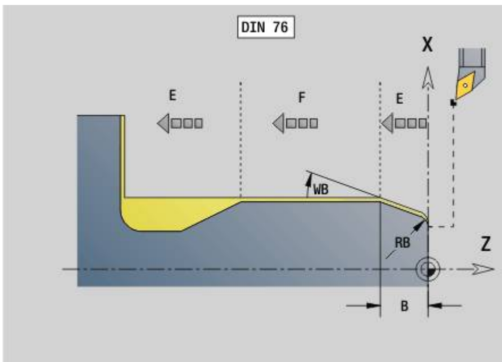
Example: G852

%852.nc	
N1 T2 G95 F0.23 G96 S248 M3	
N2 G0 X60 Z2	
N3 G852 I3 K15 W30 R2 P0.2 A8 B5 RB2 WB30E0.2 H1	
N4 G0 X50 Z0	
N5 G1 Z-30	
N6 G1 X60	
N7 G80	
END	

Undercut DIN 76 with cylinder machining G853

G853 machines the adjoining cylinder, the undercut, and the plane surface. It also machines a cylinder start chamfer if you enter at least one of the parameters **Cut-in length** or **Cut-in radius**.





Parameters:

- **FP**: **Thread pitch**
- **I**: **Undercut depth** (default: value from standard table)
- **K**: **Undercut length** (default: standard table)
- **W**: **Undercut angle** (default: value from standard table)
- **R**: **Undercut radius** (default: standard table)
- **P**: **Allowance**
  - **P** is not defined: The undercut is machined in one pass
  - **P** is defined: Division into pre-turning and finish-turning – P = longitudinal oversize; the transverse oversize is preset to 0.1 mm
- **B**: **Cut-in length** (no input: no chamfer machined at start of cylinder)
- **RB**: **1st cut radius** (no input: 1st cut radius is not machined)
- **WB**: **Cut-in angle** (default: 45°)
- **E**: **Reduc. Feed** for machining the undercut (default: active feed rate)
- **H**: **Type of departure**
  - **0: To starting point**
  - **1: Plane surface end**

Parameters that are not programmed are automatically calculated by the control from the standard table

- **FP** from the diameter
- **I**, **K**, **W** and **R** from the **FP** (**Thread pitch**)

Blocks following the cycle call

N.. G853 FP.. I.. K.. W..	Cycle call
N.. G0 X.. Z..	Corner point of cylinder start chamfer
N.. G1 Z..	Undercut corner
N.. G1 X..	End point on plane surface
N.. G80	End of contour definition

- Undercuts can only be executed in orthogonal, paraxial contour corners along the longitudinal axis
- Cutter radius compensation is active
- Oversizes are not offset

Example: G853

```
%853.nc
N1 T2 G95 F0.23 G96 S248 M3
N2 G0 X60 Z2
N3 G853 FP1.5 I47 K15 W30 R2 P1 B5 RB2WB30 E0.2 H1
N4 G0 X50 Z0
N5 G1 Z-30
N6 G1 X60
N7 G80
END
```

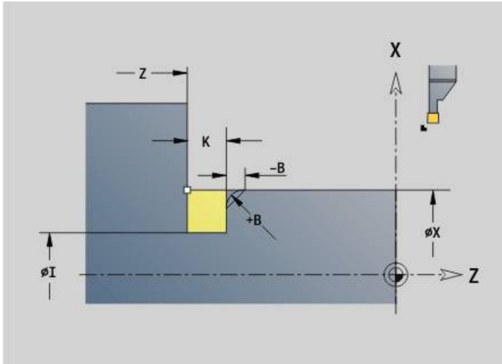
Undercut type U G856

G856 machines an undercut and finishes the adjoining plane surface. A **Chamf./round.** (optional) can be machined.

Tool position at the end of the cycle: Cycle starting point.

Parameters:

- **I**: **Undercut diameter** (default: standard table)
- **K**: **Undercut length** (default: standard table)
- **B**: **-B cham./+B round.**
  - **B** > 0: Radius of rounding
  - **B** < 0: Width of chamfer



Blocks following the cycle call

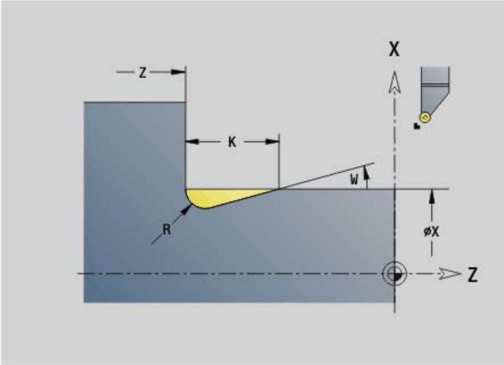
N.. G856 I.. K..	Cycle call
N.. G0 X.. Z..	Undercut corner
N.. G1 X..	End point on plane surface
N.. G80	End of contour definition

- Undercuts can only be executed in orthogonal, paraxial contour corners along the longitudinal axis
- Cutter radius compensation is active
- Oversizes are not offset
- If the cutting width of the tool is not defined, the control assumes that the tool's cutting width equals **K**.

Example: G856

%856.nc	
N1 T2 G95 F0.23 G96 S248 M3	
N2 G0 X60 Z2	
N3 G856 I47 K7 B1	
N4 G0 X50 Z-30	
N5 G1 X60	
N6 G80	
END	

Undercut type H G857



G857 machines an undercut. The end point is determined from the **Plunging angle** in accordance with **Undercut form H**.

Tool position at the end of the cycle: Cycle starting point

Parameters:

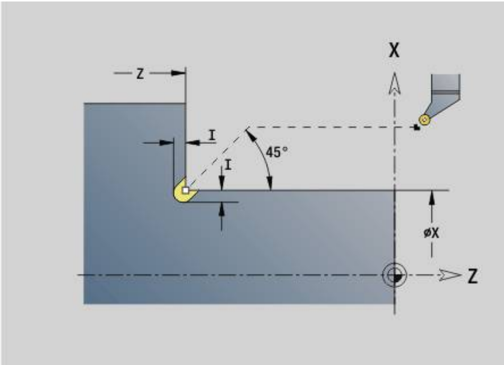
- **X**: **Edge** (diameter value)
- **Z**: **Edge**
- **K**: **Undercut lgth.**
- **R**: **Radius** (no input: no circular element; tool radius = undercut radius)
- **W**: **Plunging angle** (default: **W** is calculated)

> - Undercuts can only be executed in orthogonal, paraxial contour corners along the longitudinal axis
> - Cutter radius compensation is active
> - Oversizes are not offset

Example: G857

%857.nc	
N1 T2 G95 F0.23 G96 S248 M3	
N2 G0 X60 Z2	
N3 G857 X50 Z-30 K7 R2 W30	
END	

Undercut type K G858



G858 machines an undercut. This cycle performs only one linear cut at an angle of 45°. The resulting contour geometry therefore depends on the tool that is used.

Tool position at the end of the cycle: Cycle starting point

Parameters:

- **X**: **Edge** (diameter value)
- **Z**: **Edge**
- **I**: **Undercut depth**

> - Undercuts can only be executed in orthogonal, paraxial contour corners along the longitudinal axis
> - Cutter radius compensation is active
> - Oversizes are not offset

Example: G858

%858.nc	
N1 T9 G95 F0.23 G96 S248 M3	
N2 G0 X60 Z2	
N3 G858 X50 Z-30 I0.5	
END	

6.22 Drilling cycles

Overview of drilling and boring cycles and contour reference

The drilling and boring cycles can be used with driven or stationary tools.

Drilling and boring cycles:

- **G71 Simple drilling**
  Further information: "Simple drilling G71", Page 423
- **G72 Boring/cnt-sink** (only with contour reference (**ID**, **NS**)
  Further information: "Boring/cnt-sink G72", Page 425
- **G73 Tapping** (not with **G743** – **G746**)
  Further information: "Tapping G73", Page 426
- **G74 Deep-hole drilling**
  Further information: "Deep boring G74", Page 428
- **G36 Tapping** – single path (direct position input)
  Further information: "Tapping G36 – Single path", Page 422
- **G799 Thread milling** (direct position input)
  Further information: "Thread milling axial G799", Page 439

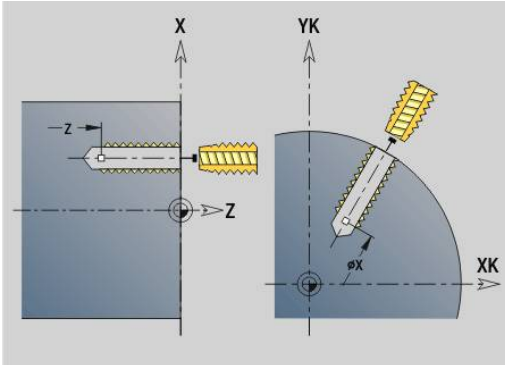
Pattern definitions:

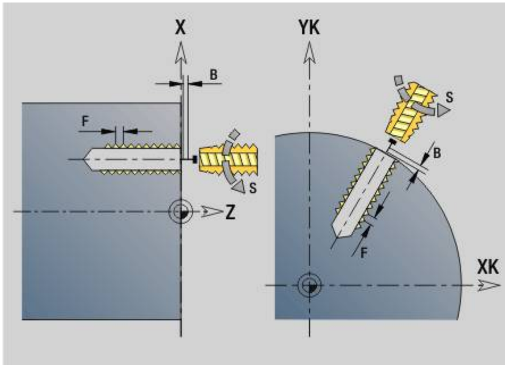
- **G743 Pattern linear face** for drilling and milling cycles
  Further information: "Linear pattern on front face G743", Page 433
- **G744 Pattern linear surf** for drilling and milling cycles
  Further information: "Linear pattern on lateral surface G744", Page 436
- **G745 Pattern circ. face** for drilling and milling cycles
  Further information: "Circular pattern on front face G745", Page 434
- **G746 Pattern circ. surf.** for drilling and milling cycles
  Further information: "Circular pattern on lateral surface G746", Page 437

Possibilities of defining a contour reference:

- Path definition directly in the cycle
- Reference to a hole or pattern definition in the contour section (**ID**, **NS**) for machining on the front face or lateral surface
- Centric hole in the turning contour (**G49**)
  Further information: "Bore hole (centric) G49–Geo", Page 303
- Pattern definition in the block before the cycle call (**G743** - **G746**)

Tapping G36 – Single path





G36 cuts axial and radial threads using driven or stationary tools. Depending on **X**/**Z**, **G36** decides whether a radial or axial hole will be machined.

Move to the starting point before **G36**. **G36** returns to the starting position after having cut the thread.

Parameters:

- **X**: **Diameter** – end point of radial hole
- **Z**: **Target point**
- **F**: **Feed per rot.** (thread pitch)
- **B**: **Run-in lgth** for spindle and feed drive synchronization
- **S**: **Return speed** (default: tapping speed)
- **P**: **Chip breaking depth**
- **I**: **Return distance**

Types of taps:

- Stationary tap: Main spindle and feed drive are synchronized
- Driven tap: Driven tool and feed drive are synchronized

> - **NC STOP** interrupts the tapping operation
> - **NC START** resumes the tapping operation
> - Use the feed rate override function for speed changes
> - Spindle override is not effective
> - Use a floating tap holder if the driven tool is not controlled, e.g. by a ROD encoder

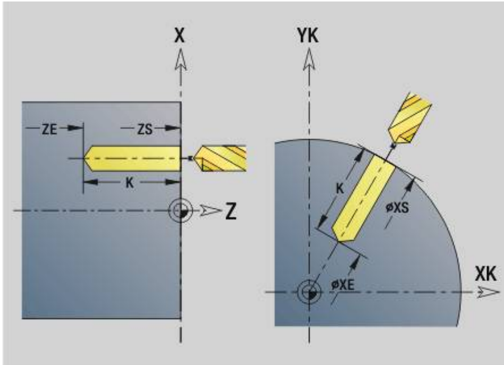
> If you interrupt program run during a tapping cycle, you can manually retract the tool from the hole in the Z axis. The control moves the spindle on a path matching the traverse.
>
> If the optional machine parameter **CfgBackTrack** (no. 122000) is active, use the **Start blck search** soft key to resume program run after the manual traverse.

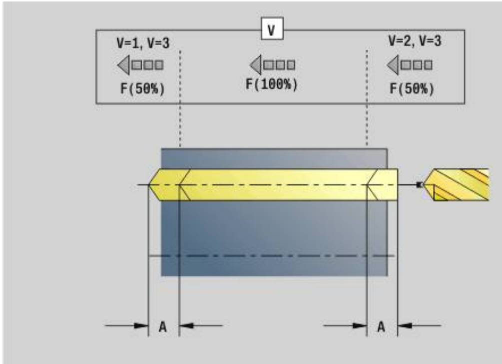
Example: G36

. . .	
N1 T5 G97 S1000 G95 F0.2 M3	
N2 G0 X0 Z5	
N3 G71 Z-30	
N4 G14 Q0	
N5 T6 G97 S600 M3	
N6 G0 X0 Z8	
N7 G36 Z-25 F1.5 B3	Tapping
. . .	

Simple drilling G71

G71 is used for axial and radial bore holes using driven or stationary tools.





Parameters:

- **ID**: **Hole dimensions** – name of the hole definition
- **NS**: **Starting block no. of contour** – beginning of contour section
  - Reference to the contour of the hole (**G49**, **G300** or **G310**-Geo)
  - No input: Single hole without contour description
- **NF**: **Position mark** – reference at which the cycle stores the predrill hole positions (range: 1 to 127)
- **XS**: **Start point** of radial hole (diameter value)
- **ZS**: **Start point** of axial hole
- **XE**: **Final point** of radial hole (diameter value)
- **ZE**: **Final point** of axial hole
- **K**: **Boring depth** (alternative to **XE** and **ZE**)
- **A**: **Tap/Drill. lth** (default: 0)
- **V**: **Bore variant** – feed rate reduction 50 % (default: 0)
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **RB**: **Return plane** (default: retract to starting position or to safety clearance; diameter value with radial holes and holes in the YZ plane)
- **E**: **Period of dwell** for chip breaking at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **DF**: **Retraction feed rate**
- **BS**: **Start elem.no.** – number of the first hole to be machined in a pattern
- **BE**: **End: elem.no.** – number of the last hole to be machined in a pattern
- **H**: **Brake off (1)** (default: 0)
  - 0: Spindle brake on
  - 1: Spindle brake off

> - Single hole without contour description: Program **XS** or **ZS** as alternative.
> - Hole with contour description: Do not program **XS**, **ZS**
> - Hole pattern: **NS** refers to the hole contour, and not the definition of the pattern

Example: G71

...	
N1 T5 G97 S1000 G95 F0.2 M3	
N2 G0 X0 Z5	
N3 G71 Z-25 A5 V2	Drilling
...	

Parameter combinations for single holes without contour description	
XS, **XE**	**ZS**, **ZE**
XS, **K**	**ZS**, **K**
XE, **K**	**ZE**, **K**

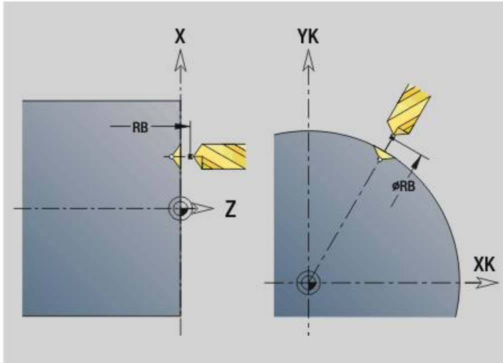
Feed rate reduction:

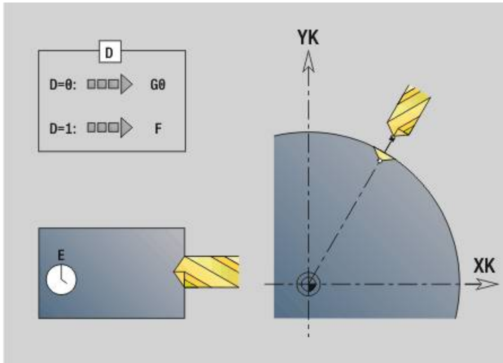
- Indexable insert drill and twist drill with 180° drilling angle
  - A feed rate reduction is only effective if the **Tap/Drill. lth A** parameter has been programmed
- Other drills
  - Beginning of hole: Feed rate reduction as programmed in **V**
  - End of hole: Reduction as of hole end point—length of first cut —safety clearance
- Length of first cut = tool tip
- Set-up clearance
  Further information: "Safety clearance", Page 349

Cycle run:

1 Approach behavior:
   - Hole without contour definition: Tool is located at the starting point (safety distance from the bore hole)
   - Hole with contour definition: Tool moves at rapid traverse to the starting point
     - **RB** not programmed: Moves up to the safety clearance
     - **RB** programmed: Moves to the position **RB** and then to the safety clearance
2 Spot drilling. Feed rate reduction depending on **V**
3 Drill hole at feed rate.
4 Through drilling. Feed rate reduction depending on **V**
5 Retraction at rapid traverse or feed rate, depending on **D**
6 Retraction position:
   - **RB** not programmed: Retraction to the starting point
   - **RB** programmed: Retraction to the position **RB**

Boring/cnt-sink G72





G72 is used for holes with contour definition (individual hole or hole pattern).

Use **G72** for the following axial and radial drilling functions using driven or stationary tools:

- Boring
- Countersinking
- Reaming
- NC drilling
- Centering

Parameters:

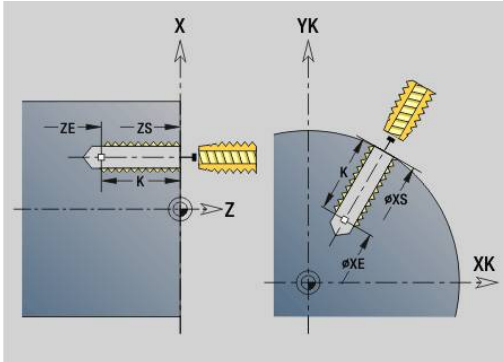
- **ID**: **Hole dimensions** – name of the hole definition
- **NS**: **Starting block no. of contour** – beginning of contour section
  - Reference to the contour of the hole (**G49**, **G300** or **G310**-Geo)
- **RB**: **Return plane** (default: retract to starting position or to safety clearance; diameter value with radial holes and holes in the YZ plane)
- **E**: **Period of dwell** for chip breaking at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **DF**: **Retraction feed rate**
- **BS**: **Start elem.no.** – number of the first hole to be machined in a pattern
- **BE**: **End: elem.no.** – number of the last hole to be machined in a pattern
- **H**: **Brake off (1)** (default: 0)
  - 0: Spindle brake on
  - 1: Spindle brake off

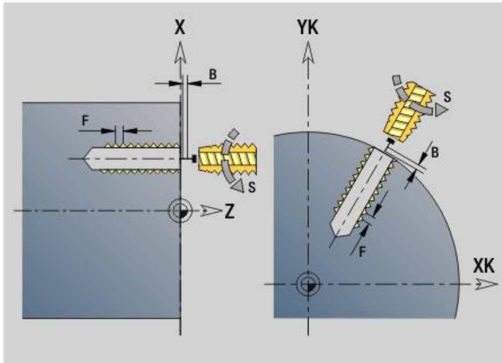
Cycle run:

1. Moves to the starting point at rapid traverse, depending on **RB**:
   - **RB** not programmed: Moves up to the safety clearance
   - **RB** programmed: Moves to the position **RB** and then to the safety clearance
2. Drills at reduced feed rate (50%)
3. Moves at feed rate to end of hole
4. Retraction at rapid traverse or feed rate, depending on **D**
5. Retraction position:
   - **RB** not programmed: Retraction to the starting point
   - **RB** programmed: Retraction to the position **RB**

> Hole pattern: **NS** refers to the hole contour, and not the definition of the pattern.

Tapping G73





G73 cuts axial and radial threads using driven or stationary tools.

Parameters:

- **ID**: **Hole dimensions** – name of the hole definition
- **NS**: **Starting block no. of contour** – beginning of contour section
  - Reference to the contour of the hole (**G49**, **G300** or **G310**-Geo)
  - No input: Single hole without contour description
- **XS**: **Start point** of radial hole (diameter value)
- **ZS**: **Start point** of axial hole
- **XE**: **Final point** of radial hole (diameter value)
- **ZE**: **Final point** of axial hole
- **K**: **Boring depth** (alternative to **XE** and **ZE**)
- **F**: **Thread pitch** (prevails over the contour description)
- **B**: **Run-in lgth**
- **S**: **Return speed** (default: tapping speed)
- **J**: **Retract length** when using floating tap holders (default: 0)
- **RB**: **Return plane** (default: back to start position)
- **P**: **Chip breaking depth**
- **I**: **Return distance**
- **BS**: **Start elem.no.** – number of the first hole to be machined in a pattern
- **BE**: **End: elem.no.** – number of the last hole to be machined in a pattern
- **H**: **Brake off (1)** (default: 0)
  - 0: Spindle brake on
  - 1: Spindle brake off

The starting position is calculated from the safety clearance and the **Slop.length B**.

Parameter combinations for single holes without contour description	
XS, **XE**	**ZS**, **ZE**
XS, **K**	**ZS**, **K**
XE, **K**	**ZE**, **K**

Retract length J: Use this parameter for floating tap holders. The cycle calculates a new nominal pitch on the basis of the thread depth, the programmed **Thread pitch**, and the **Retract length**. The nominal pitch is somewhat smaller than the **Thread pitch** of the tap. During tapping, the tap is pulled away from the chuck by the **Retract length**. With this method, you can achieve higher service life of the taps.

- Hole pattern: **NS** refers to the hole contour, and not the definition of the pattern
- Single hole without contour description: Program **XS** or **ZS** as alternative
- Hole with contour description: Do not program **XS**, **ZS**.
- The **NC STOP** key interrupts the tapping operation
- The **NC START** key resumes the tapping operation
- Use the feed rate override function for speed changes
- Spindle override is not effective
- Use a floating tap holder if the driven tool is not controlled, e.g. by a ROD encoder

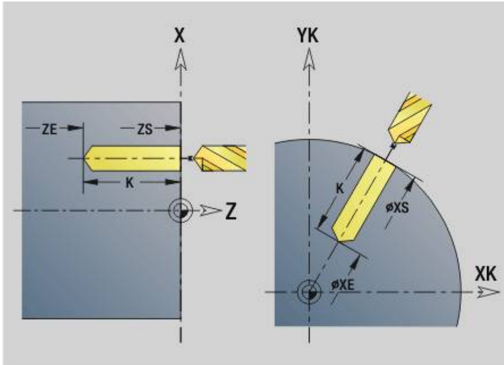
Cycle run:

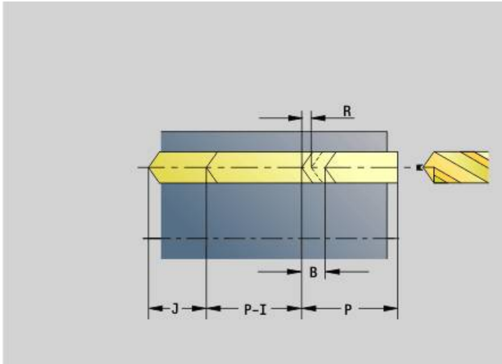
1 Moves at rapid traverse to the starting point:
   - **RB** not programmed: Moves up to the safety clearance
   - **RB** programmed: Moves to the position **RB** and then to the safety clearance
2 Moves along **Slop.length B** at feed rate (synchronization of spindle and feed drives)
3 Cuts the thread
4 Retraction position:
   - **RB** not programmed: Retraction to the starting point
   - **RB** programmed: Retraction to the position **RB**

If you interrupt program run during a tapping cycle, you can manually retract the tool from the hole in the Z axis. The control moves the spindle on a path matching the traverse.

If the optional machine parameter **CfgBackTrack** (no. 122000) is active, use the **Start blck search** soft key to resume program run after the manual traverse.

Deep boring G74





G74 is used for axial and radial holes in several stages using driven or stationary tools.

Parameters:

- **ID**: **Hole dimensions** – name of the hole definition
- **NS**: **Starting block no. of contour** – beginning of contour section
  - Reference to the contour of the hole (**G49**, **G300** or **G310**-Geo)
  - No input: Single hole without contour description
- **XS**: **Start point** of radial hole (diameter value)
- **ZS**: **Start point** of axial hole
- **XE**: **Final point** of radial hole (diameter value)
- **ZE**: **Final point** of axial hole
- **K**: **Boring depth** (alternative to **XE** and **ZE**)
- **P**: **1. Boring depth**
- **I**: **Reducing value** (default: 0)
- **B**: **Return distance** (default: to starting point of hole)
- **J**: **Minimum hole depth** (default: 1/10 of **P**)
- **R**: inside **Safety clearance**
- **A**: **Tap/Drill. lth** (default: 0)
- **V**: **Bore variant** – feed rate reduction 50 % (default: 0)
  - **0: Without reduction**
  - **1: At end of the hole**
  - **2: At start of the hole**
  - **3: At start & end of hole**
- **RB**: **Return plane** (default: retract to starting position or to safety clearance; diameter value with radial holes and holes in the YZ plane)
- **E**: **Period of dwell** for chip breaking at end of hole (default: 0)
- **D**: **Retraction type**
  - **0: Rapid traverse**
  - **1: Feed rate**
- **DF**: **Retraction feed rate**
- **BS**: **Start elem.no.** – number of the first hole to be machined in a pattern
- **BE**: **End: elem.no.** – number of the last hole to be machined in a pattern
- **H**: **Brake off (1)** (default: 0)
  - 0: Spindle brake on
  - 1: Spindle brake off

Example: G74

. . .	
N1 M5	
N2 T4 G197 S1000 G195 F0.2 M103	
N3 M14	
N4 G110 C0	
N5 G0 X80 Z2	
N6 G745 XK0 YK0 Z2 K80 Wi90 Q4 V2	
N7 G74 ZS-40 R2 P12 I2 B0 J8	Drilling
N8 M15	
. . .	

Parameter combinations for single holes without contour description	
XS, **XE**	**ZS**, **ZE**
XS, **K**	**ZS**, **K**
XE, **K**	**ZE**, **K**

The cycle is used for:

- Single hole without contour description
- Holes with contour definition (individual hole or hole pattern)

1. Boring depth P is used for the first pass. For each further drilling pass, the depth will be reduced by the **Reducing value I**, but the depth will not fall below the **Min.drill.depth J**. After each pass, the drill is retracted either by the **Return distance B** or to the starting point of the hole. If the inside **Safety clearance R** is defined, the tool is positioned to this distance at rapid traverse inside the hole.

Feed rate reduction:

- Indexable insert drill and twist drill with 180° drilling angle
  - A feed rate reduction is only effective if the **Tap/Drill. lth A** parameter has been programmed
- Other drills
  - Beginning of hole: Feed rate reduction as programmed in **V**
  - End of hole: Reduction as of hole end point—length of first cut —safety clearance
- Length of first cut = tool tip
- Set-up clearance
  Further information: "Safety clearance", Page 349

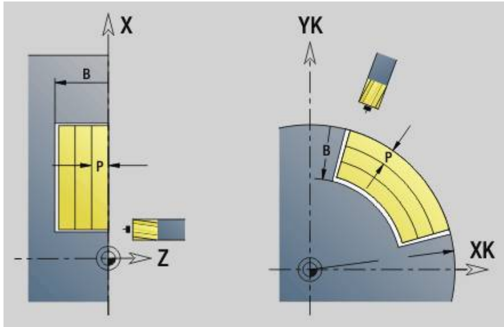
- Single hole without contour description: Program **XS** or **ZS** as alternative
- Hole with contour description: Do not program **XS**, **ZS**
- Hole pattern: **NS** refers to the hole contour, and not the definition of the pattern
- A feed rate reduction at end goes into effect only at the last drilling step

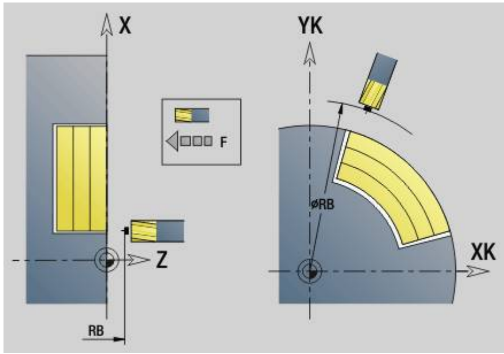
Cycle run:

1. Approach behavior:
   - Hole without contour definition: Tool is located at the starting point (safety distance from the bore hole)
   - Hole with contour definition: Tool moves at rapid traverse to the starting point
     - **RB** not programmed: Moves up to the safety clearance
     - **RB** programmed: Moves to the position **RB** and then to the safety clearance
2. Spot drilling. Feed rate reduction depending on **V**
3. Drill hole at feed rate.
4. Through drilling. Feed rate reduction depending on **V**
5. Retraction at rapid traverse or feed rate, depending on **D**
6. Retraction position:
   - **RB** not programmed: Retraction to the starting point
   - **RB** programmed: Retraction to the position **RB**

Bore milling G75

G75 is used for machining or deburring axial and radial bore holes or hole patterns with a milling cutter. The milling cutter can also be used to machine counterbores or enlarge holes.





Parameters:

- **ID**: **Hole dimensions** – name of the hole definition
- **NS**: **Starting block no. of contour** – beginning of contour section
  - Reference to the contour of the hole (**G49**-Geo, **G300**-Geo,**G310**-Geo, **G71** or **G73**)
  - No input: single hole without contour description
- **O**: **Machining operation:**
  - 0: Roughing
  - 1: Finishing
  - 2: Roughing and finishing
  - 3: Deburring
- **B**: **Milling depth** (default: depth from the contour description)
- **P**: **Max. approach** (default: milling in one infeed)
- **U**: **Overlap factor** – overlap of milling paths = **U** * milling diameter (default: 0.5)
- **H**: **Direction**
  - 0: Up-cut
  - 1: Climb
- **I**: **O-size X**
- **K**: **O-size Z**
- **F**: **Approach feed** for plunging (default: active feed rate)
- **RB**: **Return plane** (default: retract to starting position or to safety clearance; diameter value with radial holes and holes in the YZ plane)
- **W**: **Plunging angle** in infeed direction
- **WB**: **Diameter of the helix**

Programming notes:

- Only the contour description (ICP) for the C axis or Y axis is used for bore milling.
- **NS** refers to the hole contour, and not the definition of the pattern.
- If this cycle is used with the C axis, it machines funnel-shaped oval contours on the lateral surface, and not circles. Circles are machined when the Y axis is used. **Further information:** "Units G75 Bore milling Y", Page 239
- An active mirror function does not influence the type of milling defined in the cycle.
- Note that if the infeed distance is too large, the tool or the workpiece may be damaged.

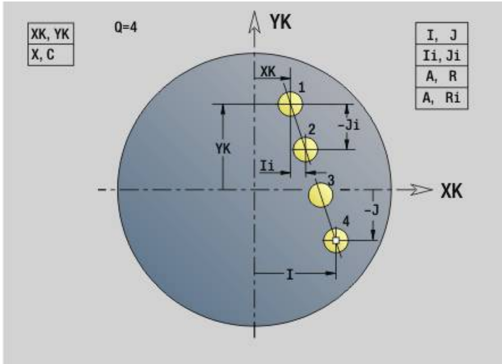
Example: G75

. . .	
N7 G300 XK30 YK25 B16 P30 W180	
. . .	
N8 M14	
N9 T3	
N10 G197 S1250 G195 F0.2 M103	
N11 M108	
N12 G110 C0	
N13 G0 X50 Z5	
N14 G147 K2	
N15 G75 NS7 P10 H1 W15	Bore milling
N16 G47 M109	
N17 G14 Q0	
. . .	

Cycle run:

1. Cutter moves at rapid traverse to the starting point
   - **RB** not programmed: moves up to the safety clearance
   - **RB** programmed: moves to the position **RB** and then to the safety clearance
2. The tool mills in a helix from the current position to the first plunging depth at the programmed feed rate
3. When the drilling depth has been reached, the tool moves along helical paths to the outside until it reaches the programmed hole diameter
4. Finally, the tool moves by a full circle to remove the material left.
5. Repeats 2 to 3 if the maximum infeed **P** does not correspond to the drilling depth
6. Retraction position:
   - **RB** not programmed: retraction to the starting point
   - **RB** programmed: retraction to the position **RB**

Linear pattern on front face G743



Cycle **G743** is used to machine linear drilling or milling patterns in which the individual features are arranged at a regular spacing on the face.

If you have not specified the **Final point ZE**, the drilling/milling cycle of the next NC block is used as a reference

Using this principle, you can combine pattern definitions with

- Drilling cycles (**G71**, **G74**, **G36**)
- The milling cycle for a linear slot (**G791**)
- The contour milling cycle with free contour (**G793**)

Parameters:

- **XK**: **Start point** (in Cartesian coordinates)
- **YK**: **Start point** (in Cartesian coordinates)
- **ZS**: **Start point** of drilling/milling operation
- **ZE**: **Final point** of drilling/milling operation
- **X**: **Start point** (in polar coordinates)
- **C**: **Start angle** (angle, in polar coordinates)
- **A**: **Pattern ang.** (reference: XK axis)
- **I**: **Final point** of pattern (in Cartesian coordinates)
- **Ii**: **Final point** – pattern distance (in Cartesian coordinates)
- **J**: **Final point** of pattern (in Cartesian coordinates)
- **Ji**: **Final point** – pattern distance (in Cartesian coordinates)
- **R**: **Distance to first/last hole**
- **Ri**: **Length** – **Incremental distance**
- **Q**: **Number of holes**

Parameter combinations for defining the starting point and the pattern positions:

- Starting point of pattern:
  - **XK**, **YK**
  - **X**, **C**
- Pattern positions:
  - **I**, **J** and **Q**
  - **Ii**, **Ji** and **Q**
  - **R**, **A** and **Q**
  - **Ri**, **Ai** and **Q**

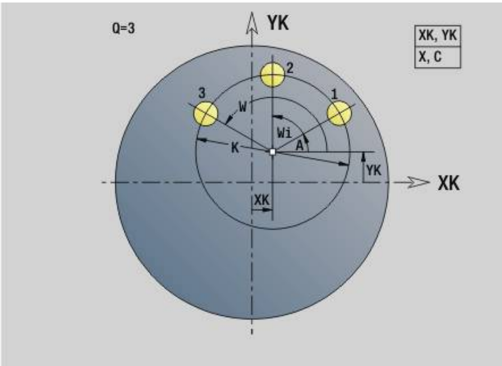
Example: G743

%743.nc	
N1 T7 G197 S1200 G195 F0.2 M104	
N2 M14	
N3 G110 C0	
N4 G0 X100 Z2	
N5 G743 XK20 YK5 A45 Ri30 Q2	
N6 G791 X50 C0 ZS0 ZE-5 P2 F0.15	
N7 M15	
END	

Example: Sequence of commands

	Simple drilling pattern
N.. G743 XK.. YK.. ZS.. ZE.. I.. J.. Q..	
. . .	
	Drilling pattern with deep-hole drilling
N.. G743 XK.. YK.. ZS.. I.. J.. Q..	
N.. G74 ZE.. P.. I..	
. . .	
	Milling pattern with linear slot
N.. G743 XK.. YK.. ZS.. I.. J.. Q..	
N.. G791 K.. A.. Z..	
. . .	

Circular pattern on front face G745



Cycle **G745** is used to machine drilling or milling patterns in which the individual features are arranged at a regular spacing in a circle or circular arc on the face.

If you have not specified the **Final point ZE**, the drilling/milling cycle of the next NC block is used as a reference

Using this principle, you can combine pattern definitions with:

- Drilling cycles (**G71**, **G74**, **G36**)
- The milling cycle for a linear slot (**G791**)
- The contour milling cycle with free contour (**G793**)

Parameters:

- **XK**: **Center** (in Cartesian coordinates)
- **YK**: **Center** (in Cartesian coordinates)
- **ZS**: **Start point** of drilling/milling operation
- **ZE**: **Final point** of drilling/milling operation
- **X**: **Diameter** – **Center** (in polar coordinates)
- **C**: **Angle** – **Center** (in polar coordinates)
- **K**: **Diameter** – pattern diameter
- **A**: **Start angle** – position of the first figure (reference: positive X axis; default: 0°)
- **W**: **Final angle** – position of the last figure (reference: positive X axis; default: 360°)
- **Wi**: **End angle** – **Angle increment**
- **Q**: **Number of holes**

- **V**: **Rotation dir.** (default: 0)
  - **V** = 0, without **W**: Figures are arranged on a full circle
  - **V** = 0, with **W**: Figures are arranged on the longer circular arc
  - **V** = 0, with **Wi**: The algebraic sign of **Wi** defines the direction (**Wi** < 0: clockwise)
  - **V** = 1, with **W**: Clockwise
  - **V** = 1, with **Wi**: Clockwise (algebraic sign of **Wi** has no effect)
  - **V** = 2, with **W**: Counterclockwise
  - **V** = 2, with **Wi**: Counterclockwise (algebraic sign of **Wi** has no effect)

Parameter combinations for defining the center of the pattern and the pattern positions:

- Center point of pattern:
  - **XK**, **YK**
  - **X**, **C**
- Pattern positions:
  - **A**, **W** and **Q**
  - **A**, **Wi** and **Q**

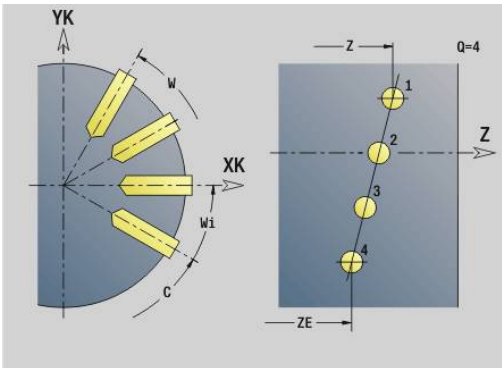
Example: G745

%745.nc	
N1 T7 G197 S1200 G195 F0.2 M104	
N2 M14	
N3 G110 C0	
N4 G0 X100 Z2	
N5 G745 XK0 YK0 K50 A0 Q3	
N6 G791 K30 A0 ZS0 ZE-5 P2 F0.15	
N7 M15	
END	

Example: Sequence of commands

	Simple drilling pattern
N.. G745 XK.. YK.. ZS.. ZE.. A.. W.. Q..	
. . .	
	Drilling pattern with deep-hole drilling
N.. G745 XK.. YK.. ZS.. A.. W.. Q..	
N.. G74 ZE.. P.. I..	
. . .	
	Milling pattern with linear slot
N.. G745 XK.. YK.. ZS.. ZE.. A.. W.. Q..	
N.. G791 K.. A.. Z..	
. . .	

Linear pattern on lateral surface G744



Cycle **G744** is used to machine linear drilling patterns or milling patterns in which the individual features are arranged at a regular spacing on the lateral surface.

Parameter combinations for defining the starting point and the pattern positions:

- Starting point of pattern: **Z**, **C**
- Pattern positions:
  - **W** and **Q**
  - **Wi** and **Q**

If you have not specified the **Final point XE**, the drilling/milling cycle or the figure definition of the next NC block is used as a reference.

Using this principle, you can combine pattern definitions with:

- Drilling cycles (**G71**, **G74**, **G36**)
- or milling cycles (figure definitions with **G314**, **G315**, **G317**)

Parameters:

- **XS**: **Start point** of drilling/milling operation (diameter value)
- **Z**: **Start point** of pattern (in polar coordinates)
- **XE**: **Final point** of drilling/milling operation (diameter value)
- **ZE**: **Final point** of pattern (default: Z)
- **C**: **Start angle** (in polar coordinates)
- **W**: **Final angle** of pattern (no input: holes/figures are arranged on the lateral surface at regular spacing)
- **Wi**: **End angle** – **Angle increment**
- **Q**: **Number of holes**
- **A**: **Angle** – orientation angle of the pattern
- **R**: **Length** – distance between first and last position (reference: unrolled lateral surface using **XS**)
- **Ri**: **Length** – distance to next position (reference: unrolled lateral surface using **XS**)

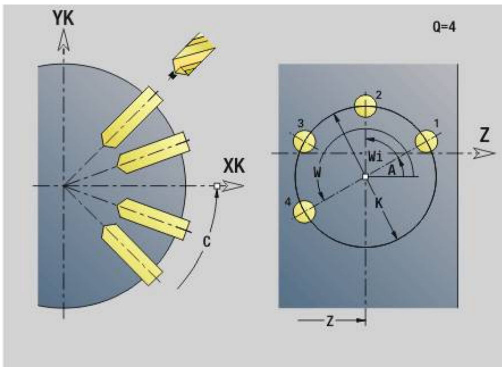
Example: G744

%744.nc	
N1 T7 G197 S1200 G195 F0.2 M104	
N2 M14	
N3 G110 C0	
N4 G0 X110 Z2	
N5 G744 XS102 Z-10 ZE-35 C0 W270 Q5	
N6 G71 XS102 K7	
N7 M15	
End, return to start M30END	

Example: Sequence of commands

	Simple drilling pattern
N.. G744 Z.. C.. XS.. XE.. ZE.. W.. Q..	
. . .	
	Drilling pattern with deep-hole drilling
N.. G744 Z.. C.. XS.. XE.. ZE.. W.. Q..	
N.. G74 XE.. P.. I..	
. . .	
	Milling pattern with linear slot
N.. G744 Z.. C.. XS.. XE.. ZE.. W.. Q..	
N.. G792 K.. A.. XS..	
. . .	

Circular pattern on lateral surface G746



Cycle **G746** is used to machine drilling patterns or milling patterns in which the individual features are arranged at a regular spacing in a circle or circular arc on the lateral surface.

Parameter combinations for defining the center of the pattern and the pattern positions:

- Center of pattern: **Z**, **C**
- Pattern positions:
  - **W** and **Q**
  - **Wi** and **Q**

If you have not specified the **Final point XE**, the drilling/milling cycle or the figure definition of the next NC block is used as a reference.

Using this principle, you can combine pattern definitions with:

- Drilling cycles (**G71**, **G74**, **G36**)
- or milling cycles (figure definitions with **G314**, **G315**, **G317**)

Parameters:

- **Z**: **Center** (in polar coordinates)
- **C**: **Angle** – center point (in polar coordinates)
- **XS**: **Start point** of drilling/milling operation (diameter value)
- **XE**: **Final point** of drilling/milling operation (diameter value)
- **K**: **Diameter** – pattern diameter
- **A**: **Start angle** – position of the first hole/figure
- **W**: **Final angle** – position of the last hole/figure
- **Wi**: **End angle** – **Angle increment**
- **Q**: **Number of holes**

- **V**: **Rotation dir.** (default: 0)
  - **V** = 0, without **W**: Figures are arranged on a full circle
  - **V** = 0, with **W**: Figures are arranged on the longer circular arc
  - **V** = 0, with **Wi**: The algebraic sign of **Wi** defines the direction (**Wi** < 0: clockwise)
  - **V** = 1, with **W**: Clockwise
  - **V** = 1, with **Wi**: Clockwise (algebraic sign of **Wi** has no effect)
  - **V** = 2, with **W**: Counterclockwise
  - **V** = 2, with **Wi**: Counterclockwise (algebraic sign of **Wi** has no effect)

Example: G746

%746.nc	
N1 T6 G197 S1200 G195 F0.2 M104	
N2 M14	
N3 G110 C0	
N4 G0 X110 Z2	
N5 G746 Z-40 C0 K40 Q8	
N6 G71 XS102 K7	
N7 M15	
END	

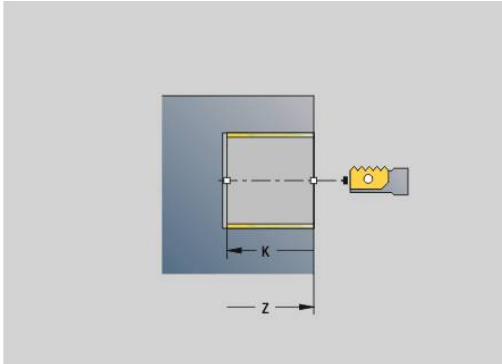
Example: Sequence of commands

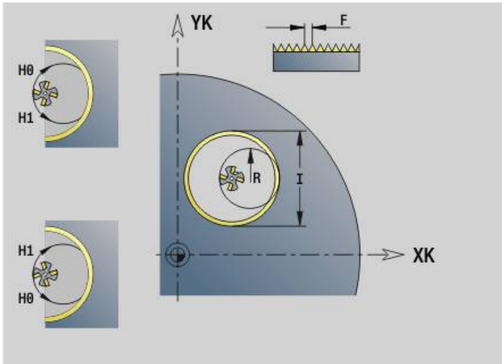
	Simple drilling pattern
N.. G746 Z.. C.. XS.. XE.. K.. A.. W.. Q..	
. . .	
	Drilling pattern with deep-hole drilling
N.. G746 Z.. C.. XS.. K.. A.. W.. Q..	
N.. G74 XE.. P.. I..	
. . .	
	Milling pattern with linear slot
N.. G746 Z.. C.. XS.. K.. A.. W.. Q..	
N.. G792 K.. A.. XS..	
. . .	

Thread milling axial G799

G799 mills a thread in existing holes.

Place the tool at the center of the hole before calling **G799**. The cycle positions the tool to the end point of the thread inside the hole. Then, the tool approaches at **Apprch angle R** and mills the thread. With each rotation, the tool executes an infeed movement by the **Thread pitch F**. Then, the cycle retracts the tool and returns it to the **Start pt. Z**. The **V** parameter allows you to define whether the thread is to be milled in one rotation or, with single-point tools, in multiple rotations.





Parameters:

- **ID**: **Milling contour** – name of the milling contour
- **NS**: **Block number of contour** – reference to the contour description
- **I**: **Thread diameter**
- **Z**: **Start pt. Z**
- **K**: **Thread depth**
- **R**: **Approach radius**
- **F**: **Thread pitch**
- **J**: **Direction of thread:**
  - **0: Right-hand thread**
  - **1: Left-hand thread**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **V**: **Milling method**
  - **0: One revolution** – the thread is milled in a 360-degree helix
  - **1: Two or more revolutions** – the thread is milled in several helix paths (single-point tool)

> Use thread-milling tools for cycle **G799**.

Example: G799

%799.nc	
N1 T9 G195 F0.2 G197 S800	
N2 G0 X100 Z2	
N3 M14	
N4 G110 Z2 C45 X100	
N5 G799 I12 Z0 K-20 F2 J0 H0	
N6 M15	
END	

6.23 C-axis commands

Reference diameter G120

G120 determines the **Reference diameter** of the unrolled lateral surface. Program **G120** when using **CY** for **G110** to **G113**. **G120** is a modal function.

Parameters:

- **X**: **Diameter**

Example: G120

. . .	
N1 T7 G197 S1200 G195 F0.2 M104	
N2 M14	
N3 G120 X100	Reference diameter
N4 G110 C0	
N5 G0 X110 Z5	
N6 G41 Q2 H0	
N7 G110 Z-20 CY0	
N8 G111 Z-40	
N9 G113 CY39.2699 K-40 J19.635	
N10 G111 Z-20	
N11 G113 CY0 K-20 J19.635	
N12 G40	
N13 G110 X105	
N14 M15	
. . .	

Datum shift in C axis G152

G152 defines an absolute datum for the C axis (reference: Reference point, C axis). The datum is valid until the end of the program.

Parameters:

- **C**: **Angle** – spindle position of new C-axis datum

Example: G152

. . .	
N1 M5	
N2 T7 G197 S1010 G193 F0.08 M104	
N3 M14	
N4 G152 C30	C-axis datum
N5 G110 C0	
N6 G0 X122 Z-50	
N7 G71 X100	
N8 M15	
. . .	

Standardize C axis G153

G153 resets a traverse angle >360° or <0° to an angle between 0° and 360°—without moving the C axis.

> **G153** is only used for lateral-surface machining. An automatic modulo 360° function is carried out on the face.

Short path in C G154

G154 defines that the C axis moves on the shorter path during positioning.

Parameters:

- **H**: Traverse on shorter path **On/Off**
  - **0: OFF**
  - **1: ON**

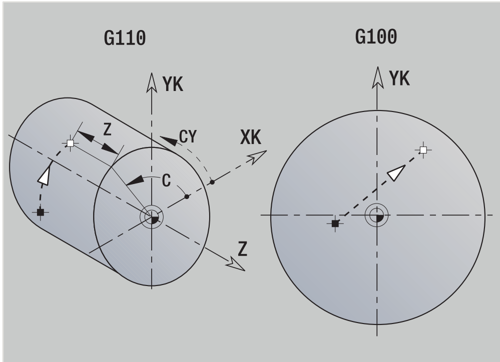
Example: G154

. . .	
N1 G110 C0	
N2 G154 H1	
N3 G110 C350	Traverse path -10°
N4 G110 C10	Traverse path +20°
N5 G154 H0	
N6 G110 C350	Traverse path +340°
. . .	

6.24 Front and rear face machining

Rapid traverse on front/rear face G100

G100 moves at rapid traverse along the shortest path to the **Final point**.



> With **G100** the tool performs a linear movement.
> To position the workpiece to a defined angle, use **G110.**

Parameters:

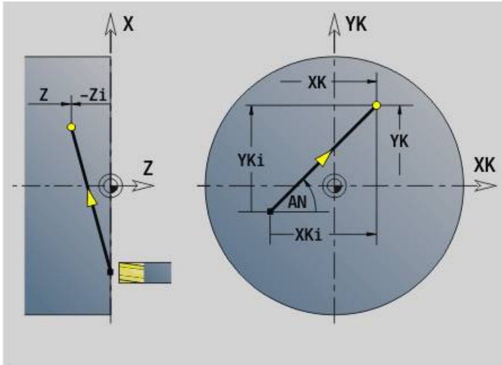
- **X**: **Final point** (diameter value)
- **C**: **End angle**
- **XK**: **Final point** (in Cartesian coordinates)
- **YK**: **Final point** (in Cartesian coordinates)
- **Z**: **Final point**

> Programming:
> - **X**, **C**, **XK**, **YK**, **Z**: Absolute, incremental or modal
> - Program either **X**–**C** or **XK**–**YK**

Example: G100

. . .	
N1 T7 G197 S1200 G195 F0.2 M104	
N2 M14	
N3 G110 C0	
N4 G0 X100 Z2	
N6 G100 XK20 YK5	Rapid traverse, face
N7 G101 XK50	
N8 G103 XK5 YK50 R50	
N9 G101 XK5 YK20	
N10 G102 XK20 YK5 R20	
N11 G14	
N12 M15	
. . .	

Linear segment on front/rear face G101



G101 moves the tool on a linear path at the feed rate to the **Final point**.

Parameters:

- **X**: **Final point** (diameter value)
- **C**: **End angle**
- **XK**: **Final point** (in Cartesian coordinates)
- **YK**: **Final point** (in Cartesian coordinates)
- **Z**: **Final point**

Parameters for contour description (**G80**):

- **AN**: **Angle** to positive XK axis
- **BR**: **Chamf./round.** – defines the transition to the next contour element
  When entering a **Chamf./round.**, program the theoretical end point.
  - No entry: Tangential transition
  - **BR** = 0: No tangential transition
  - **BR** > 0: Rounding radius
  - **BR** < 0: Width of chamfer
- **Q**: **Intersect. pt.** or **Final point** if the line segment intersects a circular arc (default: 0)
  - 0: Near point of intersection
  - 1: Far point of intersection

Programming:
- **X**, **C**, **XK**, **YK**, **Z**: Absolute, incremental or modal
- Program either **X**–**C** or **XK**–**YK**

Using the parameters **AN**, **BR** and **Q** is only allowed if the contour description is concluded by **G80** and used for a cycle.

Example: G101

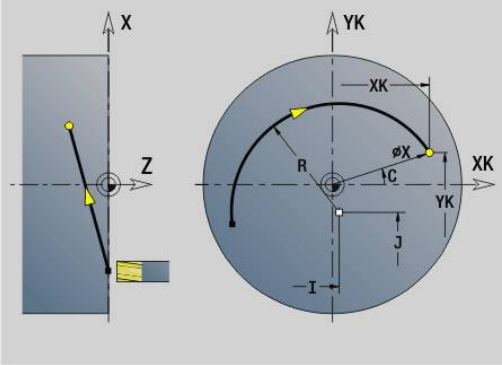
. . .	
N1 T70 G197 S1200 G195 F0.2 M104	
N2 M14	
N3 G110 C0	
N4 G0 X110 Z2	
N5 G100 XK50 YK0	
N6 G1 Z-5	
N7 G42 Q1	
N8 G101 XK40	Linear path, face
N9 G101 YK30	
N10 G103 XK30 YK40 R10	
N11 G101 XK-30	
N12 G103 XK-40 YK30 R10	
N13 G101 YK-30	
N14 G103 XK-30 YK-40 R10	
N15 G101 XK30	
N16 G103 XK40 YK-30 R10	
N17 G101 YK0	
N18 G100 XK110 G40	
N19 G0 X120 Z50	
N20 M15	
. . .	

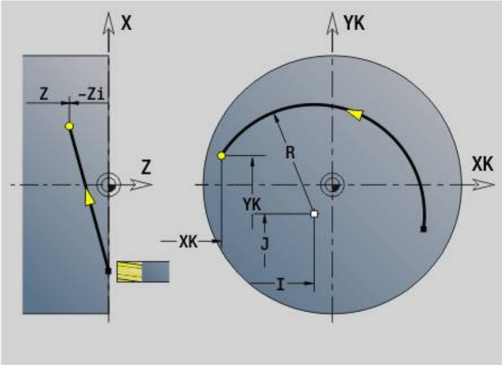
Circular arc on front/rear face G102/G103

G102 and **G103** move the tool along a circular arc at the feed rate to the **Final point**. The direction of rotation is shown in the help graphic.

Parameters:

- **X**: **Final point** (diameter value)
- **C**: **End angle**
- **XK**: **Final point** (in Cartesian coordinates)
- **YK**: **Final point** (in Cartesian coordinates)
- **R**: **Radius**
- **I**: **Center** (in Cartesian coordinates)
- **J**: **Center** (in Cartesian coordinates)
- **K**: **Center** for **H** = 2 or 3 (Z direction)
- **Z**: **Final point**
- **H**: **Rot. plane** – machining plane (default: 0)
  - **H** = 0 or 1: Machining in XY plane (front face)
  - **H** = 2: Machining in YZ plane
  - **H** = 3: Machining in XZ plane





Parameters for contour description (**G80**):

- **AN**: **Angle** to positive XK axis
- **BR**: **Chamf./round.** – defines the transition to the next contour element
  When entering a **Chamf./round.**, program the theoretical end point.
  - No entry: Tangential transition
  - **BR** = 0: No tangential transition
  - **BR** > 0: Rounding radius
  - **BR** < 0: Width of chamfer
- **Q**: **Intersect. pt.** or **Final point** if the line segment intersects a circular arc (default: 0)
  - 0: Near point of intersection
  - 1: Far point of intersection

> Using the parameters **AN**, **BR** and **Q** is only allowed if the contour description is concluded by **G80** and used for a cycle.

Example: G102, G103

. . .	
N1 T7 G197 S1200 G195 F0.2 M104	
N2 M14	
N3 G110 C0	
N4 G0 X100 Z2	
N6 G100 XK20 YK5	
N7 G101 XK50	
N8 G103 XK5 YK50 R50	Arc
N9 G101 XK5 YK20	
N10 G102 XK20 YK5 R20	
N12 M15	
. . .	

If you program **H**=2 or **H**=3, you can machine linear slots with a circular base.

Define the circle center with:

- **H** = 2: with **I** and **K**
- **H** = 3: with **J** and **K**

Programming:

- **X**, **C**, **XK**, **YK**, **Z**: Absolute, incremental or modal
- **I**, **J**, **K**: Absolute or incremental
- Program either **X**–**C** or **XK**–**YK**
- Program either center or radius
- For radius: Only arcs <= 180° are possible
- End point in the coordinate origin: Program **XK**=0 and **YK**=0.

6.25 Lateral surface machining

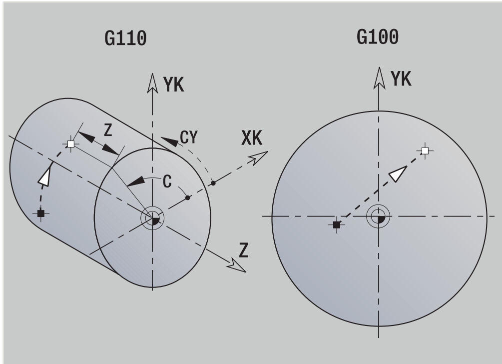
Rapid traverse on lateral surface G110

G110 moves at rapid traverse to the **Final point**.

G110 is recommended for positioning the C axis to a defined angle (programming: **N.. G110 C...**).

Parameters:

- **Z**: **Final point**
- **C**: **End angle**
- **CY**: **Final point** as a linear dimension (reference: unrolled lateral surface, using the **Reference diameter**)
- **X**: **Final point** (diameter value)



> Programming:
> - **Z**, **C**, **CY**: Absolute, incremental, or modal
> - Program either **Z**–**C** or **Z**–**CY**

Example: G110

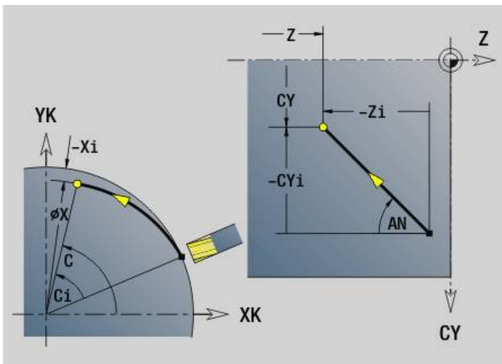
. . .	
N1 T8 G197 S1200 G195 F0.2 M104	
N2 M14	
N3 G120 X100	
N4 G110 C0	Rapid traverse, lateral surface
N5 G0 X110 Z5	
N6 G110 Z-20 CY0	
N7 G111 Z-40	
N8 G113 CY39.2699 K-40 J19.635	
N9 G111 Z-20	
N10 G113 CY0 K-20 J19.635	
N11 M15	
. . .	

Surface linear G111

G111 moves the tool on a linear path at the feed rate to the **Final point**.

Parameters:

- **Z**: **Final point**
- **C**: **End angle**
- **CY**: **Final point** as a linear dimension (reference: unrolled lateral surface, using the **Reference diameter**)
- **X**: **Final point** (diameter value)



Parameters for contour description (**G80**):

- **AN**: **Angle** to positive Z axis

- **BR**: **Chamf./round.** – defines the transition to the next contour element
  When entering a **Chamf./round.**, program the theoretical end point.
  - No entry: Tangential transition
  - **BR** = 0: No tangential transition
  - **BR** > 0: Rounding radius
  - **BR** < 0: Width of chamfer
- **Q**: **Intersect. pt.** or **Final point** if the line segment intersects a circular arc (default: 0)
  - 0: Near point of intersection
  - 1: Far point of intersection

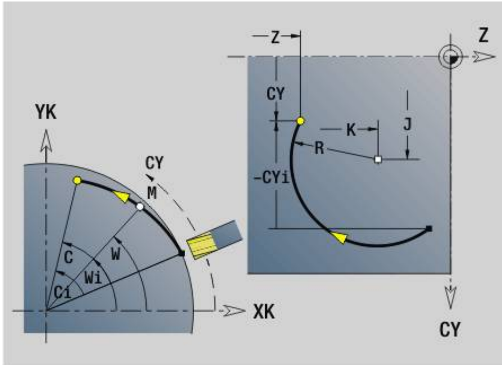
> Programming:
> - **Z**, **C**, **CY**: Absolute, incremental, or modal
> - Program either **Z**–**C** or **Z**–**CY**

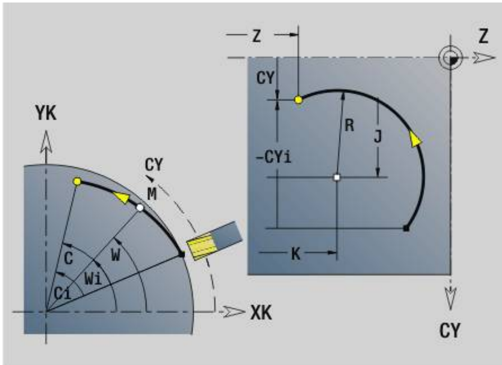
> Using the parameters **AN**, **BR** and **Q** is only allowed if the contour description is concluded by **G80** and used for a cycle.

Example: G111

...	
N1 T8 G197 S1200 G195 F0.2 M104	
N2 M14	
N3 G120 X100	
N4 G110 C0	
N5 G0 X110 Z5	
N6 G41 Q2 H0	
N7 G110 Z-20 CY0	
N8 G111 Z-40	Linear path, lateral surface
N9 G113 CY39.2699 K-40 J19.635	
N10 G111 Z-20	
N11 G113 CY0 K-20 J19.635	
N12 G40	
N13 G110 X105	
N14 M15	
...	

Circular arc on lateral surface G112/G113





G112 and **G113** move the tool along a circular arc at the feed rate to the **Final point**.

Parameters:

- **Z**: **Final point**
- **C**: **End angle**
- **CY**: **Final point** as a linear dimension (reference: unrolled lateral surface, using the **Reference diameter**)
- **R**: **Radius**
- **K**: **Center** (in Z)
- **J**: **Center** as linear dimension (reference: reference diameter of the unrolled lateral surface)
- **W**: **Center** – **Angle** (angular direction: see help graphic)
- **X**: **Final point** (diameter value)

Parameters for contour description (**G80**):

- **AN**: **Angle** to positive Z axis
- **BR**: **Chamf./round.** – defines the transition to the next contour element
  When entering a **Chamf./round.**, program the theoretical end point.
  - No entry: Tangential transition
  - **BR** = 0: No tangential transition
  - **BR** > 0: Rounding radius
  - **BR** < 0: Width of chamfer
- **Q**: **Intersect. pt.** or **Final point** if the line segment intersects a circular arc (default: 0)
  - 0: Near point of intersection
  - 1: Far point of intersection

> Using the parameters **AN**, **BR** and **Q** is only allowed if the contour description is concluded by **G80** and used for a cycle.

> Programming:
> - **Z**, **C**, **CY**: Absolute, incremental, or modal
> - **K**, **W**, **J**: Absolute or incremental
> - Program either **Z**–**C** or **Z**–**CY** and **K**–**J**
> - Program either center or radius
> - For radius: Only arcs <= 180° are possible

Example: G112, G113

. . .	
N1 T8 G197 S1200 G195 F0.2 M104	
N2 M14	
N3 G120 X100	
N4 G110 C0	
N5 G0 X110 Z5	
N7 G110 Z-20 CY0	
N8 G111 Z-40	
N9 G113 CY39.2699 K-40 J19.635	Arc
N10 G111 Z-20	
N11 G112 CY0 K-20 J19.635	
N13 M15	
. . .	

6.26 Milling cycles

Overview of milling cycles

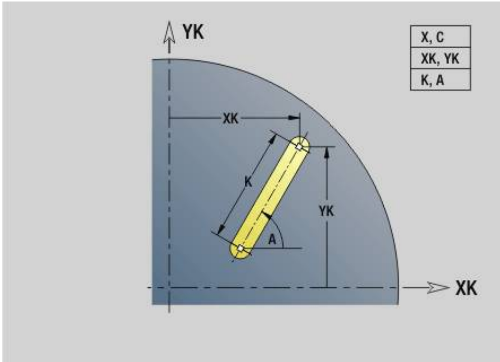
- **G791 Linear slot, face**. The position and length of the slot are defined directly in the cycle; slot width = cutter diameter
  Further information: "Linear slot, front face G791", Page 453
- **G792 Linear slot, surf.**. The position and length of the slot are defined directly in the cycle; slot width = cutter diameter
  Further information: "Linear slot, lat. surface G792", Page 454
- **G793 Contour milling cycle, face**. The contour is described directly after the cycle and concluded with **G80** (MANUALplus 4110 compatibility cycle)
  Further information: "Contour and figure milling cycle on front face G793", Page 455
- **G794 Contour milling cycle, lateral**. The contour is described directly after the cycle and concluded with **G80** (MANUALplus 4110 compatibility cycle)
  Further information: "Contour and figure milling cycle on lateral surface G794", Page 457
- **G797 Area milling**. Mills figures (circles, polygons, individual surfaces, contours) as islands on the front face
  Further information: "Area milling on front face G797", Page 460
- **G798 Hel. slot milling**. Mills a helical slot on the lateral surface; slot width = cutter diameter
  Further information: "Helical slot milling G798", Page 462
- **G840 Contour milling**. Mills ICP contours and figures. For closed contours, milling operations can be done inside/outside the contour or on the contour itself. For open contours, milling operations can be done at the left/right of the contour or on the contour itself. **G840** is used on the face and lateral surface
  Further information: "Mill. contour G840", Page 463
- **G845 Pocket milling - roughing**. Roughs out closed ICP contours and figures on the face and lateral surface
  Further information: "Pocket milling - roughing G845", Page 472
- **G846 Pocket milling - finishing**. Finishes closed ICP contours and figures on the face and lateral surface
  Further information: "Pocket milling - finishing G846 (Y axis)", Page 664
- **G847 Trochoidal contour milling**. Roughs out open or closed ICP contours on the face and lateral surface using trochoidal milling
  Further information: "Trochoidal contour milling G847 ", Page 480
- **G848 Trochoidal pocket milling**. Roughs out figures or figure patterns on the face and lateral surface using trochoidal milling
  Further information: "Trochoidal pocket milling G847 ", Page 482

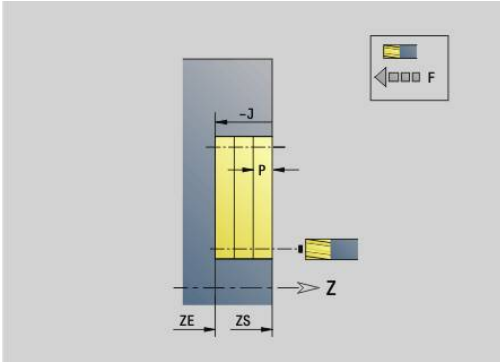
Contour definitions in the machining section (figures):

- Face
  - **G301 Linear slot**
    Further information: "Linear slot on front/rear face G301-Geo", Page 317
  - **G302/G303 Circular slot**
    Further information: "Circular slot on front/rear face G302-/G303-Geo", Page 318
  - **G304 Full circle**
    Further information: "Full circle on front/rear face G304-Geo", Page 318
  - **G305 Rectangle**
    Further information: "Rectangle on front/rear face G305-Geo", Page 319
  - **G307 Eccentric polygon**
    Further information: "Polygon on front/rear face G307-Geo", Page 320
- Lateral surface
  - **G311 Linear slot**
    Further information: "Linear slot on lateral surface G311-Geo", Page 325
  - **G312/G313 Circular slot**
    Further information: "Circular slot on lateral surface G312-/G313-Geo", Page 326
  - **G314 Full circle**
    Further information: "Full circle on lateral surface G314-Geo", Page 326
  - **G315 Rectangle**
    Further information: "Rectangle, surface G315-Geo", Page 327
  - **G317 Eccentric polygon**
    Further information: "Polygon on lateral surface G317-Geo", Page 328

Linear slot, front face G791

G791 mills a slot from the current tool position to the **Final point**. The slot width equals the diameter of the milling cutter. Oversizes (allowances) are not taken into account.





Parameters:

- **X**: **Diameter** – final point of slot (in polar coordinates)
- **C**: **End angle** – final point of slot (in polar coordinates; for angle direction, see help graphic)
- **XK**: **Final point** (in Cartesian coordinates)
- **YK**: **Final point** (in Cartesian coordinates)
- **K**: **Length**
- **A**: **Angle** – Angle of rotation
- **ZE**: **Milling floor**
- **ZS**: **Millg. top edge**
- **J**: **Milling depth**
  - **J** > 0: Infeed direction -Z
  - **J** < 0: Infeed direction +Z
- **P**: **Max. approach** (default: milling in one infeed)
- **F**: **Approach feed** for plunging (default: active feed rate)

Parameter combinations for definition of the end point: see help graphic

Parameter combinations for definition of the milling plane:

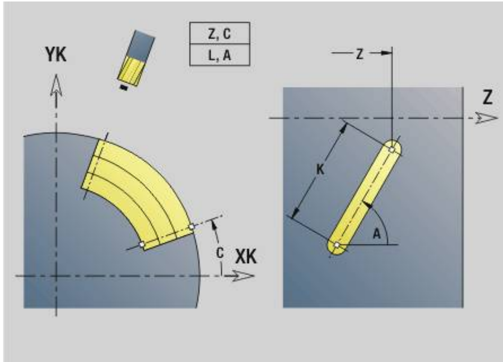
- **Milling floor ZE**, **Millg. top edge ZS**
- **Milling floor ZE**, **Milling depth J**
- **Millg. top edge ZS**, **Milling depth J**
- **Milling floor ZE**

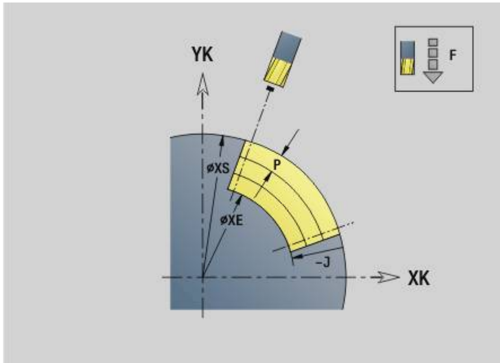
> - Rotate the spindle to the desired angle position before calling **G791**.
> - If you use a spindle positioning device (no C axis), an axial slot is machined centrically to the rotary axis.
> - If **J** or **ZS** is defined, the tool approaches to safety clearance in **Z** and then mills the slot. If **J** and **ZS** are not defined, the milling cycle starts from the current tool position

Example: G791

%791.nc	
N1 T7 G197 S1200 G195 F0.2 M104	
N2 M14	
N3 G110 C0	
N4 G0 X100 Z2	
N5 G100 XK20 YK5	
N6 G791 XK30 YK5 ZE-5 J5 P2	
N7 M15	
END	

Linear slot, lat. surface G792





G792 mills a slot from the current tool position to the **Final point**. The slot width equals the diameter of the milling cutter. Oversizes (allowances) are not taken into account.

Parameters:

- **Z**: **Final point**
- **C**: **End angle**
- **K**: **Length**
- **A**: **Angle** – Angle of rotation
- **XE**: **Milling floor**
- **XS**: **Upper edge of milling**
- **J**: **Milling depth**
  - **J** > 0: Infeed direction –X
  - **J** < 0: Infeed direction +X
- **P**: **Max. approach** (default: milling in one infeed)
- **F**: **Approach feed** for plunging (default: active feed rate)

Parameter combinations for definition of the end point: see help graphic

Parameter combinations for definition of the milling plane:

- **Milling floor XE**, **Millg. top edge XS**
- **Milling floor XE**, **Milling depth J**
- **Millg. top edge XS**, **Milling depth J**
- **Milling floor XE**

> - Rotate the spindle to the desired angle position before calling **G792**
> - If you use a spindle positioning device (no C axis), a radial slot is machined parallel to the Z axis.
> - If **J** or **XS** is defined, the tool approaches to safety clearance in X and then mills the slot. If **J** and **XS** are not defined, the milling cycle starts from the current tool position

Example: G792

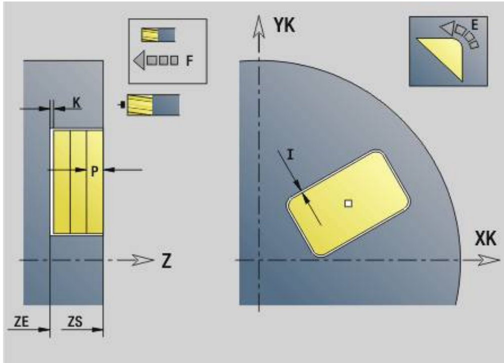
%792.nc	
N1 T8 G197 S1200 G195 F0.2 M104	
N2 M14	
N3 G110 C0	
N4 G0 X110 Z5	
N5 G0 X102 Z-30	
N6 G792 K25 A45 XE97 J3 P2 F0.15	
N7 M15	
END	

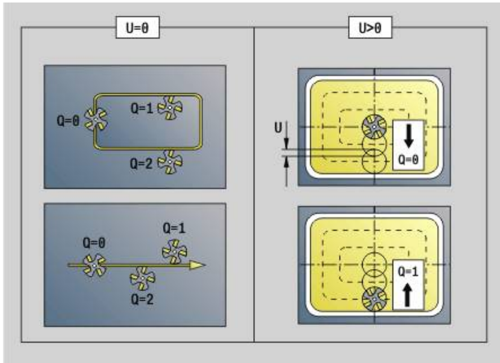
Contour and figure milling cycle on front face G793

G793 mills figures or (open or closed) free contours.

G793 is followed by:

- The figure to be milled with:
  - Contour definition of the figure (**G301** to **G307**)
    Further information: "Front/Rear face contours", Page 315
  - Conclusion of milling contour (**G80**)
- The free contour with:
  - Starting point of milling contour (**G100**)
  - Milling contour (**G101**, **G102**, **G103**)
  - Conclusion of milling contour (**G80**)





> **i** Preferentially use **ICP** and the **G840**, **G845** and **G846** cycles to program the contour description in the geometry section.

Parameters:

- **ZS**: **Millg. top edge**
- **ZE**: **Milling floor**
- **P**: **Max. approach** (default: milling in one infeed)
- **U**: **Overlap factor** – contour milling or pocket milling (default: 0)
  - **U** = 0: Contour milling
  - **U** > 0: Pocket milling—minimum overlap of milling paths = **U***milling diameter
- **R**: **Apprch angle** (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
- **I**: **Contour-parallel oversize**
- **K**: **O-size Z**
- **F**: **Approach feed** for plunging (default: active feed rate)
- **E**: **Reduced feed** for circular elements (default: active feed rate)
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**

- Q: **Cycle type** (default: 0) – Depending on **U**, the following applies:
  - Contour milling (**U** = 0)
    - **Q**=0: Center of milling cutter on the contour
    - **Q**=1, closed contour: Inside milling
    - **Q**=1, open contour: Left in machining direction
    - **Q**=2, closed contour: Outside milling
    - **Q**=2, open contour: Right in machining direction
    - **Q**=3, open contour: Milling location depends on H and the direction of tool rotation—see help graphic
  - Pocket milling (**U** > 0)
    - **Q**=0: From the inside toward the outside
    - **Q**=1: From the outside toward the inside
- O: **Roughing/Finish**
  - **0: Roughing**
  - **1: Finishing**

- Milling depth: The cycle calculates the depth from the milling top edge and the milling floor—taking the oversizes into account
- Milling cutter radius compensation: Effective (except for contour milling with **Q** = 0)
- Approach and departure: For closed contours, the point of the surface normal from the tool position to the first contour element is the point of approach and departure. If no surface normal intersects the tool position, the starting point of the first element is the point of approach and departure. For contour milling and finishing (pocket milling), define with the approach radius whether the tool is to approach directly or in an arc.
- **G57**/**G58** oversizes are taken into account if the oversizes **I**, **K** are not programmed:
  - **G57**: Oversize in X, Z direction
  - **G58**: The oversize shifts the milling contour as follows:
    - With inside milling and closed contour: The contour is contracted
    - With outside milling and closed contour: The contour is expanded
    - With open contour and **Q**=1: Left in machining direction
    - With open contour and **Q**=2: Right in machining direction

Contour and figure milling cycle on lateral surface G794

G794 mills figures or (open or closed) free contours.

G794 is followed by:

- The figure to be milled with:
  - Contour definition of the figure (**G311** to **G317**)
    Further information: "Lateral surface contours", Page 323
  - Conclusion of contour definition (**G80**)
- The free contour with:
  - Starting point (**G110**)
  - Contour definition (**G111**, **G112**, **G113**)
  - Conclusion of milling contour (**G80**)

> Preferentially use **ICP** and the **G840**, **G845** and **G846** cycles to program the contour description in the geometry section.

Parameters:

- **XS**: **Upper edge of milling**
- **XE**: **Milling floor**
- **P**: **Max. approach** (default: milling in one infeed)
- **U**: **Overlap factor** – contour milling or pocket milling (default: 0)
  - **U** = 0: Contour milling
  - **U** > 0: Pocket milling—minimum overlap of milling paths = **U***milling diameter
- **R**: **Apprch angle** (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
- **K**: **Contour-parallel oversize**
- **I**: **O-size X**
- **F**: **Approach feed** for plunging (default: active feed rate)
- **E**: **Reduced feed** for circular elements (default: active feed rate)
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**

- **Q**: **Cycle type** (default: 0) – Depending on **U**, the following applies:
  - Contour milling (**U** = 0)
    - **Q**=0: Center of milling cutter on the contour
    - **Q**=1, closed contour: Inside milling
    - **Q**=1, open contour: Left in machining direction
    - **Q**=2, closed contour: Outside milling
    - **Q**=2, open contour: Right in machining direction
    - **Q**=3, open contour: Milling location depends on H and the direction of tool rotation—see help graphic
  - Pocket milling (**U** > 0)
    - **Q**=0: From the inside toward the outside
    - **Q**=1: From the outside toward the inside
- **O**: **Roughing/Finish**
  - **0: Roughing**
  - **1: Finishing**

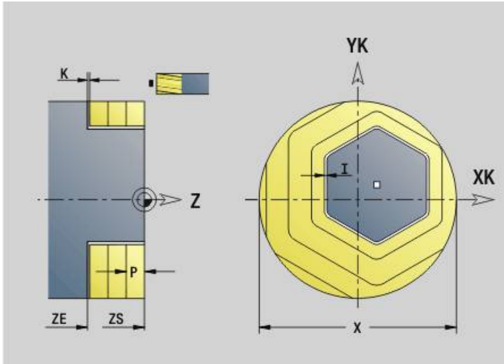
Example: G794

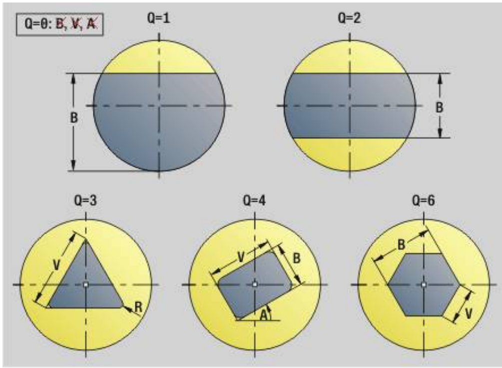
%794.nc	
N1 T7 G197 S1200 G195 F0.2 M104	
N2 M14	
N3 G110 C0	
N4 G0 X110 Z5	
N5 G794 XS100 XE97 P2 U0.5 R0 K0.5 F0.15	
N6 G314 Z-35 C0 R20	
N7 G80	
N8 M15	
END	

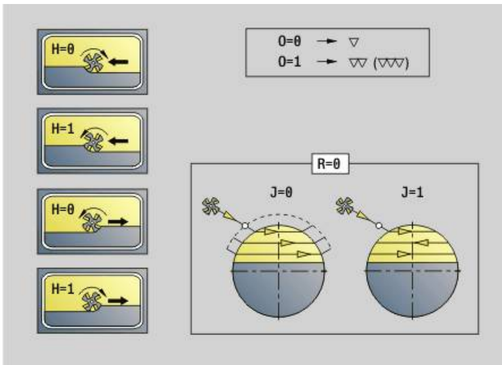
- Milling depth: The cycle calculates the depth from the milling top edge and the milling floor—taking the oversizes into account
- Milling cutter radius compensation: Effective (except for contour milling with **Q** = 0)
- Approach and departure: For closed contours, the point of the surface normal from the tool position to the first contour element is the point of approach and departure. If no surface normal intersects the tool position, the starting point of the first element is the point of approach and departure. For contour milling and finishing (pocket milling), define with the approach radius whether the tool is to approach directly or in an arc.
- **G57**/**G58** oversizes are taken into account if the oversizes **I**, **K** are not programmed:
  - **G57**: Oversize in X, Z direction
  - **G58**: The oversize shifts the milling contour as follows:
    - With inside milling and closed contour: The contour is contracted
    - With outside milling and closed contour: The contour is expanded
    - With open contour and **Q**=1: Left in machining direction
    - With open contour and **Q**=2: Right in machining direction

Area milling on front face G797

Depending on **Q**, **G797** mills surfaces, a polygon, or the figure defined in the command following **G797**.







Parameters:

- **ID**: **Milling contour** – name of the milling contour
- **NS**: **Starting block no. of contour** – beginning of contour section
  - Figures: Block number of the figure
  - Free closed contour: First contour element (not starting point)
- **X**: **Limit diameter**
- **ZS**: **Millg. top edge**
- **ZE**: **Milling floor**
- **B**: **Width/Width across flats**

  Omit for **Q**=0: Defines the remaining material. For an even number of surfaces, you can program **B** as an alternative to **V**.
  - **Q** =1: **B**=Residual depth
  - **Q**>=2: **B**=Width across flats
- **V**: **Edge length** (omitted for **Q**=0)
- **R**: **Chamf./round.** (default: 0)
- **A**: **Inclinat. ang.** omitted for **Q** = 0 (reference: see help graphic)
- **Q**: **No. of surfaces** (default: 0; range: 0 <= **Q** <= 127)
  - **Q** = 0: **G797** is followed by a figure definition (**G301**.. **G307**, **G80**) or a closed contour definition (**G100**, **G101** to **G103**, **G80**)
  - **Q** = 1: One surface
  - **Q** = 2: Two surfaces offset by 180°
  - **Q** = 3: Triangle
  - **Q** = 4: Rectangle, square
  - **Q** > 4: Polygon
- **P**: **Max. approach** (default: milling in one infeed)
- **U**: **Overlap factor** – overlap of milling paths = **U** * milling diameter (default: 0.5)
- **I**: **Contour-parallel oversize**
- **K**: **O-size Z**
- **F**: **Approach feed** for plunging (default: active feed rate)
- **E**: **Reduced feed** for circular elements (default: active feed rate)
- **H**: **Mill cutting direction**
  - **0: Roughing**
  - **1: Finishing**
- **O**: **Roughing/Finish**
  - **0: Roughing**
  - **1: Finishing**
- **J**: **Mill direction**
  - **0: Unidirectional**
  - **1: Bidirectional**

> Programming:
> - The cycle calculates the milling depth from **ZS** and **ZE**, taking the oversizes into account
> - Surfaces and figures defined with **G797** (**Q**>0) are symmetric with respect to the center. A figure defined in the following command can be outside the center.

G797 Q0 .. is followed by:

- The figure to be milled with:
  - Contour definition of the figure (**G301** to **G307**)
    Further information: "Front/Rear face contours", Page 315
  - Conclusion of contour definition (**G80**)
- The free contour with:
  - Starting point of milling contour (**G100**)
  - Milling contour (**G101**, **G102**, **G103**)
  - Conclusion of milling contour (**G80**)

Example: G797

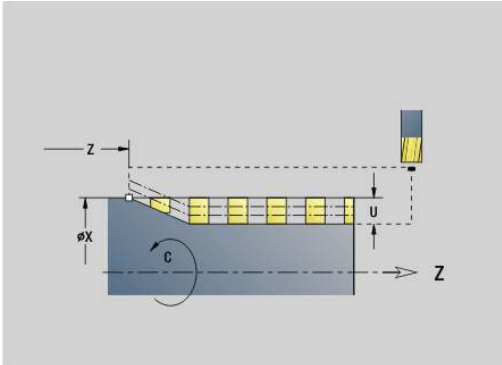
```
%797.nc
N1 T9 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X100 Z2
N5 G797 X100 Z0 ZE-5 B50 R2 A0 Q4 P2 U0.5
N6 G100 Z2
N7 M15
END
```

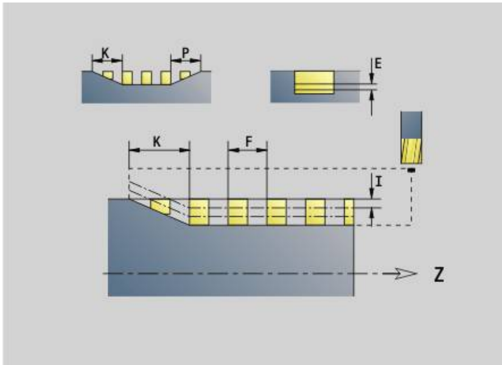
Example: G797 / G304

```
%304_G305.nc
N1 T7 G197 S1200 G195 F0.2 M104
N2 M14
N3 G110 C0
N4 G0 X100 Z2
N5 G797 X100 ZS0 ZE-5 Q0 P2 F0.15
N6 G304 XK20 YK5 R20
N7 G80
N4 G0 X100 Z2
N5 G797 X100 ZS0 ZE-5 Q0 P2 F0.15
N6 G305 XK20 YK5 R6 B30 K45 A20
N7 G80
N8 M15
END
```

Helical slot milling G798

G798 mills a helical slot from the current tool position to the **Final point X**, **Z**. The slot width equals the diameter of the milling cutter.





Parameters:

- X: **Final point** (diameter value)
- Z: **Final point**
- C: **Start angle**
- F: **Thread pitch**
  - **F** positive: Right-hand thread
  - **F** negative: Left-hand thread
- P: **Run-in lgth** – Ramp at the beginning of the slot
- K: **Thread runout length** – Ramp at the end of the slot
- U: **Thread depth**
- I: **Max. approach**
- E: **Reducing value** for infeed reduction (default: 1)
- D: **No.gears**

Infeeds:

- **Max. approach I** is used for the first infeed movement.
- The control calculates all subsequent infeed movements as follows: Current infeed = **I** * (1 – (**n** – 1) * **E**)
(**n**: **n** - **nth** infeed)
- The infeed movement is reduced down to >= 0.5 mm. Following that, each infeed movement will amount to 0.5 mm.

> You can mill a helical slot only from the outside.

Example: G798

%798.nc	
N1 T9 G197 S1200 G195 F0.2 M104	
N2 M14	
N3 G110 C0	
N4 G0 X80 Z15	
N5 G798 X80 Z-120 C0 F20 K20 U5 I1	
N6 G100 Z2	
N7 M15	
END	

Mill. contour G840

G840 – fundamentals

G840 mills or deburrs open or closed contours (figures or free contours).

Plunge strategies: Depending on the cutter you are using, select one of the following strategies:

- Vertical plunge: The cycle moves the tool to the starting point; the tool plunges and mills the contour
- Calculate positions, predrill, mill. The machining process is performed in the following steps:
  - Insert drill.
  - Calculate hole positions with **G840 A1 ..**
  - Predrill with **G71 NF..**
  - Call cycle **G840 A0 ..** The cycle positions the tool above the hole; the tool plunges and mills the contour
- Predrilling, milling. The machining process is performed in the following steps:
  - Predrill with **G71 ..**
  - Position the cutter above the hole. Call cycle **G840 A0 ..** The cycle plunges and mills the contour or contour section

If the milling contour consists of multiple sections, **G840** takes all the sections of the contour into account for predrilling and milling. Call **G840 A0 ..** separately for each section when calculating the hole positions without **G840 A1 ..**.

Oversize: A **G58** oversize shifts the contour to be milled in the direction given in cycle type **Q**.

- With inside milling and closed contour: Shifted inward
- With outside milling and closed contour: Shifted outward
- Open contour: Shifts to the left or right depending on **Q**

> - If **Q** = 0, oversizes are not taken into account
> - **G57** and negative **G58** oversizes are not taken into account

G840 – calculating hole positions

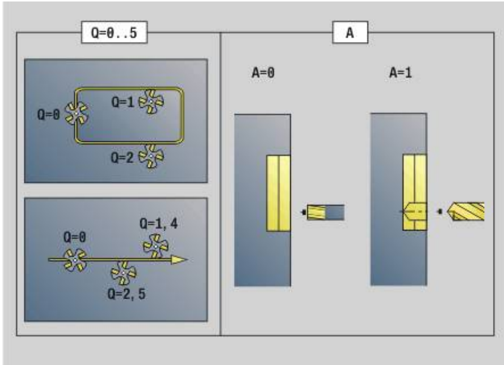
G840 A1 .. calculates the hole positions and stores them at the reference specified in **NF**. Program only the parameters given in the following table.

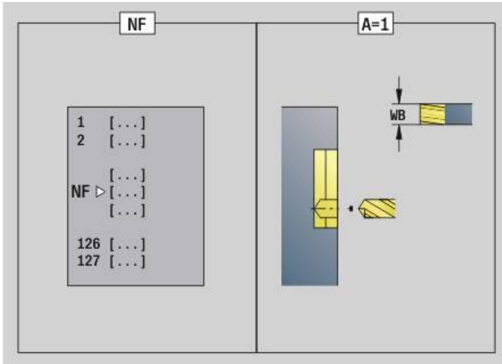
See also:

- **G840**—Fundamentals
  Further information: "G840 – fundamentals", Page 463
- **G840**—Milling
  Further information: "G840 – milling", Page 466

Parameters:

- **Q**: **Cycle type** – milling location
  - Open contour. If there is any overlapping, **Q** defines whether the first section (as of starting point) or the entire contour is to be machined
    - **Q** = 0: Center of milling cutter on the contour (hole position = starting point)
    - **Q** = 1: Machining at the left of the contour. If there is any overlapping, only the first area of the contour is machined
    - **Q** = 2: Machining at the right of the contour. If there is any overlapping, only the first area of the contour is machined
    - **Q** = 3: Not allowed
    - **Q** = 4: Machining at the left of the contour. If there is any overlapping, the entire contour is machined
    - **Q** = 5: Machining at the right of the contour. If there is any overlapping, the entire contour is machined
  - Closed contours
    - **Q** = 0: Center of milling cutter on the contour (hole position = starting point)
    - **Q** = 1: Inside milling
    - **Q** = 2: Outside milling
    - **Q** = 3 to 5: Not allowed
- **ID**: **Milling contour** – name of the milling contour
- **NS**: **Starting block no. of contour** – beginning of contour section
  - Figures: Block number of the figure
  - Free closed contour: First contour element (not starting point)
  - Open contour: First contour element (not starting point)





- **NE**: **Contour end block no.** – end of contour section
  - Figures, free closed contour: No input
  - Open contour: last contour element
  - Contour consists of one element:
    - No input: Machining in contour direction
    - **NS** = **NE** programmed: Machining against the contour direction
- **D**: **Start elem.no.**

  The direction of contour definition for figures is counterclockwise.

  The first contour element for figures:
  - Circular slot: The larger arc
  - Full circle: The upper semicircle
  - Rectangles, polygons and linear slots: The orientation angle points to the first contour element.
- **V**: **End: elem.no.**
- **A**: **(Mill=0/PredrillPos=1)**
- **NF**: **Position mark** – reference at which the cycle stores the predrill hole positions (range: 1 to 127)
- **WB**: **Diameter rework**

Program **D** and **V** to machine parts of a figure.

- The cycle takes the diameter of the active tool into account when calculating the hole positions. Therefore, you need to insert the drill before calling **G840 A1 ..**.
- Program oversizes for calculating the hole positions and for milling

NOTICE

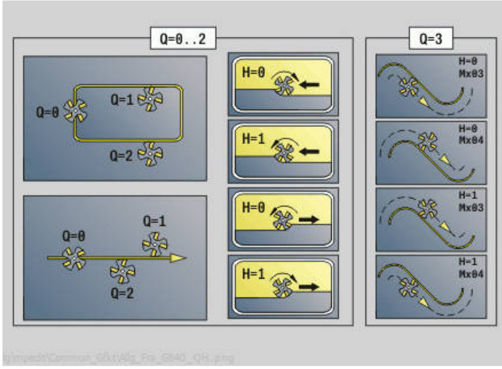
Danger of collision!

G840 overwrites any predrill hole positions that may still be stored at the **Position mark NF** without displaying a confirmation prompt. Danger of collision during subsequent machining operations!

- Observe the behavior of **G840** during programming

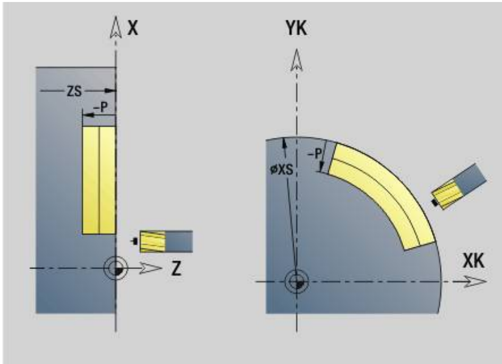
G840 – milling

You can change the machining direction and the cutter radius compensation (**TRC**) with the **cycle type Q**, the cutting direction **H** and the rotational direction of the tool. Program only the parameters given in the following table.



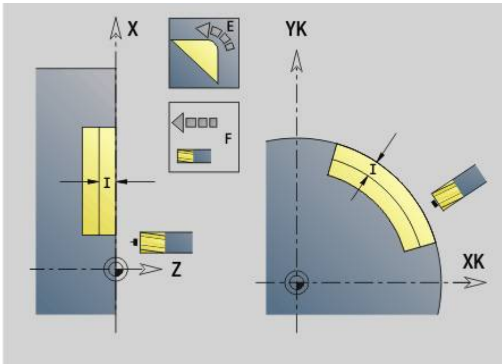
See also:

- **G840**—Fundamentals
  Further information: "G840 – fundamentals", Page 463
- **G840**—Calculating hole positions
  Further information: "G840 – calculating hole positions", Page 464

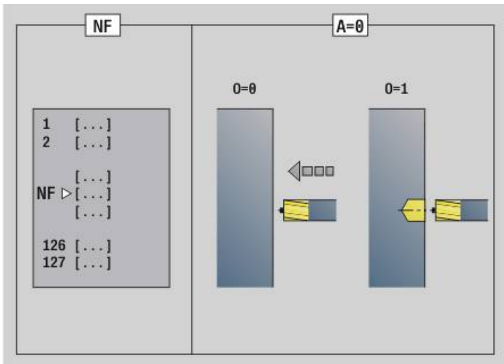


Parameters:

- **Q**: **Cycle type** – milling location
  - Open contour. If there is any overlapping, **Q** defines whether the first section (as of starting point) or the entire contour is to be machined
    - **Q** = 0: Center of milling cutter on the contour (hole position = starting point)
    - **Q** = 1: Machining at the left of the contour. If there is any overlapping, only the first area of the contour is machined
    - **Q** = 2: Machining at the right of the contour. If there is any overlapping, only the first area of the contour is machined
    - **Q** = 3: Not allowed
    - **Q** = 4: Machining at the left of the contour. If there is any overlapping, the entire contour is machined
    - **Q** = 5: Machining at the right of the contour. If there is any overlapping, the entire contour is machined
  - Closed contours
    - **Q** = 0: Center of milling cutter on the contour (hole position = starting point)
    - **Q** = 1: Inside milling
    - **Q** = 2: Outside milling
    - **Q** = 3 to 5: Not allowed



- **ID**: **Milling contour** – name of the milling contour
- **NS**: **Starting block no. of contour** – beginning of contour section
  - Figures: Block number of the figure
  - Free closed contour: First contour element (not starting point)
  - Open contour: First contour element (not starting point)
- **NE**: **Contour end block no.** – end of contour section
  - Figures, free closed contour: No input
  - Open contour: last contour element
  - Contour consists of one element:
    - No input: Machining in contour direction
    - **NS** = **NE** programmed: Machining against the contour direction



- **BF**: **Machine form elements** (default: 0)
  A chamfer/rounding arc is machined
  - **0: No machining**
  - **1: At beginning**
  - **2: At end**
  - **3: At beginning and end**
  - **4: Only chamfer/rounding** is machined—not the basic element (requirement: the contour section consists of a single element)
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **I**: **Max. approach**
- **F**: **Approach feed** for plunging (default: active feed rate)
- **E**: **Reduced feed** for circular elements (default: active feed rate)
- **R**: **Apprch angle** (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 with inside corners: Tool moves on approaching/departing arc that connects tangentially to the contour element
  - **R** < 0 with outside corners: Contour element is approached/departed tangentially on a linear path
- **P**: **Milling depth** (default: depth from the contour definition)
- **XS**: **Millg. top edge** lateral surface (replaces the reference plane from the contour definition)
- **ZS**: **Millg. top edge** face (replaces the reference plane from the contour definition)
- **RB**: **Return plane** (default: back to start position)
  - Front or rear face: Return position in Z direction
  - Lateral surface: Return position in X direction (diameter)

- **D**: **Start elem.no.**

  The direction of contour definition for figures is counterclockwise.

  The first contour element for figures:
  - Circular slot: The larger arc
  - Full circle: The upper semicircle
  - Rectangles, polygons and linear slots: The orientation angle points to the first contour element.
- **V**: **End: elem.no.**
- **A**: **(Mill=0/PredrillPos=1)**
- **NF**: **Position mark** – reference at which the cycle stores the predrill hole positions (range: 1 to 127)
- **O**: **Plunging behavior** (default: 0)
  - **O**=0: Vertical plunging
  - **O**=1: With predrilling
    - If **NF** is programmed: The cycle positions the milling cutter above the first hole position saved in **NF**, then plunges and mills the first section. If applicable, the cycle positions the tool to the next pre-drilled hole and mills the next section, etc.
    - If **NF** is not programmed: The tool plunges at the current position and mills the section. If required, repeat this operation for the next section, etc.

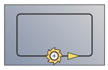
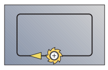
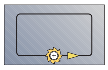
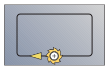
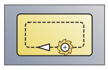
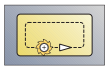
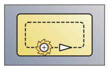
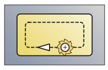
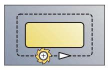
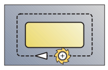
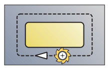
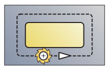
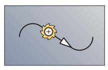
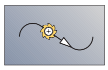
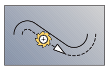
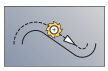
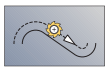
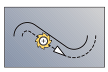
Approach and departure: For closed contours, the point of the surface normal from the tool position to the first contour element is the point of approach and departure. If no surface normal intersects the tool position, the starting point of the first element is the point of approach and departure. For figures, use **D** and **V** to select the approach/departure element.

Cycle run:

1. Starting position (**X**, **Z**, **C**) is the position before the cycle begins
2. Calculates the milling depth infeeds
3. Approaches to safety clearance:
   - If **O**=0: Infeed to the first milling depth
   - If **O**=1: Plunges to the first milling depth
4. Mills the contour
5. Infeeds:
   - For open contour and slots with slot width equal to the cutter diameter: Moves at feed rate to the next milling depth, or plunges at rapid to the next milling depth and mills the contour in reverse direction
   - For closed contours and slots: Lifts off by the safety clearance, moves forward and approaches to the next milling depth, or plunges to the next milling depth
6. Repeats steps 4 and 5 until the complete contour is milled
7. Returns to **Return plane RB**

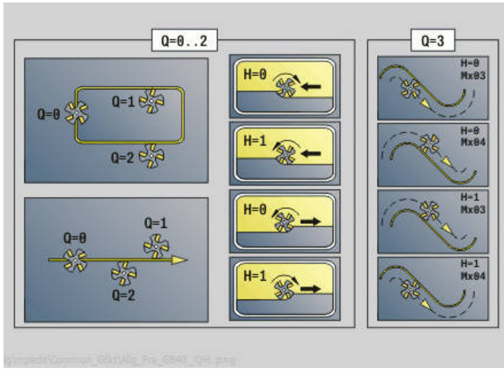
You can change the machining direction and the cutter radius compensation (**TRC**) with the cycle type **Q**, the cutting direction **H** and the rotational direction of the tool. Program only the parameters given in the following table.

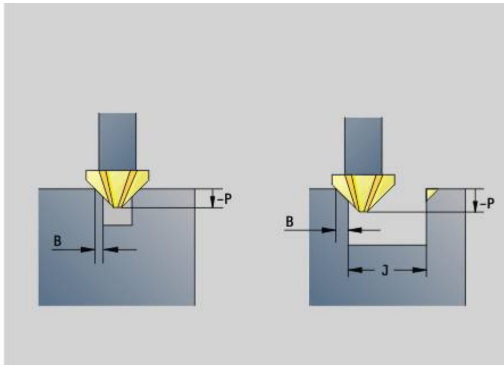
Contour milling G840

Cycle type	Cutting direction	Direction of tool rotation	MCRC	Execution
Contour (**Q**=0)	–	Mx03	–	
Contour	–	Mx03	–	
Contour	–	Mx04	–	
Contour	–	Mx04	–	
Inside (**Q**=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 (**Q**=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	
Contour (**Q**=0)	–	Mx03	–	
Contour	–	Mx04	–	
Right (**Q**=3)	Up-cut milling (**H**=0)	Mx03	Right	
Left (**Q**=3)	Up-cut milling (**H**=0)	Mx04	Left	
Left (**Q**=3)	Climb milling (**H**=1)	Mx03	Left	
Right (**Q**=3)	Climb milling (**H**=1)	Mx04	Right	

G840 – deburring

G840 performs a deburring operation if **Chamfer width B** is programmed. If there is any overlapping in the contour, use the **cycle type Q** to specify whether the first section (starting from the starting point) or the entire contour is to be machined. Program only the parameters given in the following table.





Parameters:

- **Q**: **Cycle type** – milling location
  - Open contour. If there is any overlapping, **Q** defines whether the first section (as of starting point) or the entire contour is to be machined
    - **Q** = 0: Center of milling cutter on the contour (hole position = starting point)
    - **Q** = 1: Machining at the left of the contour. If there is any overlapping, only the first area of the contour is machined
    - **Q** = 2: Machining at the right of the contour. If there is any overlapping, only the first area of the contour is machined
    - **Q** = 3: Not allowed
    - **Q** = 4: Machining at the left of the contour. If there is any overlapping, the entire contour is machined
    - **Q** = 5: Machining at the right of the contour. If there is any overlapping, the entire contour is machined
  - Closed contours
    - **Q** = 0: Center of milling cutter on the contour (hole position = starting point)
    - **Q** = 1: Inside milling
    - **Q** = 2: Outside milling
    - **Q** = 3 to 5: Not allowed
- **ID**: **Milling contour** – name of the milling contour
- **NS**: **Starting block no. of contour** – beginning of contour section
  - Figures: Block number of the figure
  - Free closed contour: First contour element (not starting point)
  - Open contour: First contour element (not starting point)
- **NE**: **Contour end block no.** – end of contour section
  - Figures, free closed contour: No input
  - Open contour: last contour element
  - Contour consists of one element:
    - No input: Machining in contour direction
    - **NS** = **NE** programmed: Machining against the contour direction
- **E**: **Reduced feed** for circular elements (default: active feed rate)

- **R**: **Apprch angle** (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 with inside corners: Tool moves on approaching/departing arc that connects tangentially to the contour element
  - **R** < 0 with outside corners: Contour element is approached/departed tangentially on a linear path
- **P**: **Plunging depth** (indicated as a negative value)
- **XS**: **Millg. top edge** lateral surface (replaces the reference plane from the contour definition)
- **ZS**: **Millg. top edge** face (replaces the reference plane from the contour definition)
- **RB**: **Return plane** (default: back to start position)
  - Front or rear face: Return position in Z direction
  - Lateral surface: Return position in X direction (diameter)
- **J**: **Premach. diam.**
  For open contours, the control calculates the contour to be deburred from the programmed contour and **J**.
  - **J** programmed: The cycle deburrs both sides of the slot
  - **J** not programmed: The deburring tool is so wide that both sides of the slot are deburred in one pass
- **D**: **Start elem.no.**
- **V**: **End: elem.no.**
- **A**: **(Mill=0/PredrillPos=1)**

Approach and departure: For closed contours, the point of the surface normal from the tool position to the first contour element is the point of approach and departure. If no surface normal intersects the tool position, the starting point of the first element is the point of approach and departure. For figures, use **D** and **V** to select the approach/departure element.

Cycle run:

1. Starting position (**X**, **Z**, **C**) is the position before the cycle begins
2. Moves to the safety clearance and infeed to the first milling depth
3. Milling:
   - **J** not programmed: Mills the programmed contour
   - **J** programmed, open contour: Calculates and mills the new contour
4. Returns to **Return plane RB**

Pocket milling - roughing G845

G845 – fundamentals

G845 roughs closed contours.

Choose one of the following plunge strategies, depending on the milling cutter you are using:

- Plunge vertically
- Plunge at a pre-drilled position
- Plunge in a reciprocating or helical motion

When plunging at a pre-drilled position, you have the following alternatives:

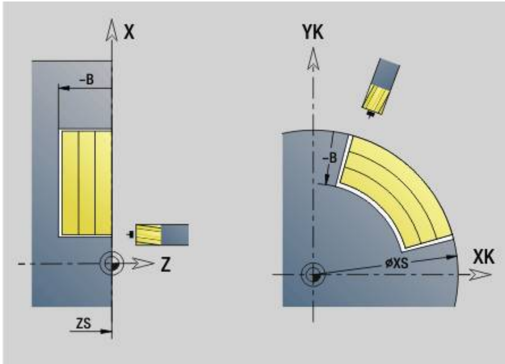
- Calculate positions, drill, mill—the machining process is performed in the following steps:
  - Insert drill.
  - Calculate hole positions with **G845 A1 ..** or set the hole position at the center of the figure with **A2**
  - Predrill with **G71 NF..**
  - Call cycle **G845 A0 ..**. The cycle positions the tool above the hole; the tool plunges and mills the pocket
- Drill, mill—the machining process is performed in the following steps:
  - Drill a hole inside the pocket with **G71 ..**
  - Position the milling cutter above the hole and call **G845 A0 ..**. The tool plunges and mills the section

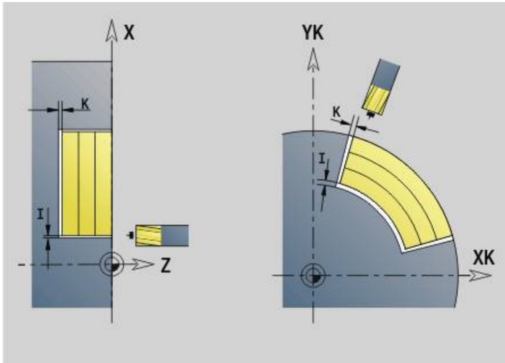
> The parameters **O**=1 and **NF** must be defined.

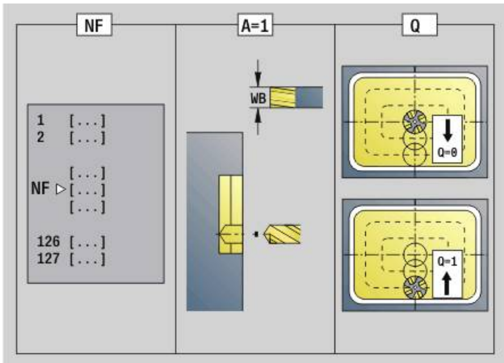
If the pocket consists of multiple sections, **G845** takes all the sections of the pocket into account for drilling and milling. Call **G845 A0 ..** separately for each section when calculating the hole positions without **G845 A1 ..**.

> **G845** takes the following oversizes into account:
> - **G57**: Oversize in X, Z direction
> - **G58**: Equidistant oversize in the milling plane
>
> Program oversizes for calculating the hole positions and for milling.

G845 – calculating hole positions







G845 A1 .. calculates the hole positions and stores them at the reference specified in **NF**. The cycle takes the diameter of the active tool into account when calculating the hole positions. Therefore, you need to insert the drill before calling **G845 A1..**. Program only the parameters given in the following table.

See also:

- **G845** – fundamentals
  Further information: "G845 – fundamentals", Page 472
- **G845** – milling
  Further information: "G845 – milling", Page 474

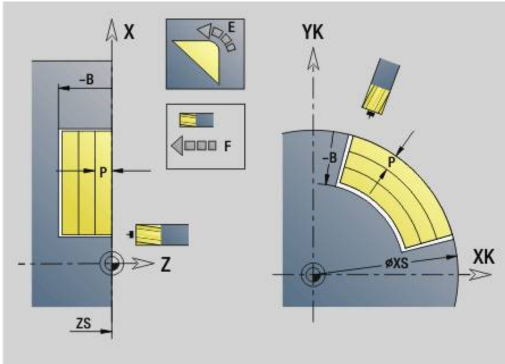
Parameters:

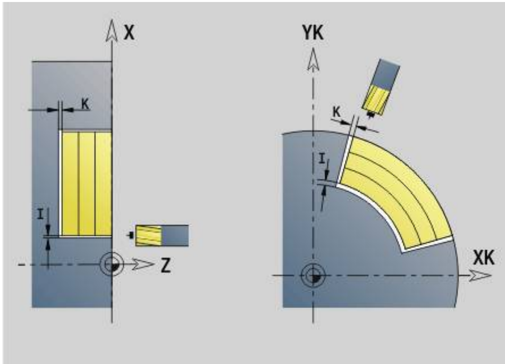
- **ID**: **Milling contour** – name of the milling contour
- **NS**: **Starting block no. of contour** – beginning of contour section
  - Figures: Block number of the figure
  - Free closed contour: First contour element (not starting point)
- **B**: **Milling depth** (default: depth from the contour description)
- **XS**: **Millg. top edge** lateral surface (replaces the reference plane from the contour definition)
- **ZS**: **Millg. top edge** face (replaces the reference plane from the contour definition)
- **I**: **O-size X**
- **K**: **O-size Z**
- **Q**: **Mach. direction** (default: 0)
  - **0: From the inside out**
  - **1: From the outside in**
- **A**: **Flow** (sequence)
  - **0: Milling**
  - **1: Determine predrilling pos**
  - **2: predrill. pos. fig. center**
- **NF**: **Position mark** – reference at which the cycle stores the predrill hole positions (range: 1 to 127)
- **WB**: **Plunging length** – diameter of the milling cutter

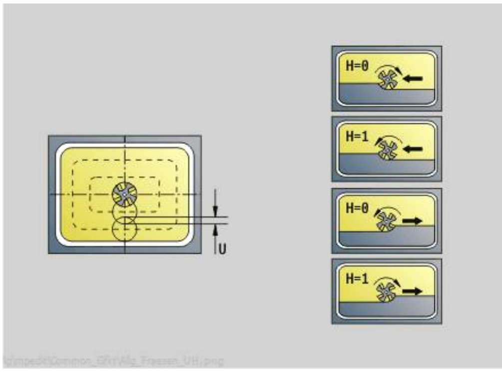
> - **G845** overwrites any hole positions that may still be stored at the reference **NF**
> - The parameter **WB** is used both for calculating the hole positions and for milling. When calculating the hole positions, **WB** describes the diameter of the milling cutter.

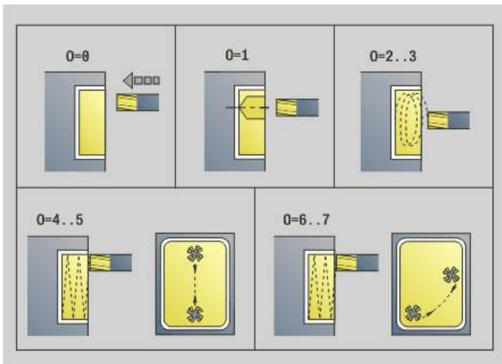
G845 – milling

You can change the cutting direction with the cutting direction **H**, the machining direction **Q** and the direction of tool rotation. Program only the parameters given in the following table.







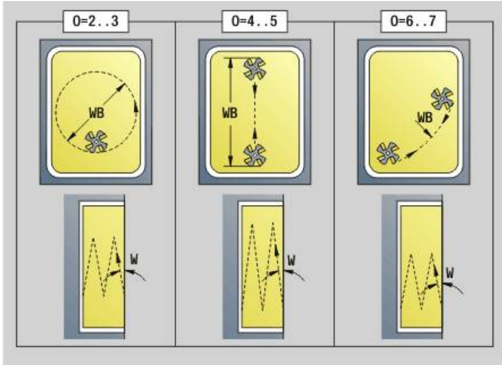


See also:

- **G845** – fundamentals
  Further information: "G845 – fundamentals", Page 472
- **G845** – calculating hole positions
  Further information: "G845 – calculating hole positions", Page 473

Parameters:

- **ID**: **Milling contour** – name of the milling contour
- **NS**: **Starting block no. of contour** – beginning of contour section
  - Figures: Block number of the figure
  - Free closed contour: First contour element (not starting point)
- **B**: **Milling depth** (default: depth from the contour description)
- **P**: **Max. approach** (default: milling in one infeed)
- **XS**: **Millg. top edge** lateral surface (replaces the reference plane from the contour definition)
- **ZS**: **Millg. top edge** face (replaces the reference plane from the contour definition)
- **I**: **O-size X**
- **K**: **O-size Z**
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
- **V**: **Overrun factor** (no effect in C-axis machining)
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **F**: **Approach feed** for plunging (default: active feed rate)
- **E**: **Reduced feed** for circular elements (default: active feed rate)
- **FP**: **Infeed rate in the plane** for the infeed to the next milling path
- **RB**: **Return plane** (default: back to start position)
  - Front or rear face: Return position in Z direction
  - Lateral surface: Return position in X direction (diameter)
- **Q**: **Mach. direction** (default: 0)
  - **0: From the inside out**
  - **1: From the outside in**
- **A**: **Flow** (sequence)
  - **0: Milling**
  - **1: Determine predrilling pos**
  - **2: predrill. pos. fig. center**
- **NF**: **Position mark** – reference at which the cycle stores the predrill hole positions (range: 1 to 127)
- **O**: **Plunging behavior** (default: 0)



- **O**=0 (vertical plunging): The cycle moves the tool to the starting point; the tool plunges at the feed rate for infeed and mills the pocket
- **O**=1 (plunge at pre-drilled position):
  - If **NF** is programmed: The cycle positions the milling cutter above the first pre-drilled hole; the tool plunges and mills the first area. If applicable, the cycle positions the tool to the next pre-drilled hole and mills the next area, etc.
  - If **NF** is not programmed: The cycle plunges at the current position and mills the area. If applicable, position the tool to the next pre-drilled hole and mill the next area, etc.
- **O**=2 or 3 (plunge in a helical motion): The tool plunges at the angle **W** and mills full circles with the diameter **WB**. As soon as it reaches the milling depth **P**, the cycle switches to face milling
  - **O**=2—manually: The cycle plunges at the current position and machines the area that can be reached from this position
  - **O**=3—automatically: The cycle calculates the plunging position, plunges and machines this area. The plunging motion ends at the starting point of the first milling path, if possible. If the pocket consists of multiple areas, the cycle successively machines all the areas
- **O**=4 or 5 (reciprocating linear plunge): The tool plunges at the angle **W** and mills a linear path of length **WB**. You can define the orientation angle in **WE**. The cycle then mills along this path in the opposite direction. As soon as it reaches the milling depth **P**, the cycle switches to face milling
  - **O**=4—manually: The cycle plunges at the current position and machines the area that can be reached from this position
  - **O**=5—automatically: The cycle calculates the plunging position, plunges and machines this area. The plunging motion ends at the starting point of the first milling path, if possible. If the pocket consists of multiple areas, the cycle successively machines all the areas. The plunging position is determined from the type of figure and from **Q** as follows:
    - **Q0** (from the inside toward the outside):
      - Linear slot, rectangle, polygon: Reference point of the figure
      - Circle: Circle center
      - Circular slot, free contour: Starting point of the innermost milling path
    - **Q1** (from the outside toward the inside):
      - Linear slot: Starting point of the slot
      - Circular slot, circle: Not machined
      - Rectangle, polygon: Starting point of the first linear element
      - Free contour: Starting point of the first linear element (at least one linear element must exist)

- **O**=6 or 7 (reciprocating circular plunge): The tool plunges at the plunging angle **W** and mills a circular arc of 90°. The cycle then mills along this path in the opposite direction. As soon as it reaches the milling depth **P**, the cycle switches to face milling. **WE** defines the arc center, **WB** the arc radius
  - **O**=6—manually: The tool position corresponds to the center of the circular arc. The tool moves to the arc starting point and plunges
  - **O**=7—automatically (only permitted for circular slots and circles): The cycle calculates the plunging position on the basis of **Q**:
    - **Q0** (from the inside toward the outside):
      - Circular slot: The circular arc lies on the curvature radius of the slot
      - Circle: Not permitted
    - **Q1** (from the outside toward the inside): Circular slot, circle: The circular arc lies on the outermost milling path

- **W**: **Plunging angle** in infeed direction
- **WE**: **Position angle** of the milling path/circular arc

  Reference axis:
  - Front or rear face: Positive XK axis
  - Lateral surface: Positive Z axis

  Default orientation angle, depending on **O**:
  - **O**=4: **WE**=0°
  - **O**=5 and
    - Linear slot, rectangle, polygon: **WE** = orientation angle of the figure
    - Circular slot, circle: **WE**=0°
    - Free contour and **Q0** (from the inside toward the outside): **WE**=0°
    - Free contour and **Q1** (from the outside toward the inside): Orientation angle of the starting element
- **WB**: **Diameter rework** (default: 1.5 * milling diameter)

> For the machining direction **Q**=1 (from the outside toward the inside), please note:
> - The contour must start with a linear element
> - If the starting element is < **WB**, **WB** is reduced to the length of the starting element
> - The length of the starting element must not be less than 1.5 times the diameter of the milling cutter

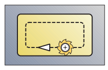
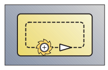
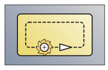
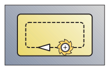
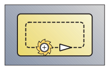
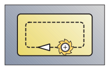
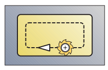
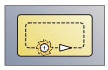
Cycle run:

1. Starting position (**X**, **Z**, **C**) is the position before the cycle begins
2. Calculates the proportioning of cuts (infeeds to the milling planes, infeeds in the milling depths) and the plunging positions and paths for reciprocating or helical plunges.
3. Approaches to safety clearance and, depending on **O**, feeds to the first milling depth or approaches helically or on a reciprocating path

4 Mills a plane
5 Retracts by the safety clearance, returns and cuts to the next milling depth.
6 Repeats steps 4 and 5 until the complete area is milled.
7 Returns to **Return plane RB**

You can change the cutting direction with the cutting direction **H**, the machining direction **Q** and the direction of tool rotation. Program only the parameters given in the following table.

Pocket milling - roughing G845

Cutting direction	Machining direction	Direction of tool rotation	Execution
Up-cut milling (**H**=0)	from inside (**Q**=0)	Mx03	
Up-cut milling (**H**=0)	from inside (**Q**=0)	Mx04	
Up-cut milling (**H**=0)	from outside (**Q**=1)	Mx03	
Up-cut milling (**H**=0)	from outside (**Q**=1)	Mx04	
Climb milling (**H**=1)	from inside (**Q**=0)	Mx03	
Climb milling (**H**=1)	from inside (**Q**=0)	Mx04	
Climb milling (**H**=1)	from outside (**Q**=1)	Mx03	
Climb milling (**H**=1)	from outside (**Q**=1)	Mx04	

Pocket milling - finishing G846

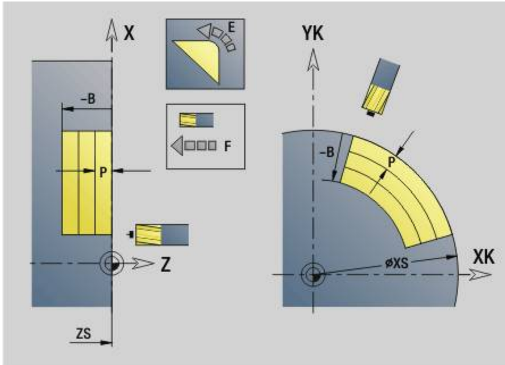
G846 finish-machines closed contours.

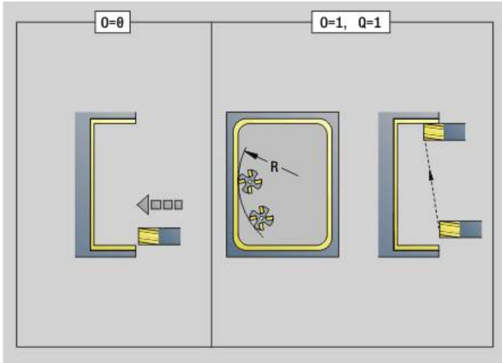
If the pocket consists of multiple sections, **G846** takes all the sections of the pocket into account.

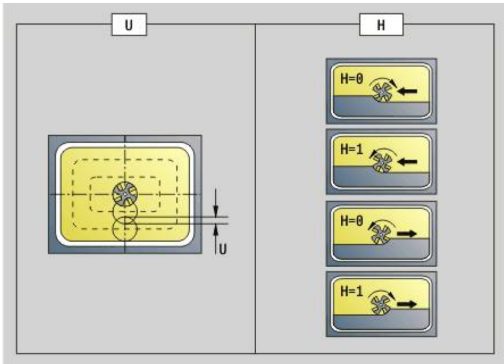
You can change the cutting direction with the cutting direction **H**, the machining direction **Q** and the direction of tool rotation.

Parameters:

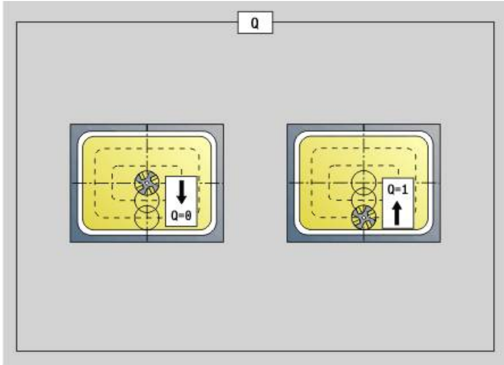
- **ID**: **Milling contour** – name of the milling contour
- **NS**: **Starting block no. of contour** – beginning of contour section
  - Figures: Block number of the figure
  - Free closed contour: First contour element (not starting point)
- **B**: **Milling depth** (default: depth from the contour description)
- **P**: **Max. approach** (default: milling in one infeed)
- **XS**: **Millg. top edge** lateral surface (replaces the reference plane from the contour definition)
- **ZS**: **Millg. top edge** face (replaces the reference plane from the contour definition)
- **R**: **Apprch angle** (default: 0)
  - **R**=0: Contour element is approached directly. Feed to the starting point above the milling plane, then vertical plunge
  - **R**>0: Tool moves on approaching/departing arc that connects tangentially to the contour element
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
- **V**: **Overrun factor** (no effect in C-axis machining)
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **F**: **Approach feed** for plunging (default: active feed rate)
- **E**: **Reduced feed** for circular elements (default: active feed rate)
- **FP**: **Infeed rate in the plane** for the infeed to the next milling path
- **RB**: **Return plane** (default: back to start position)
  - Front or rear face: Return position in Z direction
  - Lateral surface: Return position in X direction (diameter)
- **Q**: **Mach. direction** (default: 0)
  - **0: From the inside out**
  - **1: From the outside in**

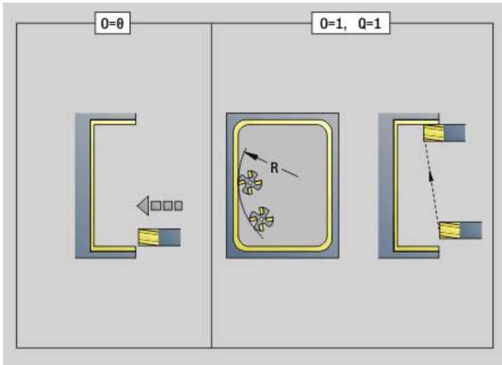






- **O**: **Plunging behavior** (default: 0)
  - **O**=0 (plunge vertically): The cycle moves the tool to the starting point; the tool plunges and finishes the pocket
  - **O**=1 (approaching arc with depth feed): When machining the upper milling planes, the tool advances to the milling plane and then approaches on an arc. When machining the bottom milling plane, the tool plunges to the milling depth while moving on the approaching arc (three-dimensional approaching arc). You can use this approach behavior only in conjunction with an approaching arc **R**. The precondition is machining from the outside toward the inside (**O**=1)



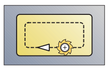
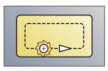
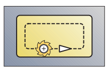
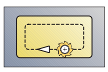


Cycle run:

1. Starting position (**X**, **Z**, **C**) is the position before the cycle begins
2. Calculates the number of cutting passes (infeeds to the milling planes, infeeds in the milling depths)
3. Move to the safety clearance and plunge to the first milling depth.
4. Mill the first plane.
5. Retract by the safety clearance, return and cut to the next milling depth.
6. Repeat steps 4 and 5 until the complete area is milled.
7. Returns to **Return plane RB**

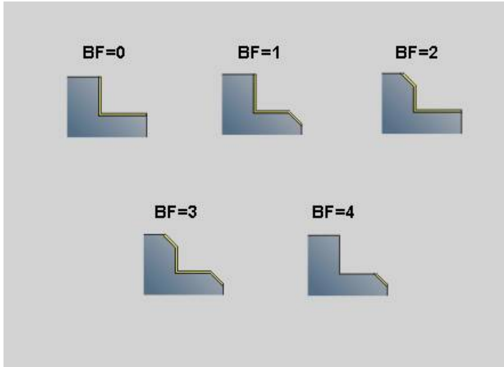
You can change the cutting direction with the cutting direction **H**, the machining direction **Q** and the direction of tool rotation.

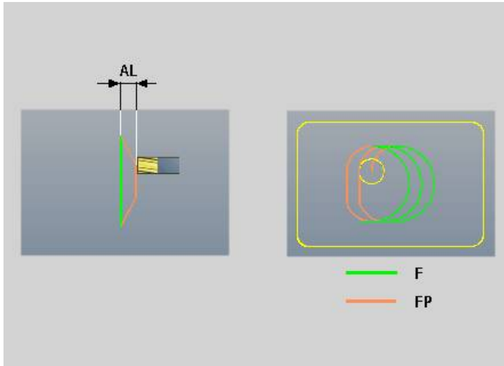
Pocket milling, finishing G846

Cutting direction	Direction of tool rotation	Execution
Up-cut milling (**H**=0)	Mx03	
Up-cut milling (**H**=0)	Mx04	
Climb milling (**H**=1)	Mx03	
Climb milling (**H**=1)	Mx04	

Trochoidal contour milling G847

G847 roughs out open or closed contours using trochoidal milling.





Parameters:

- **Q**: **Cycle type** (default: 0)
  - **0: On the contour**
  - **1: Within/left of contour**
  - **2: Outside/right of contour**
- **ID**: **Milling contour** – name of the milling contour
- **NS**: **Block number of contour** – reference to the contour description
- **NE**: **Contour end block no.** – end of contour section
- **BF**: **Machine form elements** (default: 0)

  A chamfer/rounding arc is machined
  - **0: No machining**
  - **1: At beginning**
  - **2: At end**
  - **3: At beginning and end**
  - **4: Only chamfer/rounding** is machined—not the basic element (requirement: the contour section consists of a single element)
- **H**: **Direction** (default: 1)
  - 0: Up-cut
  - 1: Climb
- **BR**: **Trochoid width**
- **R**: **Radius for return**
- **FP**: **Feed rate for return** (default: active feed rate)
- **AL**: **Retraction path for return**
- **U**: **Overlap factor** – overlap of milling paths = **U** * milling diameter (default: 0.9)
- **HC**: **Contour smoothing**
  - **0: No smoothing cut**
  - **1: With smoothing cut**
- **I**: **Max. approach**

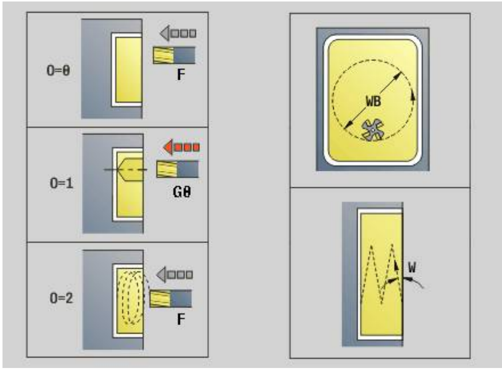
- **O**: **Plunging behavior** (default: 2)
  - **O** = 0 (vertical plunging): The cycle moves the tool to the starting point; the tool plunges at the feed rate for infeed and mills the contour
  - **O** = 1 (vertical plunging, e.g. at the predrilled position):
    - If **NF** is programmed: The control positions the milling cutter above the first predrill hole position; the tool plunges at rapid traverse to safety clearance and mills the first area. If applicable, the cycle positions the tool to the next predrill hole position and mills the next area etc.
    - If **NF** is not programmed: The cycle plunges at the current position at rapid traverse and mills the area. If applicable, position the tool to the next predrill hole position and mill the next area etc.
  - **O** = 2 (plunge in helical motion): The tool plunges at the current position at the angle **W** and mills full circles with the diameter **WB**.
- **F**: **Approach feed** (default: active feed)
- **W**: **Plunging angle**
- **WB**: **Diameter of the helix** (default: helix diameter = 1.5 * milling diameter)
- **RB**: **Return plane** (default: back to start position)
- **A**: **(Mill=0/PredrillPos=1)** (default: 0)
  - **0: Milling**
  - **1: Determine predrilling pos**
- **NF**: **Position mark** (only with **O** = 1)
- **P**: **Milling depth** (default: depth from the contour definition)
- **XS**: **Millg. top edge** lateral surface (replaces the reference plane from the contour definition)
- **ZS**: **Millg. top edge** face (replaces the reference plane from the contour definition)

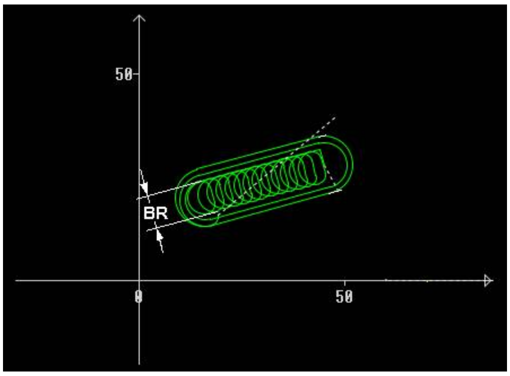
Trochoidal pocket milling G847

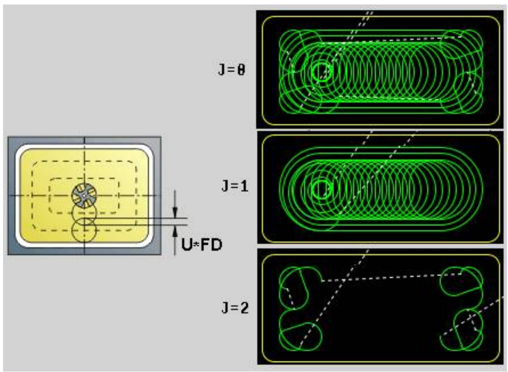
G848 roughs out figures or figure patterns using trochoidal milling.

Parameters:

- **ID**: **Milling contour** – name of the milling contour
- **NS**: **Block number of contour** – reference to the contour description
- **H**: **Direction** (default: 1)
  - 0: Up-cut
  - 1: Climb
- **BR**: **Trochoid width**
- **R**: **Radius for return**
- **FP**: **Feed rate for return** (default: active feed rate)
- **AL**: **Retraction path for return**
- **O**: **Plunging behavior** (default: 2)
  - **O**=0 (vertical plunging): The cycle moves the tool to the starting point; the tool plunges at the feed rate for infeed and mills the figure
  - **O** = 1 (vertical plunging, e.g. at the predrilled position):
    - If **NF** is programmed: The control positions the milling cutter above the first predrill hole position; the tool plunges at rapid traverse to safety clearance and mills the first area. If applicable, the cycle positions the tool to the next predrill hole position and mills the next area etc.
    - If **NF** is not programmed: The cycle plunges at the current position at rapid traverse and mills the area. If applicable, position the tool to the next predrill hole position and mill the next area etc.
  - **O** = 2 (plunge in helical motion): The tool plunges at the current position at the angle **W** and mills full circles with the diameter **WB**.
- **F**: **Approach feed** (default: active feed)
- **W**: **Plunging angle**
- **WB**: **Diameter of the helix** (default: helix diameter = 1.5 * milling diameter)
- **U**: **Overlap factor** – overlap of milling paths = **U** * milling diameter (default: 0.9)
- **J**: **Machining operation**
  - **0: Complete**
  - **1: W/o corner machining**
  - **2: Only corner machining**
- **P**: **Max. approach**
- **I**: **O-size X**
- **K**: **O-size Z**
- **RB**: **Return plane** (default: back to start position)
- **B**: **Milling depth** (default: depth from the contour definition)
- **XS**: **Millg. top edge** lateral surface (replaces the reference plane from the contour definition)
- **ZS**: **Millg. top edge** face (replaces the reference plane from the contour definition)
- **A**: **(Mill=0/PredrillPos=1)** (default: 0)







  - **0: Milling**
  - **1: Determine predrilling pos**
- **NF**: **Position mark** (only with **O** = 1)

> For slots and rectangles, you need to program width **BR** of the trochoidal tool path; for circles and polygons, this is not necessary.

6.27 Engraving cycles

Character sets

The control 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.

You can also use the engraving cycles to engrave string variables. In **ID**, enter the variable to be engraved using the **Variables** soft key.

Further information: "Variable types", Page 515

Small letters

NF	Character
97	a
98	b
99	c
100	d
101	e
102	f
103	g
104	h
105	i
106	j
107	k
108	l
109	m
110	n
111	o
112	p
113	q
114	r
115	s
116	t
117	u
118	v
119	w
120	x
121	y
122	z

Capital letters

NF	Character
65	A
66	B
67	C
68	D
69	E
70	F
71	G
72	H
73	I
74	J
75	K
76	L
77	M
78	N
79	O
80	P
81	Q
82	R
83	S
84	T
85	U
86	V
87	W
88	X
89	Y
90	Z

Umlauts

NF	Character
196	Ä
214	Ö
220	Ü
223	ß
228	ä
246	ö
7252	ü

Numbers

NF	Character
48	0
49	1
50	2
51	3
52	4
53	5
54	6
55	7
56	8
57	9

Special characters

NF	Character
32	Space
37	%
40	(
41	)
43	+
44	,
45	-
46	.
47	/
58	:
60	<
61	=
62	>
64	@
91	[
93	]
95	_
8364	€
181	µ
186	°
215	*
33	!
38	&
63	?
174	®
216	Ø

Engraving on front face G801

G801 engraves character strings in linear or polar layout on the front face.

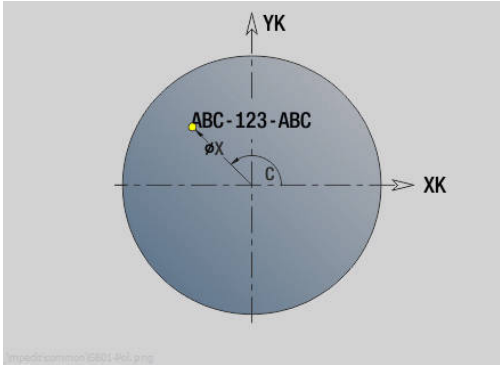
Further information: "Character sets", Page 484

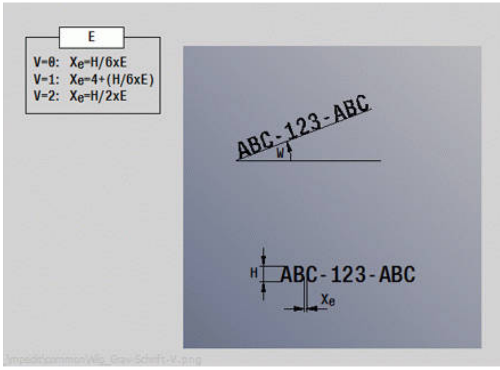
The cycles start engraving from the starting position or from the current position, if no starting position is defined.

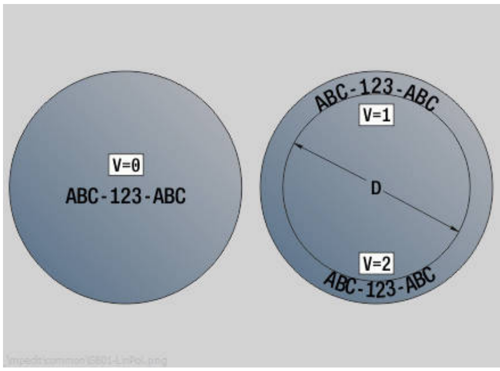
Example: If a character string is engraved with several calls, define the starting position in the first call. All other calls are programmed without a starting position.

Parameters:

- **X**, **C**: **Start point** and **Start. angle** (in polar coordinates)
- **XK**, **YK**: **Start point** (in Cartesian coordinates)
- **Z**: **Final point** – Z position, infeed depth during milling
- **RB**: **Return plane** – Z position to which the tool retracts for positioning
- **ID**: **Text** to be engraved
- **NF**: **Char. no.** – ASCII code of the character to be engraved
- **NS**: **Block number of contour** – reference to the contour description
- **W**: **Inclinat. ang.** of the character string
- **H**: **Font height**
- **E**: **Distance factor**
  The spacing between the characters is calculated according to the following formula: **H** / 6 * **E**
- **V**: **Execution (linear/polar)**
  - **0: Linear**
  - **1: Arched above**
  - **2: Arched below**
- **D**: **Reference diameter**
- **F**: **Plunging feed rate factor** (plunging feed rate = current feed rate * **F**)
- **O**: **Mirror writing**
  - **0** (**No**): Engraving is not mirrored
  - **1** (**Yes**): Engraving is mirrored (mirror writing)







Engraving on lateral surface G802

G802 engraves character strings aligned linearly on the lateral surface.

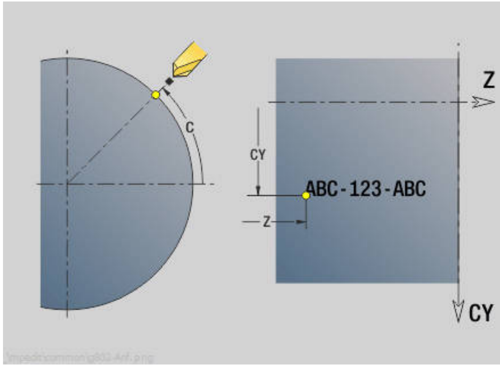
Further information: "Character sets", Page 484

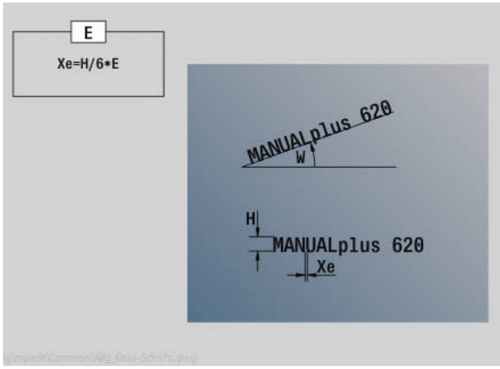
The cycles start engraving from the starting position or from the current position, if no starting position is defined.

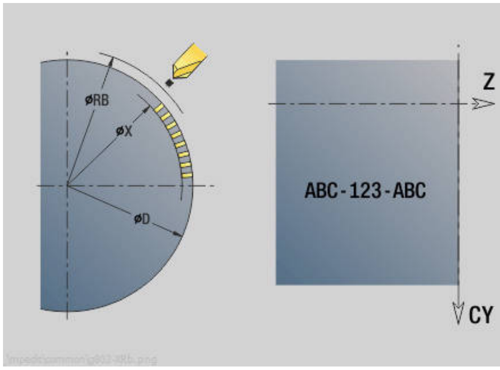
Example: If a character string is engraved with several calls, define the starting position in the first call. All other calls are programmed without a starting position.

Parameters:

- **Z**: **Start point**
- **C**: **Start. angle**
- **CY**: **Start point** of first character
- **X**: **Final point** – X position, infeed depth during milling (diameter value)
- **RB**: **Return plane** – X position to which the tool retracts for positioning
- **ID**: **Text** to be engraved
- **NF**: **Char. no.** – ASCII code of the character to be engraved
- **NS**: **Block number of contour** – reference to the contour description
- **W**: **Inclinat. ang.** of the character string
- **H**: **Font height**
- **V**: **Overrun factor** (no effect in C-axis machining)
- **H**: **Mill cutting direction**
- **E**: **Distance factor**
  The spacing between the characters is calculated according to the following formula: **H** / 6 * **E**
- **D**: **Reference diameter**
- **F**: **Plunging feed rate factor** (plunging feed rate = current feed rate * **F**)
- **O**: **Mirror writing**
- **O**: **Mirror writing**
  - **0** (**No**): Engraving is not mirrored
  - **1** (**Yes**): Engraving is mirrored (mirror writing)







6.28 Contour follow-up

Automatic Contour follow-up is not possible with program branches or repetitions. In these cases, you can control Contour follow-up with the following commands.

Saving/loading Contour follow-up G702

G702 saves the current contour or loads a saved contour.

Parameters:

- **ID**: **Workpc. blank contour** – name of the auxiliary workpiece blank
- **Q**: **0=save 1=load 2=internal**
  - 0: Saves the current contour. The contour follow-up is not affected
  - 1: Loads the specified contour. The contour follow-up is continued with the loaded contour
  - 2: The following cycle uses the internal workpiece blank
- **H**: **Memory number** (range: 0 to 9)
- **V**: **0=all, 1=variable, 2=blank** – selection of information to be saved
  - 0: Everything (variable contents and workpiece blank contours)
  - 1: Variable contents
  - 2: Workpiece blank contours

G702 Q2 switches off global Contour follow-up for the following cycle. Once the cycle has been executed, global Contour follow-up is effective again.

The affected cycle uses the internal Workpiece blank. The cycle determines the internal workpiece blank from the contour and the tool position.

G702 Q2 must be programmed before the cycle.

Contour follow-up on/off G703

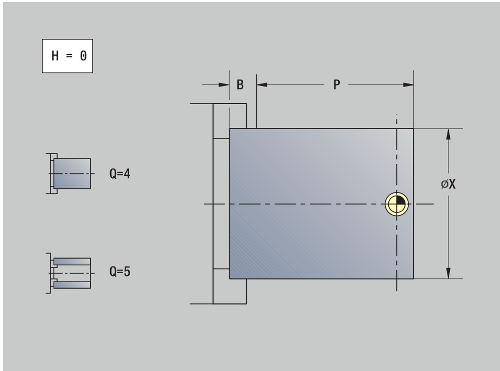
G703 is used to activate/deactivate Contour follow-up.

Parameters:

- **Q**: **On=1 Off=0** – contour follow-up on/off
  - 0: Off
  - 1: On

6.29 Other G codes

Clamping G65



G65 displays the selected chucking equipment in the simulation graphics.

Parameters:

- **H**: **No. of clamping** – always **H** = 0
- **D**: **Fixture** – no input
- **X**: **Start point** – diameter of workpiece blank
- **Z**: **Start point** (default: no input)
- **Q**: **Chuck form**
  - **5: Outside chucking**
  - **5: Inside chucking**
- **B**: **Clamping length** (**B** + **P** = length of blank)
- **P**: **Free length**
- **V**: **Delete chucking equipment**

Blank contour G67 (for graphics)

G67 displays an **Aux. workpiece blank** in **Simulation** submode.

Parameters:

- **ID**: **Workpc. blank contour** – name of the auxiliary workpiece blank
- **NS**: **Starting block no. of contour** – beginning of contour section

Period of dwell G4

With **G4**, the control interrupts the program run for the **Period of dwell F** or until the revolutions at the recess base **D** have been completed before executing the next NC block. If **G4** is programmed together with a traverse path in the same block, the **Period of dwell** or **No. of spindle revolutions** at the recess base only becomes effective after the traverse path has been executed.

Parameters:

- **F**: **Delay** in seconds (range: 0 < **F** <= 999)
- **D**: **Rev. on recessing floor**

Precision stop ON G7

G7 switches **Precision stop** on. It is a modal function. With a **Precision stop**, the control only starts running the subsequent block after the last point has been reached in the tolerance window for the position. The tolerance window is defined in the **posTolerance** machine parameter (no. 401101). **Precision stop** affects both single paths and cycles. The NC block containing **G7** is also executed with a precision stop.

Precision stop OFF G8

G8 switches **Precision stop** off. The block containing **G8** is executed without a **Precision stop**.

Precision stop blockwise G9

G9 activates a **Precision stop** for the NC block in which it is programmed. With a **Precision stop**, the control only starts running the subsequent block after the last point has been reached in the tolerance window for the position. The tolerance window is defined in the **posTolerance** machine parameter (no. 401101).

Switch off protection zone G60

G60 is used to cancel protection zone monitoring. **G60** is programmed before the traversing command to be monitored or not monitored.

Parameters:

- **Q**: activate/deactivate – **Self-locking=1**
  - 0: Activate protection zone (modal)
  - 1: Deactivate protection zone (modal)

Application example: With **G60**, you can temporarily deactivate a programmed monitoring of the protection zone in order to machine a centric through hole.

Example: G60

. . .	
N1 T4 G97 S1000 G95 F0.3 M3	
N2 G0 X0 Z5	
N3 G60 Q1	Deactivate protection zone
N4 G71 Z-60 K65	
N5 G60 Q0	Activate protection zone
. . .	

Actual values in variable G901

G901 transfers the actual values of all the axes of a slide into the variables for the interpolation information.

Further information: "Fill variable memory G904", Page 492

Datum in variable G902

G902 transfers the datum shifts into the variables for the interpolation information.

Further information: "Fill variable memory G904", Page 492

Lag error in variable G903

G903 transfers the current following error (distance by which the actual value lags the nominal value) into the variables for the interpolation information.

Further information: "Fill variable memory G904", Page 492

Fill variable memory G904

G904 transfers all the current interpolation information on the current slide to the variable memory.

Interpolation information

#a0(Z,1)	Datum shift of the Z axis of slide $1
#a1(Z,1)	Actual position of the Z axis of slide $1
#a2(Z,1)	Nominal position of the Z axis of slide $1
#a3(Z,1)	Lag error of the Z axis of slide $1
#a4(Z,1)	Distance to go in the Z axis of slide $1
#a5(Z,1)	Logical axis number of the Z axis of slide $1
#a5(0,1)	Logical axis number of the main spindle
#a6(0,1)	Rotational direction of main spindle $1
#a9(Z,1)	Trigger position of the touch probe **#a10(Z,1)** IPO axis value

Interpolation information syntax

Syntax: **#an(axis,channel)**

- **n** = number of the information
- **axis** = name of the axis
- **channel** = slide number

Feed rate override 100 % G908

G908 sets the feed override for traverse paths (**G0**, **G1**, **G2**, **G3**, **G12**, **G13**) to 100 %.

Program **G908** and the traverse path in the same NC block.

Parameters:

- **H**: **Limiting type** (default: 0)
  - 0: Activate blockwise feed rate override
  - 1: Activate modal feed rate override—if the feed-rate potentiometer is set to 0, the axes are stopped
  - 2: Deactivate feed rate override

Interpreter stop G909

The control pre-interprets the NC blocks. If variables are assigned shortly before the evaluation, old values are processed. **G909** stops the pre-interpretation. The NC blocks are processed up to **G909**. Only after G909, are the subsequent NC blocks processed.

Apart from **G909**, the NC block should only contain synchronous functions. (Some **G** codes generate an interpreter stop).

Spindle override 100% G919

G919 is used to deactivate/activate the spindle speed override.

Parameters:

- **Q**: **No. of spindle** (default: 0)
- **H**: **Limiting type** (default: 0)
  - 0: Activate spindle speed override
  - 1: Spindle override at 100 %—modal
  - 2: Spindle override at 100 %—for the current NC block

Deactivating datum shifts G920

G920 deactivates the workpiece datum and datum shifts. Traverse paths and position values are referenced to the distance tool tip – machine datum.

Deactivating datum shift, tool lengths G921

G921 deactivates the workpiece datum, datum shifts and tool dimensions. Traverse paths and position values are referenced to the slide reference point—machine datum.

Fluctuating spindle speed G924

When resonances are to be reduced, you can use **G924** to program a changing spindle speed. In **G924**, define the **Repetition rate** and the range for the **Change of spindle speed**. The **G924** function is automatically reset at the end of the program. You can also deactivate the function through another call with the setting **H0** (off).

Parameters:

- **Q**: **No. of spindle** (default: 0)
- **K**: **Repetition rate** – time interval in hertz (repetitions per second)
- **I**: **Change of spindle speed**
- **H**: Function **G924 On=1 Off=0**
  - 0: Off
  - 1: On

Convert lengths G927

Function **G927** is used to convert the tool lengths at the current tool insert angle to the initial position of the tool (reference position in B axis = 0).

The results can be interrogated in the variables **#n927(X)**, **#n927(Z)**, and **#n927(Y)**.

Parameters:

- **H**: **Type of calculation**
  - 0: Convert tool length to reference position (take **I** + **K** of the tool into account)
  - 1: Convert tool length to reference position (do not take **I** + **K** of the tool into account)
  - 2: Convert tool length from the reference position to the current work position (take **I** + **K** of the tool into account)
  - 3: Convert tool length from the reference position to the current work position (do not take **I** + **K** of the tool into account)
- **X**, **Y**, **Z**: Axis values (X value = radius; If nothing is entered, the value 0 is used)

TCPM G928

With the **TCPM G928** function, you can change the behavior of the rotary axes for tilting. With **TCPM** disabled, the axis will rotate about its mechanical center of rotation; with **TCPM** enabled, the tool tip will remain at the center of rotation and the linear axes perform a compensation movement.

With the **D** parameter, you can specify how the position of the virtual tool tip will be converted before the control calculates the TCPM compensation movement.

The parameter **Q** enables you to exclude individual rotary axes from **TCPM**.

Parameters:

- **H: Activate TCPM**
  - 0: Off
  - 1: On
- **E**: **Fmax in compensating motion** – speed limit during the compensation movement of the linear axes
- **D**: **Flow**
  - **0: Tool-center path**
  - **1: Tool tip path**
- **Q: TCPM with/without** (default: 0)
  - 0: All axes
  - 1: Without A axis
  - 2: Without B axis
  - 3: Without C axis

Look-ahead parameter G932

Refer to your machine manual.

This function must be set up by your machine manufacturer.

The **G932** G code allows you to influence the machining speed, the accuracy, and the surface definition.

The control tries to move to all contour points at the programmed contouring feed rate while adhering to the path tolerance defined in the control. The control reduces the feed rate where needed so that the defined tolerance is complied with.

The **G932** G code allows you to modify the positioning behavior of the control, in order to enable higher contouring feed rates for example.

Parameters:

- **H**: **HSC mode** – machining filters predefined by the machine tool builder
  - **0: standard**

    The control uses the default filter setting that is designed for universal machining.
  - **1: roughing**

    The control uses the roughing filter setting that enables a higher feed rate.
  - **2: finishing**

    The control uses the finishing filter setting that enables a higher contour accuracy.
- **R**: **Tolerance for linear axes** – permissible contour deviation for linear axes (e.g., X axis)
- **W**: **Tolerance for rotary axes** – permissible position deviation for rotary axes (e.g., C axis with **G928 (TCPM)** active)

The entered tolerance values are effective for both turning and milling.

Calculating variables automatically G940

Use **G940** to convert metric values to inch values. When you create a new program you can select between metric units and inches. Internally the control always calculates with metric values. If you read out variables in an "inch" program, the variables are always output as metric values. Use **G940** to convert the variables to inch values.

Parameters:

- **H**: Function **G940 On=1 Off=0**
  - 0: Unit conversion active
  - 1: Values remain metric

In inch programs, a conversion is required for variables that refer to a metric unit of measurement.

Machine dimensions

#m1(n)	Machine dimensions of an axis, e.g. **#m1(X)** for machine dimensions of the X axis

Reading tool data

#wn(NL)	Usable length (inside turning tools and drilling tools)
#wn(RS)	Cutting edge radius
#wn(ZD)	Stud diameter
#wn(DF)	Milling diameter
#wn(SD)	Shank diameter
#wn(SB)	Cutting width
#wn(AL)	Cut-in length
#wn(FB)	Cutter width
#wn(ZL)	Setup dim. in Z
#wn(XL)	Setup dim. in X
#wn(YL)	Setup dim. in Y
#wn(I)	Position of tool tip center in X
#wn(K)	Position of tool tip center in Z
#wn(ZE)	Distance between tool tip and slide reference point Z
#wn(XE)	Distance between tool tip and slide reference point X
#wn(YE)	Distance between tool tip and slide reference point Y

Reading the current NC information

#n0(Z)	Last programmed position Z
#n120(X)	Reference diameter X for calculating CY
#n57(X)	Oversize in X
#n57(Z)	Oversize in Z
#n58(P)	Equidistant oversize
#n150(X)	Cutting width shifted in X by **G150**
#n95(F)	Last programmed feed rate
#n47(P)	Current safety clearance
#n147(I)	Current safety clearance in working plane
#n147(K)	Current safety clearance in infeed direction

Internal information for defining constants

__n0_x	768 Last programmed position X
__n0_y	769 Last programmed position Y
__n0_z	770 Last programmed position Z
__n120_x	787 Reference diameter X for calculating CY
__n57_x	791 Oversize in X
__n57_z	792 Oversize in Z
__n58_p	793 Equidistant oversize
__n150_x	794 Cutting width shifted in X by **G150**/**G151**
__n150_z	795 Cutting width shifted in Z by **G150**/**G151**
__n95_f	800 Last programmed feed rate

Fill variable memory G904

#a0(Z,1)	Datum shift of the Z axis of slide $1
#a1(Z,1)	Actual position of the Z axis of slide $1
#a2(Z,1)	Nominal position of the Z axis of slide $1
#a3(Z,1)	Lag error of the Z axis of slide $1
#a4(Z,1)	Distance to go in the Z axis of slide $1

Information to DNC G941

G941 enables you to send your own messages from the NC program via the HEIDENHAIN DNC interface.

The messages sent are evaluated by appropriate PC applications, such as StateMonitor.

Parameters:

- **ID**: **Output text** – text and optional definition of format of output values (max. 80 characters)

  Examples of output format:
  - **%f** – output of a floating point number in original format (contents of parameter **R**)
  - **%.0f** – output of a floating point number without decimal places
  - **%.1f** – output of a floating point number with one decimal place
  - **%+.2f** – output of a floating point number with algebraic sign and two decimal places
- **R**: **Output value** – value or variable

  Examples of output values:
  - Value, (e.g. **3.15**)
  - Variable, (e.g. **#l1**)

Example: G941

N 46 #l1=#l1+1	Part counter
N47 G941 ID"STUECKZAHL" R#l1	Send message

Misalignment compensation G976

With the **Misalignment compensation G976** function you can run the following operations on tapering contours (e.g., to counter a mechanical offset). The **G976** function is automatically reset at the end of the program. You can also deactivate the function through another call with the setting **H0** (off).

Parameters:

- **Z**: **Start point**
- **K**: **Length**
- **I**: **Incremental distance**
- **J**: **Incremental distance**
- **H**: Function **G976 On=1 Off=0**
  - 0: Off
  - 1: On

Lift off after NC stop – LIFTOFF G977

> **G977** is only effective if machine parameter **CfgLiftOff** (201401) is activated.

G977 enables you to define the lift-off movement after an NC stop with respect to the tool and cutting pass.

> **G977** cannot be used in connection with thread cycles. Machine parameter **threadLiftOff** (601804) is available for thread cycles.

Parameters:

- **H**: **On/Off**
  - 0: deactivate
  - 1: activate
- **A**: **Depart.angle** – angle with the positive Z axis (no input: lift-off angle corresponds to the bisecting line of the tool's cutting edge for turning tools; for boring or milling tools, it corresponds to the position of the tool axis)
- **W**: **Spatial angle** – angle to positive X axis
- **R**: **Length** – lift-off length (no input: value from machine parameter **distance** (201402))

After a tool change, the control will reset the **A** and **W** parameters corresponding to the tool geometry.

Tilting the B axis will change the lift-off direction by the angular difference in B.

> If you load a drilling or milling tool, the control automatically deactivates **G977** because it cannot unambiguously determine the lift-off direction.
>
> ▶ Program **G977** again if you intend to use Lift-off with drilling or milling tools.

> Operating notes:
>
> - If no value is defined in machine parameter **distance** (201402), the control uses a lift-off distance of 1 mm
> - Correctly positioned recessing tools lift off paraxially
> - **RW** tilt angles programmed for drilling and milling tools will not be considered

Example: G977

N 46 G977 H1 A30	Departure angle 30°
...	
N 55 T1	Bisecting line as departure angle
...	
N 69 G977 H1 A30	Departure angle again 30°

Activating datum shift G980

G980 activates the workpiece datum and all datum shifts. Traverse paths and position values are referenced to the distance of the tool tip to the workpiece datum, while taking the datum shifts into consideration.

Activating datum shifts, tool lengths G981

G981 activates the workpiece datum, all datum shifts and the tool dimensions. Traverse paths and position values are referenced to the distance of the tool tip to the workpiece datum, while taking the datum shifts into consideration.

Monitoring zone G995

G995 defines the **monitoring zone** and the axes to be monitored. The **monitoring zone** corresponds to the program section that is to be monitored by the control.

To begin the **monitoring zone**, program **G995** with the following parameters. To end the **monitoring zone**, program **G995** without parameters.

Parameters:

- **H**: **No. of zone** (range: 1 to 99)
- **ID**: **Axis code**
  - X: X axis
  - Y: Y axis
  - Z: Z axis
  - 0: Spindle 1 (main spindle, C axis)
  - 1: Spindle 2
  - 2: Spindle 3

> The monitoring zones must be unambiguously defined in the program. Use the **H** parameter to assign a unique number to each monitoring zone.

> If you would like to monitor more than one drive within a monitoring zone, enter the respective combination of individual parameters in the **ID** parameter. Please keep in mind, however, that the control can monitor a maximum of four drives per monitoring zone. To simultaneously monitor the Z axis and the main spindle, enter **Z0** in the **ID** parameter.

> In addition to defining the monitoring zone with **G995**, you need to activate the load monitoring function.
>
> **Further information:** "Load monitoring G996", Page 501

Example: G995

...	
N1 T4	
N2 G995 H1 ID"X0"	Beginning of monitoring zone; monitoring of X axis and main spindle
...	Machining
N9 G995	End of monitoring zone
...	

Load monitoring G996

G996 defines the type of **load monitoring** or deactivates the load monitoring temporarily.

Parameters:

- **Q**: **Lib. switch** – scope of load monitoring (default: 0)
  - 0: Off
  - 1: **G0** Off (do not monitor rapid traverse movements)
  - 2: **G0** On (monitor rapid traverse movements)
- **H**: **Monitoring 0-2** – type of load monitoring (default: 0)
  - 0: Utilization and total utilization
  - 1: Utilization only
  - 2: Total utilization only

> In addition to defining the type of load monitoring with **G996**, you need to specify the monitoring zone with **G995**.
> **Further information:** "Monitoring zone G995", Page 500

> To be able to use the load monitoring you must also define limit values and perform reference machining.

Further information: User's Manual

Example: G996

...	
N1 G996 Q1 H1	Switch on load monitoring; do not monitor rapid traverse movements
N2 T4	
N3 G995 H1 ID"X0"	
...	Machining
N9 G995	
...	

Activating direct program-run continuation G999

With the **G999** function, when running a program in Single Block mode, the following NC blocks are run with a single NC start to the end of the program. **G999** is then deactivated by again calling the function with the setting **Q0** (off).

Force reduction G925

> Refer to your machine manual.
> The machine tool builder determines the scope of function and behavior of this function.

G925 activates/deactivates the force reduction function. When monitoring is activated, the maximum **Contact force** for one axis is defined. Force reduction can be activated for only one axis per NC channel.

The **G925** function limits the **Contact force** for subsequent movements of the defined axis. **G925** does not execute any traverse movements.

Parameters:

- **H**: **Contact force** in daN – the contact force is limited to the value entered
- **Q**: **Axis number** (X = 1, Y = 2, Z = 3, U = 4, V = 5, W = 6, A = 7, B = 8, C = 9) **No. of spindle**, e.g. spindle 0 = number 10 (0 = 10, 1 = 11, 2 = 12, 3 = 13, 4 = 14, 5 = 15)
- **P**: **Sleeve monitoring on/off**
  - 0: Deactivate (contact force is not monitored)
  - 1: Activate (contact force is monitored)

> Lag error monitoring is not activated until the acceleration phase has been completed.

Sleeve monitoring G930

> Refer to your machine manual.
> The machine tool builder determines the scope of function and behavior of this function.

G930 activates/deactivates the **Sleeve monitoring** function. When monitoring is activated, the maximum **Contact force** for one axis is defined. **Sleeve monitoring** can be activated for only one axis per NC channel.

G930 moves the defined axis by the **Incremental distance K** until the defined **Contact force H** has been reached.

Parameters:

- **H**: **Contact force** in daN – the contact force is limited to the value entered
- **Q**: **Axis number** (X = 1, Y = 2, Z = 3, U = 4, V = 5, W = 6, A = 7, B = 8, C = 9)
- **K**: **Incremental distance**

Application example: G930 is applied to use the counter spindle as a mechatronic tailstock. In this case the counter spindle is equipped with a dead center and the **Contact force** is limited with **G930**. A prerequisite for this application is a PLC program from the machine tool builder that enables the user to operate the mechatronic tailstock in the Manual and Automatic operating modes.

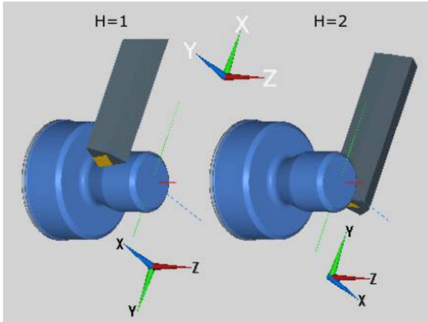
> Lag error monitoring is not activated until the acceleration phase has been completed.

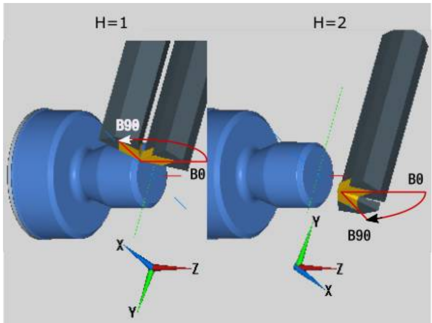
Tailstock function: With the tailstock function, the control approaches the workpiece and stops as soon as the **Contact force** has been reached. The remaining path of traverse is deleted.

Example: Tailstock function

. . .	
N.. G0 Z20	Pre-position slide 2
N.. G930 H250 D6 K-20	Activate the tailstock function—contact force 250 daN
. . .	

HDT mode G931





G931 activates or deactivates **HDT** mode. Within the function you must select whether machining takes place in front of or behind the workpiece. Optionally, you can also define an inclination of the cutting edge.

Further information: "High Dynamic Turning", Page 724

Parameters:

- **H**: **HDT mode**
  - **H** = **0: Deactivate**
  - **H** = **1: Before the workpiece**
  - **H** = **2: Behind the workpiece**
- **B**: **Angle**

Notes:

- In **HDT** mode, the B axis must be tilted to B = 90°. When **HDT** mode is active, the programmed X movements are performed by the Y axis. Consequently, Y movements are performed by the X axis. The B axis cannot be moved while **HDT** mode is active.
- In **HDT** mode, an **HDT** tool must be used.

  see User's Manual
- **HDT** tools are held by a milling spindle that is operated as a B axis (**B2**). If you do not define the **Angle B**, the control positions the **B2** axis with the cutting edge of the tool at 90°.
- While **HDT** mode is active, you can switch between the individual tool teeth of the multipoint tool. Changing to a different tool, for example from the tool magazine, is not possible.
- Depending on whether "in front of" or "behind" the workpiece was selected, you have to program the appropriate direction of rotation for the tool.
- Always program contours with circular arcs corresponding to machining behind the workpiece.
- Always program **G41** and **G42** corresponding to machining behind the workpiece.
- Positions that you program in connection with **G701** or **G14** will be interpreted by the control as axis values. The control executes these movements in the non-tilted machine coordinate system.
- Additive compensations with **G149** must be programmed after you have activated **HDT** mode. If you wish to compensate the diameter, you have to enter the desired diameter difference as an X value. A negative X value leads to a reduced diameter. A positive X value leads to an enlarged diameter.
- You cannot enter tool compensations during machining.
- Activation and deactivation of datums or tool length compensations with **G980/G981** or **G920/G921** are not permitted in combination with **HDT** mode.
- C axis machining operations are not permitted in combination with **HDT** mode.
- You can query the **HDT** mode status with the **#n931(H)** variable during machining.
- The machine data display notifies you about the current activation status of **HDT mode**.

  see User's Manual

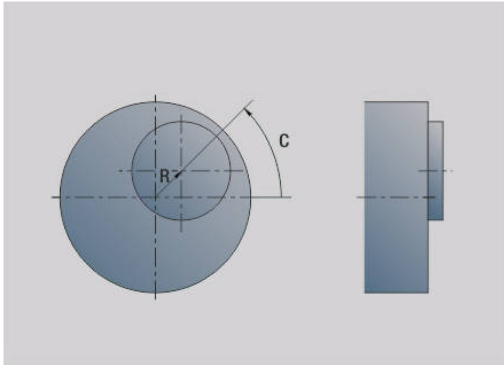
- If you cancel program run while **HDT** mode is active, the current **HDT** status remains effective. Consider this circumstance, for example when using **MDI** cycles. While **HDT** mode is active, you also cannot measure tools.

> Refer to your machine manual.
>
> The machine tool builder can provide functions for deactivating **HDT** mode in **Machine** mode.

Eccentric turning G725

G725 is used to machine turning contours outside the original turning center.

The turning contours are programmed using separate turning cycles.



Refer to your machine manual.
This function is set by your machine tool builder.
Prerequisites:
- Y-axis machining (option 70)
- Synchronizing functions (option 135)

Parameters:
- **H**: **Activate coupling**
  - **H** = 0: Deactivate coupling
  - **H** = 1: Activate coupling
- **Q**: **Reference spindle** – number of the spindle coupled with axes X and Y (machine-dependent)
- **R**: **Center offset** – distance from eccentric center point to original center of rotation (radius value)
- **C**: **Position C** – C-axis angle of the center offset
- **F**: **Max. rapid traverse** – permissible rapid traverse for X and Y axes with activated coupling
- **V**: **Direction reversal in Y** (machine-dependent)
  - **V** = 0: The control uses the configured axis direction for Y-axis movements
  - **V** = 1: The control reverses the configured axis direction for Y-axis movements

Programming notes:
- Program a workpiece blank increased by the center offset in the radius if you are using turning cycles that are referenced to the workpiece-blank definition
- Program the starting point increased by the center offset in the radius if you are using turning cycles that are not referenced to the workpiece-blank definition
- Reduce the spindle speed if you increase the center offset
- Reduce the maximum rapid traverse **F** if you increase the center offset
- Use identical values for the parameter **Q** when activating and deactivating the coupling

Programming sequence:

- Position the cursor in the **MACHINING** program section
- Program function **G725** with **H1** (activate coupling)
- Program turning cycles
- Program function **G725** with **H0** (deactivate coupling)

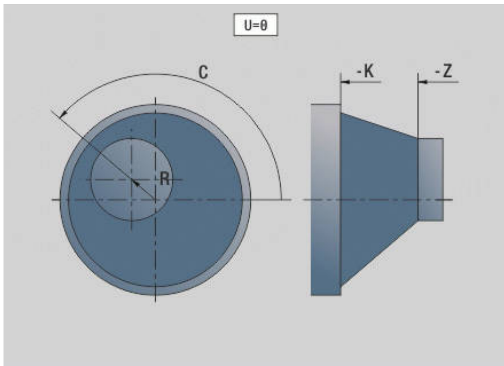
> Canceling the program run automatically deactivates the coupling.

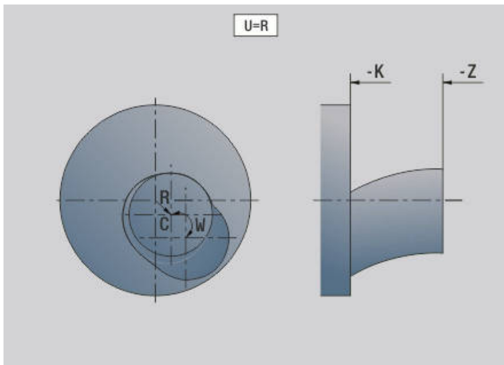
> Block scan (mid-program startup) is not available for non-circular turning with a coupled spindle (option 135, Synchronizing Funct.). Select an NC block before or after the non-circular turning program section.

Eccentric transition G726

G726 is used to machine turning contours outside the original turning center. In addition, **G726** offers the possibility to continuously change the position of the turning center along a straight line or a curve.

The turning contours are programmed using separate turning cycles.





> Refer to your machine manual.
> This function is set by your machine tool builder.
> Prerequisites:
> - Y-axis machining (option 70)
> - Synchronizing functions (option 135)

Parameters:

- **H**: **Activate coupling**
  - **H** = 0: Deactivate coupling
  - **H** = 1: Activate coupling
- **Q**: **Reference spindle** – number of the spindle coupled with axes X and Y (machine-dependent)
- **R**: **Center offset** – distance from eccentric center point to original center of rotation (radius value)
- **C**: **Position C** – C-axis angle of the center offset
- **F**: **Max. rapid traverse** – permissible rapid traverse for X and Y axes with activated coupling
- **V**: **Direction reversal in Y** (machine-dependent)
  - **V** = 0: The control uses the configured axis direction for Y-axis movements
  - **V** = 1: The control reverses the configured axis direction for Y-axis movements
- **Z**: **Z start** – reference value for the parameters **R** and **C**, as well as coordinate for tool pre-positioning
- **K**: **Z end** – reference value for the parameters **W** and **U**
- **W**: **Delta C [Z start to Z end]** – difference in C-axis angle with respect to **Z start** and **Z end**
- **U**: **Eccentricity at Z end** – distance from eccentric center point to original turning center (radius value)

NOTICE

Danger of collision!

When the coupling is activated, the control positions the tool in the Z axis to the value of the parameter **Z**. Danger of collision during approach!

- Pre-position the tool if required before activating the coupling (before the cycle)

Programming notes:

- Program a workpiece blank increased by the center offset in the radius if you are using turning cycles that are referenced to the workpiece-blank definition
- Program the starting point increased by the center offset in the radius if you are using turning cycles that are not referenced to the workpiece-blank definition
- Reduce the spindle speed if you increase the center offset
- Reduce the maximum rapid traverse **F** if you increase the center offset
- Use identical values for the parameter **Q** when activating and deactivating the coupling

Programming sequence:

- Position the cursor in the **MACHINING** program section
- Program function **G726** with **H1** (activate coupling)
- Program turning cycles
- Program function **G726** with **H0** (deactivate coupling)

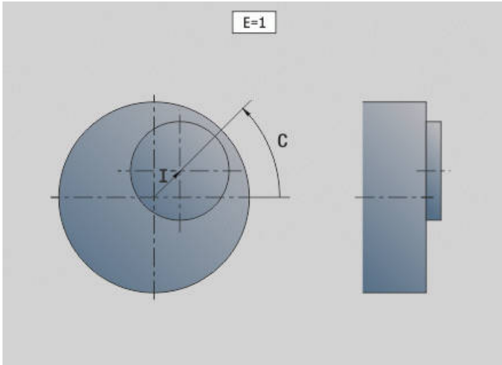
Canceling the program run automatically deactivates the coupling.

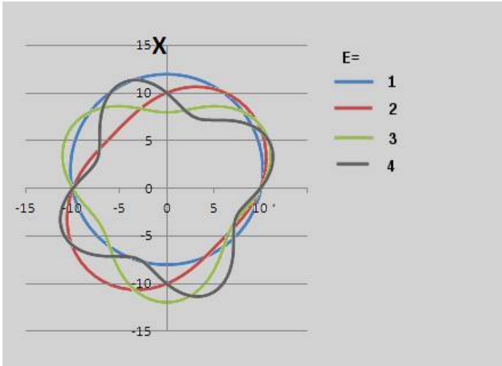
Block scan (mid-program startup) is not available for non-circular turning with a coupled spindle (option 135, Synchronizing Funct.). Select an NC block before or after the non-circular turning program section.

Non-circular X G727

G727 is used to machine elliptical polygons.

The turning contours are programmed using separate turning cycles.





> Refer to your machine manual.
> This function is set by your machine tool builder.
> Prerequisites:
> - Synchronizing functions (option 135)

Parameters:

- **H**: **Activate coupling**
  - **H** = 0: Deactivate coupling
  - **H** = 1: Activate coupling
- **Q**: **Reference spindle** – number of the spindle coupled with axes X and Y (machine-dependent)
- **I**: **X stroke +/-** – half of overlapping X movement (radius value)
- **C**: **Center offs. of C at Z start** – C-axis angle of the X stroke
- **F**: **Max. rapid traverse** – permissible rapid traverse for X and Y axes with activated coupling
- **E**: **Form factor?** – number of X strokes with respect to one spindle revolution
- **Z**: **Z start** – reference value for the parameter **C**
- **W**: **Delta C [°/mm Z]** – difference in C-axis angle with respect to a distance of 1 mm in the Z axis

NOTICE

Danger of collision!

When the coupling is activated, the control positions the tool in the Z axis to the value of the parameter **Z**. Danger of collision during approach!

- ▶ Pre-position the tool if required before activating the coupling (before the cycle)

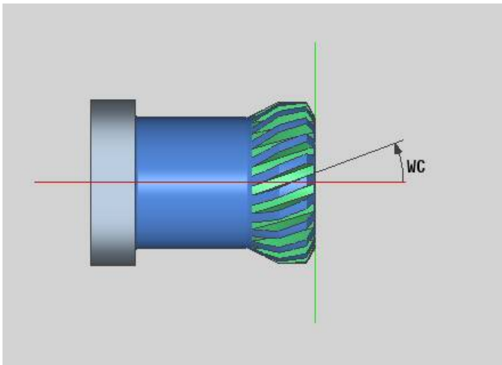
Programming notes:
- Program a workpiece blank increased by the center offset in the radius if you are using turning cycles that are referenced to the workpiece-blank definition
- Program the starting point increased by the center offset in the radius if you are using turning cycles that are not referenced to the workpiece-blank definition
- Reduce the spindle speed if you increase the center offset
- Reduce the maximum rapid traverse **F** if you increase the center offset
- Use identical values for the parameter **Q** when activating and deactivating the coupling

Programming sequence:
- Position the cursor in the **MACHINING** program section
- Program function **G727** with **H1** (activate coupling)
- Program turning cycles
- Program function **G727** with **H0** (deactivate coupling)

Canceling the program run automatically deactivates the coupling.

Compens. for helical teeth G728

With the **G728** function, you can compensate for a Z position-dependent angular offset between tool and workpiece. This function is required for hobbing helical teeth with **G808**.



Parameters:
- **H**: **Activate**:
  - **0: OFF**
  - **1: ON**
- **Q**: **Spindle with workpiece**
- **D**: **Number of teeth** – number of teeth of the workpiece
- **O**: **Module**
- **WC**: **Angle of inclination** of the gear
- **Z**: **Z start** – Z position where the angular difference is 0°
- **J**: **Offset of tool °/mm Z**

Operating notes:
- Make sure that the tool approaches the Z starting position without collision when the function is called.
- If you program offset **J**, this offset will be used directly as programmed. If you do not program **J**, the control will calculate the offset from the module, the number of teeth and the angle of inclination

6.30 Measuring the machine condition (option 155)

Refer to your machine manual.

The machine tool builder determines the performance range and behavior of these functions.

Prerequisites:

- Component Monitoring (option 155)

During their lifecycle, the machine components which are subject to loads (e.g., guides, ball screws, ...) become worn and thus, the quality of the axis movements deteriorates. This, in turn, affects the production quality.

With **Component Monitoring** (option 155) and the following cycles, the control is able to measure the current machine condition. As a result, any deviations from the machine's shipping condition due to wear and aging can be measured. The measurement results are stored in a text file that is readable for the machine tool builder. He can read and evaluate the data, and react with predictive maintenance, thereby avoiding unplanned machine downtimes.

The machine tool builder can define warning and error thresholds for the measured values and optionally specify error reactions.

Measuring the machine condition – fingerprint G238

Refer to your machine manual.

The machine tool builder determines the scope of function and behavior of this function.

In this cycle, the control performs various measurements in order to determine the current condition of the machine.

You can start a test run with parameter **H = 1**. In this mode, the machine performs the programmed movements, but no measurements are taken. During this process, you can change the traversing speed using the potentiometers.

During a measurement sequence (**H = 0** or no input), the feed-rate potentiometer is superimposed by the cycle. This means that you cannot change the traversing speed. The movement can only be stopped by setting the feed-rate potentiometer to zero.

You can select measurements for individual axes with the **Q**, **D**, and **V** parameters.

NOTICE

Danger of collision!

This cycle may perform extensive movements in one or more axes at rapid traverse! If you do not program a value in parameter **H** or set it to 0, the feed rate and rapid traverse potentiometers, and, if applicable, the spindle potentiometer, have no effect. There is a danger of collision!

- Before recording measured data, test the cycle in test mode with **H = 1**
- Contact your machine tool builder to learn about the type and range of movements before using Function **G238**.

Parameters:

- **H**: **Only axis movement (1)**
  - **0** or no input: Measurement sequence (feed-rate potentiometer is not effective)
  - **1**: Test run (feed-rate potentiometer is effective)
- **Q**: **Measurement method** – selection of the tests to be performed
  - **0**: All tests
  - **1**: Waterfall chart
  - **2**: Circular interpolation test
  - **3**: Frequency response
  - **4**: Envelope curve
- **D**: **Axis number** (X = 1, Y = 2, Z = 3, U = 4, V = 5, W = 6, A = 7, B = 8, C = 9) – first axis (only if **Q** has been defined)
- **V**: **Axis number** – second axis (only if **Q** has been defined)

To perform a circular interpolation test of the C axis on the counter spindle, define the parameters as follows:

- **Q**: **Measurement method** = 2: Circular interpolation test
- **V**: **Axis number** = 9: C

In this case, you must not define the **D** parameter.

Component monitoring G939

> Refer to your machine manual.
> This function must be set up by your machine manufacturer.

Using Function **G939**, the control performs a one-time component test. Your machine tool builder specifies the components to be tested and the type of test via the machine parameters.

> Your machine tool builder defines the components to be monitored in the machine parameter **CfgMonComponent** (130900).

Parameters:

- **ID**: **Key name** – name of the machine component to be monitored—defined by the machine tool builder in machine parameter **CfgMonComponent** (130900)

6.31 Programming variables

Fundamentals

A variable is a placeholder. Variables can contain various data during the program run.

The control provides a variety of variable types.

The following rules apply to the use of variables:

- Multiplication/division before addition/subtraction
- Up to 6 bracket levels
- Integer variables: Integer values between –32767 and +32768
- Real variables: Floating point numbers with max. 10 integers and 7 decimal places
- Do not use any blank spaces when programming variables.
- The variable number itself and an index value, if applicable, can be described by another variable, e.g.: **#g(#c2)**

The control provides the following functions:

Syntax	Functions
+	Addition
-	Subtraction
*****	Multiplication
/	Division
( )	Parenthesizing
=	Equate function
ABS(...)	Absolute amount
ROUND(...)	Round
SQRT(...)	Square root
SQRTA(.., ..)	Square root of (a^2+b^2)
SQRTS(.., ..)	Square root of ($a^2–b^2$)
INT(...)	Truncate decimal places
SIN(...)	Sine (in degrees)
COS(...)	Cosine (in degrees)
TAN(...)	Tangent (in degrees)
ASIN(...)	Arc sine (in degrees)
ACOS(...)	Arc cosine (in degrees)
ATAN(...)	Arc tangent (in degrees)
LOGN(...)	Natural logarithm
EXP(...)	Exponent function
BITSET(...)	Bitset function
STRING(...)	String
PARA(...)	Configuration data

> You can also program the listed functions by soft key.
> The soft-key row is available when the variable assignment function is active and the on-screen alphabetic keyboard is closed.

> Programming notes:
> - The distinction between variables that can be modified at runtime and those that cannot, as in predecessor controls, does not apply any longer. The NC program is no longer compiled before the program run, but at runtime.
> - Program NC blocks containing variable calculations with **Slide code $..** if your lathe has more than one slide. Otherwise, the calculations are repeated.
> - Positions and dimensions transferred into system variables are always indicated in metric form. This also applies when an NC program is run in inches.

Variable types

The control distinguishes the following variable types:

- General variables
  - Local variables
  - Global variables
  - Text variables
- Machine dimensions
- Tool compensation
- PLC variables

General variables

- **#l1 .. #L99**: Channel-dependent, local variables are effective within a main or a subprogram
- **#l1(1) .. #L99(1)**: Channel-dependent, initialized variables are effective in the initialization programming level, as well as in the subprograms called from this level.

  > This characteristic makes the channel-dependent initialized variables especially suitable for use in programs called expert programs. Undesired overlapping with the variables of the main program is thus avoided. In addition, all programmable variables continue to be available for the main program without restriction.
  > **Further information:** "Subprograms and expert programs", Page 283

- **#c1 .. #c30** Channel-dependent, global variables can be used for each slide (NC channel). The same variable numbers can be used on different slides without influencing each other. The variable content is provided globally on one channel. Global means that a variable described in a subprogram can also be evaluated in the main program, and vice versa
- **#g1 .. #g199**: Channel-independent, global real variables are provided once within the control. If an NC program changes a variable, the change applies to all slides. The variables are retained even when the control is switched off, and can be evaluated again after power-up.
- **#g200 .. #g299**: Channel-independent, global integer variables are provided once within the control. If an NC program changes a variable, the change applies to all slides. The variables are retained even when the control is switched off, and can be evaluated again after power-up.
- **#x1 .. #x20**: Channel-dependent, local text variables are effective within a main or subprogram. They can be read only on the channel for which there were defined

Example: General variables

. . .	
N.. #l1=#l1+1	
N.. G1 X#c1	
N.. G1 X(SQRT(3*(SIN(30)))	
N.. #g1=(ABS(#2+0.5))	
. . .	
N.. G1 Z#m(#l1)(Z)	
N.. #x1=“Text“	
N.. #g2=#g1+#l1*(27/9*3.1415)	
. . .	

Example: Channel-dependent, initialized variables

%_G238.ncs "TURN_V1.0"	External subprogram called from the main program
. . .	
VAR	
N.. #_debug = #l98(1)	Initialize variable
. . .	
N.. L“G938“ V1	Call another external subprogram
. . .	
%_G938.ncs "TURN_V1.0"	External subprogram called from the subprogram
. . .	
N.. IF #_debug==1	
N.. THEN	
N.. PRINT(“Debug“)	
N.. ENDIF	
. . .	
RETURN	
. . .	

If the variables are to be retained in the memory when the control is switched off, this feature must be activated by the machine tool builder in the **CfgNcPgmParState** machine parameter (no 200700).

If this feature is not activated, the variables values will always be "zero" after power-up.

You can also use variables to program M functions.

String variables

- The TIME function writes the date or time to a string variable. The value in this variable can then be engraved using an engraving cycle.
- Contents of variables can be converted to string variables and added.
- String variables can be output as floating-point numbers. They are automatically rounded off.
- File names can be specified by means of string variables.
  Further information: "File output for variables – WINDOW", Page 533""

Example: date and time

. . .	
N.. #x1=TIME("D.M.YY")	Date in string variable #x1
N.. #x2=TIME("h:m:s")	Time in string variable #x2
. . .	

Example: conversion to a string variable

. . .	
N.. #x1=STRING(#i21)	Convert variable #i21 to string variable #x1
N.. #x2=TIME("h:m:s")+STRING(#i21)	Add time and value in variable #i21
. . .	

Example: Output of floating-point number

. . .	
N.. #x1=STRING(12.43,1)	The number is rounded off and output with one decimal place.
. . .	

Machine dimensions

- **#m1(n) .. #m99(n)**: **n** is the designation of the axis (X, Z, Y) for which the machine dimension is to be read or written. The variable calculation uses the **mach_dim.hmd** table. **Simulation:** During the startup of the control, the **mach_dim.hmd** table is read by the simulation. The simulation function will now use the dedicated simulation table

Example: Machine dimensions

. . .	
N.. G1 X(#m1(X)*2)	
N.. G1 Z#m3(Z)	
N.. #m4(Z)=350	
. . .	

Tool compensation

- **#dt(n)**: **n** is the compensation direction (X, Z, Y, S) and **t** is the turret pocket number assigned to the tool. The variable calculation uses the **toolturn.htt** table. **Simulation**: When the program is selected, the **toolturn.htt** table is read by the simulation. The simulation function will now use the dedicated simulation table

Example:Tool compensations

. . .	
N.. G1 X(#m1(X)*2)	
N.. G1 Z#m3(Z)	
N.. #m4(Z)=350	
. . .	

> Tool information can also be interrogated directly via the **ID no.**. This may be necessary, for example, if no turret pocket has been assigned. For this purpose, program a comma and the **ID no.** of the tool after the desired identification, e.g. **#l1 = #d1(Z, "001")**.

PLC variables (event bits)

> Refer to your machine manual.
> This function must be set up by your machine manufacturer.

For variable programming, you can use logical, arithmetical, or text values from the PLC program. The control can read the PLC variables or write to them. The symbolic names you can use have been specified by the machine tool builder in the PLC program.

In earlier control versions, the reading part of this programming process was designed as "event bits."

- **#en(Symname)**: **n** is the data type, **Symname** the symbolic name of the PLC operand

The machine tool builder may also create symbolic names in indexed form. The index may vary.

```
#e1("Spindle[#l3].Direction")
```

- **#e1 (#e0)**: With **#e1**, the control accesses logical, integer or fractional values
- **#e2**: With **#e2**, the control accesses text values

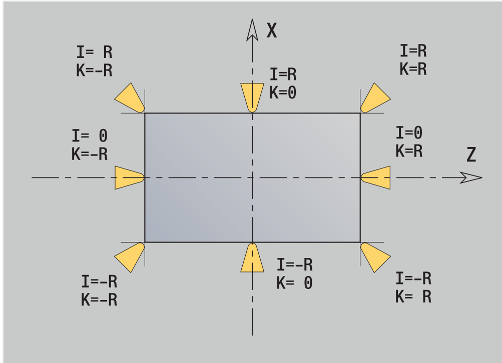
> Ensure that the variable type matches when assigning values. Text values from PLC variables can only be saved in string variables, numerical values only in normal variables

Example: PLC variable

. . .	
N.. #l4 = #e1("CoolingOn")	Read the state from the PLC variable and store it in #l4
N.. #e1("CoolingOn") =1	Overwrite the state of the PLC variable
N.. #e1("CoolingOn") =#l4	Restore the PLC variable with the stored value
...	
N.. #x3 = #e2("MyFieldName")	Write the state from the text variable to #x3
N.. #e2("MyFieldName") ="Hallo"	Overwrite the PLC variable with Hallo
N.. #e2("MyFieldName") =#x3	Restore the PLC variable with the stored value
. . .	
N.. #l1= #e1("Channel[2].Event[57]")	Save channel 2, event 57 in #l1

Reading tool data

Refer to your machine manual.
This function is also available on machines with a tool magazine.
The control will then use the magazine list instead of the turret list.



Use the following syntax to read the data of tools currently entered in the turret list: **#wn(select)**.

Access information about the currently loaded tool with the following syntax: **#w0(select)**.

Tool information can also be interrogated directly via the **ID no.**. This may be necessary, for example, if no turret pocket has been assigned: **#l1= #w1(select,"ID")**.

If a sequence of exchange is defined, program the first tool of the sequence. The control determines the data of the active tool.

Identification codes for tool information

#wn(ID)	Tool ID number (assign in text variable **#xn**)
#wn(PT)	P key of the tool *10 (e.g. 12.3 becomes 123)
#wn(WT)	Tool type (3-digit number)
#wn(WTV)	1st position of tool type
#wn(WTH)	2nd position of tool type
#wn(WTL)	3rd position of tool type
#wn(NL)	Usable length (inside turning and drilling tools)
#wn(HR)	Main machining direction (see tool orientation table)
#wn(NR)	Secondary machining direction of turning tools
#wn(AS)	Execution (see execution table)
#wn(ZZ)	Number of teeth (milling tools)
#wn(RS)	Cutting edge radius
#wn(ZD)	Stud diameter
#wn(DF)	Cutter diameter
#wn(SD)	Shank diameter
#wn(SB)	Cutting width
#wn(SL)	Tooth length
#wn(AL)	Length of first cut
#wn(FB)	Cutter width
#wn(WL)	Tool orientation
#wn(ZL)	Tool setting dimension in Z (from the tool list)
#wn(XL)	Tool setting dimension in X (from the tool list)
#wn(YL)	Tool setting dimension in Y (from the tool list)
#wn(TL)	Tool status (Tool Locked)
#wn(I)	Position of tool tip center in X
#wn(J)	Position of tool tip center in Y

#wn(K)	Position of tool tip center in Z
#wn(ZE)	Length of the tool in the current insert position: Distance between tool tip and slide reference point Z
#wn(XE)	Length of the tool in the current insert position: Distance between tool tip and slide reference point X
#wn(YE)	Length of the tool in the current insert position: Distance between tool tip and slide reference point Y
#wn(DN)	Diameter of drilling and milling tools
#wn(HW)	Principal angle in the normalized system (0° to 360°)
#wn(NW)	Secondary angle in the normalized system (0° to 360°)
#wn(EW)	Tool angle
#wn(SW)	Point angle
#wn(AW)	■ 0: No driven tool ■ 1: Driven tool
#wn(MD)	Direction of rotation: ■ 3: **M3** ■ 4: **M4**
#wn(CW)	Tilting plane angle
#wn(BW)	Angular offset
#wn(WTL)	Orientation
#wn(AC)	Cutting-edge insert angle
#wn(ZS)	Maximum cutting depth
#wn(GH)	Thread pitch
#wn(NE)	Number of secondary cutting edges
#wn(NS)	Number of the secondary cutting edge
#wn(FP)	Tool type: ■ 0 = normal tool ■ 1 = master tools ■ 2 = secondary cutting edge
#wn(Q)	Number of tool spindle
#wn(AS)	Execution left/right
#wn(X)	Setting dimension of holder in X
#wn(Z)	Setting dimension of holder in Z
#wn(Y)	Setting dimension of holder in Y
#wn(DX)	Compensation in X
#wn(DY)	Compensation in Y
#wn(DZ)	Compensation in Z
#wn(DS)	2nd compensation
#wn(BR)	Tool radius 2 (milling tool)
#wn(DC)	Compensation of tool radius 2 (milling tool)

Access to tool data of turret

#wn(select)

- **n** = turret pocket number
- **n** = 0 for the current tool
- **select** = designates the information to be read

Main machining direction

#wn(HR)

- 0: Undefined
- 1: +Z
- 2: +X
- 3: –Z
- 4: –X
- 5: +/–Z
- 6: +/–X

Execution

#wn(AS)

- 1: Right-hand
- 2: Left-hand

Tool orientation

#wn(WL) Reference: Machining direction of tool

- 0: On the contour
- 1: To the right of the contour
- – 1: To the left of the contour

Reading diagnostic bits

> Refer to your machine manual.
> This function is also available on machines with a tool magazine.
> The control will then use the magazine list instead of the turret list.

Use the following syntax to read diagnostic bits. You can access tools that are entered in the turret list.

> You can also read diagnostic bits with Multifix tools. For this purpose, program a comma and the **ID no.** of the tool after the desired identification, e.g. **#l1 = #t(3, "001")**.

Identification codes for diagnostic bits

#tn(1)	**Tool life expired or max. quantity exceeded**
#tn(2)	**Breakage detected via load monitoring (limit 2)**
#tn(3)	**Wear ascertained via load monitoring (limit 1)**
#tn(4)	**Wear according to load monitoring (total load)**
#tn(5)	**Wear measured through tool calibration**
#tn(6)	**Wear measured via in-process workpiece measurement**
#tn(7)	**Wear measured via post-process workpc. measurement**
#tn(8)	**Cutting edge is new**

Access to turret data

#tn(select)

- **n** = turret pocket number
- **n** = 0 for the current tool
- **select** = designates the information to be read

Reading the current NC information

Use the following syntax to read NC information that was programmed with G codes.

Identification codes for NC information

#n0(X)	Last programmed position X
#n0(Y)	Last programmed position Y
#n0(Z)	Last programmed position Z
#n0(A)	Last programmed position A
#n0(B)	Last programmed position B
#n0(C)	Last programmed position C
#n0(U)	Last programmed position U
#n0(V)	Last programmed position V
#n0(W)	Last programmed position W
#n0(CW)	Tool insert angle (0 or 180 degrees)
#n18(G)	Active working plane
#n40(G)	Status of TRC
#n47(P)	Current safety clearance
#n52(G)	Oversize **G52_Geo** taken into account 0=no / 1=yes
#n57(X)	Oversize in X
#n57(Z)	Oversize in Z
#n58(P)	Equidistant oversize
#n95(G)	Programmed feed type (**G93**/**G94**/**G95**)
#n95(Q)	Spindle number of the last programmed feed rate
#n95(F)	Last programmed feed rate
#n97(G)	Programmed speed type (**G96**/**G97**)
#n97(Q)	Spindle number of the last programmed speed type
#n97(S)	Last programmed speed
#n119(Q)	Number of the selected C axis
#n120(X)	Reference diameter X for calculating CY
#n147(I)	Current safety clearance in working plane
#n147(K)	Current safety clearance in infeed direction

Access to current NC information

#nx(select)

- **x** = G-code number
- **select** = designates the information to be read

Active working plane

#n18(G)

- 17: XY plane (front or rear)
- 18: XZ plane (turning)
- 19: YZ plane (plan view / lateral surface)

Status of TRC/MCRC

#n40(G)

- 40: **G40** active
- 41: **G41** active
- 42: **G42** active

Number of the selected C axis

#n119(G)

- 0: No C axis
- 1: C axis 1
- 2: C axis 2

Current machine status

Active wear compensation (G148)

#n148(O)

- 0: **DX**, **DZ**
- 1: **DS**, **DZ**
- 2: **DX**, **DS**

Pocket data of entered tool

#n601(n)

- **S**: Number of cutting edge
- **M**: Magazine number
- **ppp**: Pocket number

Output in the format **SMppp**

Free magazine pocket

#n610(H)

- **M**: Magazine number
- **ppp**: Pocket number

Output in the format **Mppp**

Software limit switches

#n707(n,1)

Identification code of axis:

- **n**: Axis X, Y, Z, U, V, W, A, B, C
- 1: Minimum value
- 2: Maximum value

Datum shift

#n920(G)

Status of functions **G920**/**G921**:

- 0: **G920**/**G921** not active
- 1: **G920** active
- 2: **G921** active

Reading general NC information

Use the following syntax to read general NC information.

Identification codes for tool information

#i1	Current operating mode
#i2	Active unit of measure (inches/metric)

#i3	■ Main spindle = 0 ■ Counterspindle with mirroring Z = 1 ■ Tool mirroring in Z = 2 ■ Tool + path mirroring in Z = 3
#i4	**G16** active = 1
#i5	Last programmed tool number
#i6	Start block search active = 1
#i7	System is DataPilot = 1
#i8	Selected language
#i9	If Y axis is configured = 1
#i10	If B axis is configured = 1
#i11	If the tool pocket in X is mirrored to the machine system = 1
#i12	If U axis is programmable = 1
#i13	If V axis is programmable = 1
#i14	If W axis is programmable = 1
#i15	If U axis is configured = 1
#i16	If V axis is configured = 1
#i17	If W axis is configured = 1
#i18	Datum shift of the Z axis
#i19	Datum shift of the X axis
#i20	Last programmed path function (**G0**, **G1**, **G2**...)
#i21	Current quantity (workpiece counter)
#i22	If U axis is coupled with X axis = 1
#i23	If V axis is coupled with Y axis = 1
#i24	If W axis is coupled with Z axis = 1
#i25	If magazine exists = 1
#i26	P key of actual tool *10 + MU from tool preselection
#i27	P key of desired tool *10 from tool preselection
#i28	Angle of oblique axis Y
#i29	P key of the tool *10 that has reached the maximum tool life
#i30	P key of the tool *10 that has reached the maximum workpiece quantity
#i31	If contour groups have been programmed = 1 Only for automatic working plan generation
#i32	Datum shift of the contour in Z from the contour group definition in DIN PLUS (1...4)
#i33	If contour group programming is to be done in AWG = 1 Only for automatic working plan generation

#i34	If only SLIDE $2 has been specified in the program header = 1
#i36	Channel number of the tilted C axis (0 to 5) Only with activation of C axis in the NC program
#i38	Read the PLC parts counter marker
#i39	Current channel number
#i99	Return code of subprograms ■ Values ■ Strings

Active operating mode

#i1

- 2: Machine
- 3: Simulation
- 5: TSF menu

Languages

#i8

- 0: **ENGLISH**
- 1: **GERMAN**
- 2: **CZECH**
- 3: **FRENCH**
- 4: **ITALIAN**
- 5: **SPANISH**
- 6: **PORTUGUESE**
- 7: **SWEDISH**
- 8: **DANISH**
- 9: **FINNISH**
- 10: **DUTCH**
- 11: **POLISH**
- 12: **HUNGARIAN**
- 14: **RUSSIAN**
- 15: **CHINESE**
- 16: **CHINESE_TRAD**
- 17: **SLOVENIAN**
- 19: **KOREAN**
- 21: **NORWEGIAN**
- 22: **ROMANIAN**
- 23: **SLOVAK**
- 24: **TURKISH**

PLC parts counter marker

#i38

- 0: Attribute not defined or parts count not reached
- 1: Parts count reached

Reading configuration data – PARA

The **PARA** function is used to read configuration data. To do this, use the parameter designations from the configuration parameters. You also use the designations from the configuration parameters to read user parameters.

When you read optional parameters, check whether the return value is valid. Depending on the data type of the parameter (**REAL**/**STRING**) the value **0** or the text **_EMPTY** is returned when reading an optional attribute that has not been set.

Access to configuration data

PARA (key, entity, attribute, index)

- **Key**: Key word
- **Entity**: Name of the configuration group
- **Attribute**: Element name
- **Index**: Array number if the attribute is from an array

Example: PARA function

. . .	
N.. #l10=PARA("","CfgDisplayLanguage","ncLanguage")	Reads the number of the currently selected language
N.. #l1=PARA("","CfgGlobalTechPara", "safetyDistWorkpOut")	Reads the external safety clearance on the machined part (SAT)
N.. #l1=PARA("Z1","CfgAxisProperties", "threadSafetyDist")	Reads the thread safety clearance for Z1
N.. #l1=PARA("","CfgCoordSystem","coordSystem")	Reads the machine orientation number
. . .	
#x2=PARA("#x30","CfgCAxisProperties"," relatedWpSpindle",0)	Check whether the optional parameter is set
IF #x2<>"_EMPTY"	Evaluation:
THEN	
	The parameter "relatedWpSpindle" was set
ELSE	
	The parameter "relatedWpSpindle" was not set
ENDIF	

Determining the index of a parameter element – PARA

You activate the index search for an element by appending the name of the list element to the attribute, separated by a comma.

Example:

Determining the logical axis number of spindle **S1**

#c1 = PARA("", "CfgAxes", "axisList,S1", 0)

The function returns the index of the **S1** element in the **axisList** attribute of the **CfgAxes** entity. The index of element **S1** equals the logical axis number in this example.

Access to configuration data

PARA (key, entity, attribute, element, index)	■ **Key**: Key word ■ **Entity**: Name of the configuration group ■ **Attribute, name**: Attribute name and element name ■ **Index**: 0 (not required)

> Without the attribute extension **S1**, the function would read the element located at the list index number **0**. Since the element is a string in this example, the result has to be assigned to a string variable.
>
> **#x1 = PARA("", "CfgAxes", "axisList", 0)**
>
> The function reads the string name of the element at list index number **0**.

Expanded variable syntax CONST – VAR

By defining the **CONST** or **VAR** keywords, you can assign names to variables. The keywords can be used in the main program and in the subprogram. To use the definitions in a subprogram, you need to declare the constant or variable before the **MACHINING** code.

> **Rules for defining constants and variables:** The names of constants and variables must be preceded by an underscore. They can comprise lower case letters, numbers and the underscore character.
>
> The maximum length must not exceed 20 characters.

Variable names with VAR

By assigning variable names, you make it easier to read an NC program. To do this, you must insert the program section **VAR**. In this program section, you assign the variable designations to the variables.

Example: Free-text variables

%abc.nc	
VAR	
#_rohdm=#l1	#_rohdm is a synonym of #l1
BLANK	
N..	
FINISHED PART	
N..	
MACHINING	
N..	
. . .	

Example: Subprogram

%SP1.ncS	
VAR	
#_wo = #c1	Tool orientation
MACHINING	
N.. #_wo = #w0(WTL)	
N.. G0 X(#_posx*2)	
N.. G0 X#_start_x	
. . .	

Example: Assignment using variables

%NC1.nc	
VAR	
#_vorschub=#l1(200)	Assignment

This has the following meaning of an initializing assignment:

#_vorschub=200	Direct assignment

If you use the variable, you can overwrite the value later in the program at any time. The variable is known later in the NC programs and the subprograms starting from the point at which it has been declared and can be edited in both. This enables the subprograms to supply further data about the calling function in addition to the RETURN value. Unlike with the #c and #g variables, these variables exist only within an NC program and are therefore especially suitable for use in expert programs.

Definition of constants with CONST

Possibilities of defining constants:

- Direct assignment of values
- Internal interpreter information as constants
- Assignment of names to the transfer variables of subprograms

Use the following internal information to define constants in the **CONST** section.

Internal information for defining constants

__n0_x	768 Last programmed position X
__n0_y	769 Last programmed position Y
__n0_z	770 Last programmed position Z
__n0_c	771 Last programmed position C
__n40_g	774 Status of TRC
__n148_o	776 Active wear compensation
__n18_g	778 Active working plane
__n120_x	787 Reference diameter X for calculating CY
__n52_g	790 Oversize **G52_Geo** taken into account 0=no / 1=yes
__n57_x	791 Oversize in X
__n57_z	792 Oversize in Z
__n58_p	793 Equidistant oversize
__n150_x	794 Cutting width shifted in X by **G150**/**G151**
__n150_z	795 Cutting width shifted in Z by **G150**/**G151**
__n95_g	799 programmed feed type **G93**/**G94**/**G95**)
__n95_q	796 Spindle number of the programmed feed rate
__n95_f	800 Last programmed feed rate
__n97_g	Programmed speed type **G96**/**G97**)
__n97_q	797 Spindle number of the programmed speed type
__n97_s	Last programmed speed
__la-__z	Subprogram transfer values

The constant **_pi** is predefined to the value 3.1415926535989 and can be used directly in every NC program.

Example: Main program

%abc.nc	
CONST	
_square_root2 = 1.414213	Direct value assignment
_square_root_2 = SQRT(2)	Direct value assignment
_posx = __n0_x	Internal information
VAR	
. . .	
BLANK	
N..	
FINISHED PART	
N..	
MACHINING	
N..	
. . .	

Example: Subprogram

%SP1.ncS	
CONST	
_start_x=__la	Subprogram transfer value
_posx = __n0_x	Internal constant
VAR	
#_wo = #c1	Tool orientation
MACHINING	
N.. #_wo = #w0(WTL)	
N.. G0 X(#_posx*2)	
N.. G0 X#_start_x	
. . .	

6.32 Data input and data output

Output window for variables – WINDOW

WINDOW (x) opens an output window with **x** lines. The window is opened as a result of the first input/output. **WINDOW (0)** closes the window.

Syntax: WINDOW (line number) (0 <= line number <= 20)

The standard output window comprises three lines. You do not need to program it.

Example: Output window for variables – WINDOW

. . .	
N 1 WINDOW(8)	
N 2 INPUT("query: ",#l1)	
N 3 #l2=17*#l1	
N 4 PRINT("result: ",#l1,"*17 = ",#l2)	
. . .	

File output for variables – WINDOW

The command **WINDOW (x, file name)** saves the **PRINT** instruction in a file with the defined name and the extension **.LOG** in the directory **V:\nc_prog**. The file is overwritten when the **WINDOW** command is run again.

Saving the **LOG** file is only possible in the **Program run** submode.

Syntax: WINDOW (line number, file name)

Example: File output for variables – WINDOW

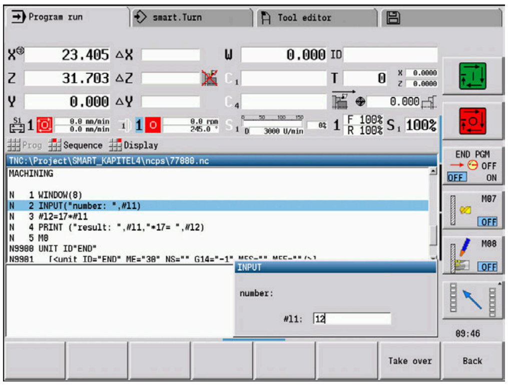
. . .	
N 1 WINDOW(8,"VARIO")	
N 2 INPUT("query: ",#l1)	
N 3 #l2=17*#l1	
N 4 PRINT("result: ",#l1,"*17 = ",#l2)	
. . .	

You can specify the file name using a string variable.

Example: data output with string variables

. . .	
N 11 #l1 = #i39	Assign the current channel number
N 12 #x3 = "Channel"	Assign the string variable
N 13 #x2 = STRING(#l1)	Convert the channel number into a string
N 14 #x3 = #x3 + #x2	Add the variable values
N 15 WINDOW(5, #x3)	
N 16 PRINT("Channelinfo")	
. . .	

Input of variables – INPUT



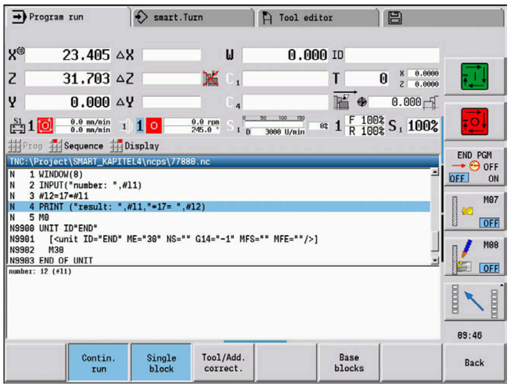
Use **INPUT** to program the input of variables.

Syntax: INPUT (text, variable)

You define the input text and the number of the variable. The control stops the interpretation at **INPUT**, outputs the text and waits for input of the variable value. Instead of an input text, you can also program a string variable, such as **#x1**.

The control displays the input after having completed the **INPUT** command.

Output of # variables – PRINT



PRINT can be used to output texts and variable values during program run. You can program a succession of several texts and variables.

Syntax: PRINT (text, variable, text, variable, ...)

Example: Output of # variables – PRINT

```
N 4 PRINT("result: ",#l1,"*17 = ",#l2)
```

6.33 Conditional block run

Program branch IF..THEN..ELSE..ENDIF

A conditional branch consists of the elements:

- **IF**, followed by a condition. The condition includes a variable or mathematical expression on either side of the relational operator.
- **THEN**. If the condition is fulfilled, the **THEN** branch is executed
- **ELSE**. If the condition is not fulfilled, the **ELSE** branch is executed
- **ENDIF** concludes the conditional program branch

Interrogate bitset: You can also use the **BITSET** function as condition. The function returns **1** if the numerical value contains the requested bit. The function returns **0** if the numerical value does not contain the requested bit.

Syntax:

- **BITSET (x,y)**
  - **x**: Bit number (0 to 15)
  - **y**: Numerical value (0 to 65535)

The relationship between bit number and numerical value is shown in the table. You can also use variables for **x**, **y**.

Programming:

- Select **Extras** > **DIN PLUS word...**. The control opens the **Insert DIN PLUS word** selection list
- Select **IF**
- Enter the condition
- Insert NC blocks of the **THEN** branch
- If required: Insert NC blocks of the **ELSE** branch

> - NC blocks with **IF**, **THEN**, **ELSE**, **ENDIF** can have no further commands
> - You can combine up to two conditions

Relational operators

<	Less than
<=	Less than or equal to
<>	Not equal to
>	Greater than
>=	Greater than or equal to
==	equal to

Combining conditions

AND	Logical AND operation
OR	Logical OR operation

Conversion table

Bit	Numerical value
0	1
1	2
2	4
3	8
4	16
5	32
6	64
7	128
8	256
9	512
10	1024
11	2048
12	4096
13	8192
14	16384
15	32768

Example: IF... THEN... ELSE... ENDIF

```
N.. IF (#l1==1) AND (#g250>50)
N.. THEN
N.. G0 X100 Z100
N.. ELSE
N.. G0 X0 Z0
N.. ENDIF
. . .
N.. IF 1==BITSET(0,#l1)
N.. THEN
N.. PRINT("Bit 0: OK")
. . .
```

Interrogating variables and constants

With the **DEF**, **NDEF**, and **DVDEF** elements, you can interrogate whether a valid value was assigned to a variable or a constant. For example, an undefined variable can return the value **0**, just like a variable that has explicitly been assigned the value **0**. You can prevent undesired program jumps by checking the variables.

Programming:

- Select **Extras** > **DIN PLUS word...**. The control opens the **Insert DIN PLUS word** selection list
- Select **IF**
- Enter the required interrogation element (**DEF**, **NDEF** or **DVDEF**)
- Enter the name of the desired variable or constant

> Enter the variable name without the character **#**, e.g. **IF NDEF(__la)**

Interrogation elements of variables and constants:

- **DEF**: A value has been assigned to a variable or constant
- **NDEF**: No value has been assigned to a variable or constant
- **DVDEF**: Interrogation of an internal constant

Example: Interrogating a variable in a subprogram

N.. IF DEF(__la)	
N.. THEN	
N.. PRINT("Value:",#__la)	
N.. ELSE	
N.. PRINT("#__la is not defined")	
N.. ENDIF	
. . .	

Example: Interrogating a variable in a subprogram

N.. IF DEF(__lb)	
N.. THEN	
N.. PRINT("#__lb is not defined")	
N.. ELSE	
N.. PRINT("Value:",#__lb)	
N.. ENDIF	
. . .	

Example: Interrogating constants

N.. IF DVDEF(__n97_s)	
N.. THEN	
N.. PRINT("__n97_s is defined",#__n97_s)	
N.. ELSE	
N.. PRINT("#__n97_s is not defined")	
N.. ENDIF	
. . .	

Program repeat WHILE..ENDWHILE

A program repeat consists of the elements:

- **WHILE**, followed by a condition. The condition includes a variable or mathematical expression on either side of the relational operator.
- **ENDWHILE** concludes the conditional program repeat

NC blocks programmed between **WHILE** and **ENDWHILE** are executed repeatedly for as long as the condition is fulfilled. If the condition is not fulfilled, the control continues execution of the program with the block programmed after **ENDWHILE**.

Interrogate bitset: You can also use the **BITSET** function as condition. The function returns **1** if the numerical value contains the requested bit. The function returns **0** if the numerical value does not contain the requested bit.

Syntax:

- **BITSET (x,y)**
  - **x**: Bit number (0 to 15)
  - **y**: Numerical value (0 to 65535)

The relationship between bit number and numerical value is shown in the table. You can also use variables for **x**, **y**.

Programming:

- Select **Extras** > **DIN PLUS word...**. The control opens the **Insert DIN PLUS word** selection list
- Select **WHILE**
- Enter the condition
- Insert NC blocks between **WHILE** and **ENDWHILE**

> - You can combine up to two conditions.
> - If the condition you program in the **WHILE** command is always true, the program remains in an endless loop. This is one of the most frequent causes of error when working with program repeats.

Relational operators

<	Less than
<=	Less than or equal to
<>	Not equal to
>	Greater than
>=	Greater than or equal to
==	equal to

Combining conditions

AND	Logical AND operation
OR	Logical OR operation

Conversion table

Bit	Numerical value
0	1
1	2
2	4
3	8
4	16
5	32
6	64
7	128
8	256
9	512
10	1024
11	2048
12	4096
13	8192
14	16384
15	32768

Example: WHILE..ENDWHILE

```
. . .
N.. WHILE (#l4<10) AND (#l5>=0)
N.. G0 Xi10
. . .
N.. ENDWHILE
. . .
```

Program branch SWITCH..CASE

The program branch consists of the elements:

- **SWITCH**, followed by a variable. The content of the variable is interrogated in the following **CASE** statement
- **CASE x**: The **CASE** branch is run with the variable value **x**. **CASE** can be programmed repeated times
- **DEFAULT**: This branch is executed if no **CASE** statement matched the variable value. **DEFAULT** can be omitted
- **BREAK**: Concludes the **CASE** branch or **DEFAULT** branch

Programming:

- Select **Extras** > **DIN PLUS word...**. The control opens the **Insert DIN PLUS word** selection list
- Select **SWITCH**
- Enter the **switch** variable
- For each **CASE** branch:
  - Under **Extras** > **DIN PLUS word...**, select **CASE**
  - Enter the **SWITCH** condition (value of the variable) and insert the NC blocks to be executed
- For the **DEFAULT** branch: Insert the NC blocks to be executed

Example: SWITCH..CASE

. . .	
N.. SWITCH #g201	
N.. CASE 1	Executed if #g201=1
N.. G0 Xi10	
. . .	
N.. BREAK	
N.. CASE 2	Executed if #g201=2
N.. G0 Xi20	
. . .	
N.. BREAK	
N.. DEFAULT	No CASE statement matched the variable value
N.. G0 Xi30	
. . .	
N.. BREAK	
N.. ENDSWITCH	
. . .	

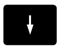
Skip level

In the **Program run** submode you can set and activate skip levels. In the next program run, the control will not execute the NC blocks defined with the set and activated skip level.

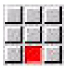
Further information: User's Manual

Before you can set and activate skip levels, you need to define them in the program:

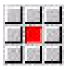
- Open the program in the **smart.Turn** operating mode



- Position the cursor in the **MACHINING** program section on the NC block to be skipped



- Select the **Extras** menu item



- Select the **Skip level...** menu item
- > The control opens a pop-up window
- Enter the number of the skip level in the **/ Deletion** parameter

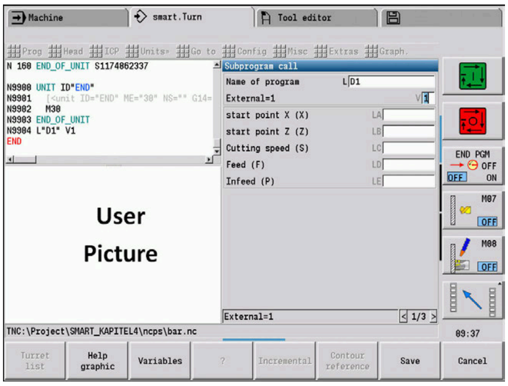
- Press the **OK** soft key

You can assign more than one skip level to an NC block by entering a string of numerals in the **/ Deletion** parameter. Thus, an input of **159** would correspond to the skip levels **1**, **5**, and **9**.

To clear the defined skip levels, program the parameter without a value and press the **OK** soft key to confirm.

6.34 Subprograms

Subprogram call L "xx" V1



The subprogram contains the following elements:

- **L**: Identifying letter for subprogram call
- **"xx"**: Name of the subprogram—file name for external subprograms (max. 16 letters or numbers)
- **V1**: Identification code for external subprograms—omitted for local subprograms

Note on using subprograms:

- External subprograms are defined in a separate file. They can be called from any main program or other subprogram
- Local subprograms are in the main program file. They can be called only from the main program
- Subprograms can be nested up to 6 times. Nesting means that another subprogram is called from within a subprogram
- Recursion should be avoided
- You can program up to 29 transfer values in one subprogram call
  - Designations: **LA** to **LF**, **LH**, **I**, **J**, **K**, **O**, **P**, **R**, **S**, **U**, **W**, **X**, **Y**, **Z**, **BS**, **BE**, **WS**, **AC**, **WC**, **RC**, **IC**, **KC** and **JC**
  - The identification code within the subprogram is: **#__..** followed by the parameter designation in lowercase letters (for example: **#__la**)
  - You can use these transfer values when programming with variables within the subprogram
- String variables: **ID** and **AT**
- The variables **#l1** to **#l99** are available in every subprogram as local variables
- To transfer a variable to the main program, program the variable or the content of a variable after the fixed word **RETURN**. In the main program, the information is available in **#i99**.
- If a subprogram is to be executed repeatedly, in the **Number of repetitions Q** parameter define the number of times the subprogram is to be repeated
- A subprogram ends with **RETURN**

> The parameter **LN** is reserved for the transfer of block numbers. This parameter may receive a new value when the NC program is renumbered.

Dialogs in subprogram calls

You can define up to 30 parameter descriptions that precede/follow the input fields in an external subprogram. The units of measure are defined using code numbers. Depending on the setting "metric" or "inches," the control shows the designations (of the units of measure). When calling external subprograms that contain a parameter list, parameters not contained in this list are omitted from the call dialog.

The parameter descriptions can be positioned within the subprogram as desired. The control searches for subprograms in the sequence: current project, standard directory and then machine manufacturer directory.

Parameter descriptions:

- **[//]** – Beginning
- **[pn=n; s=...]** (parameter text max. 25 characters)
  - **pn**: Parameter designations (**la**, **lb**, ...)
  - **n**: Code number for units of measure
    - 0: Non-dimensional
    - 1: mm or inches
    - 2: mm/rev or in./rev
    - 3: mm/min or in./min
    - 4: m/min or ft/min
    - 5: Rev/min
    - 6: Degrees (°)
    - 7: µm or µinch
- **[//]** – End

Example: Dialogs

```
...
[//]
[la=1; s=bar diameter]
[lb=1; s=starting point in Z]
[lc=1; s=chamfer/rounding arc (-/+)]
...
[//]
...
```

Help graphics for subprogram calls

With help graphics you illustrate the call parameters of subprograms. The control places the help graphics to the left next to the dialog box of the subprogram call.

If you append an underscore **_** and the input field name in capital letters (beginning with **L**) to the name of the file name, a separate graphic will be displayed for that input field. If no separate help graphic exists for an input field, the graphic for the subprogram will be displayed (if available). In the standard setting, the help window is displayed only if there is a graphic for the subprogram. Even if you want to use individual graphics for the address letters, you should define a graphic for the subprogram.

Graphic format:

- BMP, PNG, JPG images
- Size 440x320 pixels

You integrate help graphics for subprogram calls as follows:

- The subprogram name, entry field name and the appropriate extension (BMP, PNG, JPG) must be used as file name for the help graphic.
- Transfer the help graphic to the directory: **\nc_prog\Pictures**

6.35 M commands

M commands for program-run control

Refer to your machine manual.

The effect of machine commands depends on the configuration of your machine.

On your lathe, other M commands may apply for the listed functions.

M commands for program-run control

M00	**Unconditional stop** The program run stops. NC start resumes the program run.
M01	**Optional stop** If the **Contin. run** soft key is not active in Automatic mode, the program run stops with **M01**. Press NC start to resume the program run. If **Contin. run** is active, the program continues without stopping.
M18	**Counting pulse**
M30	**End of program** **M30** means "end of program" (you do not need to program **M30**). If you press NC start after **M30**, program execution is repeated from the start of the program.
M91	**Stop, no spindle stop M91**
M97	**Program synchronization** **Further information:** "Synchronous function M97", Page 550
M417	**Activate protection zone monitoring**
M418	**Deactivate protection zone monitoring**
M99 NS..	**Program end with restart** **M99** means end program and start again. The control restarts program execution from: ■ The start of program if no **NS** is defined ■ The block number **NS** if an **NS** is entered

Modal functions (feed rate, spindle speed, tool number, etc.) which are effective at the end of program remain in effect when the program is restarted. You should therefore reprogram the modal functions at the start of program or at the startup block (with **M99**).

Machine commands

Refer to your machine manual.

The effect of machine commands depends on the configuration of your machine.

On your lathe, other M commands may apply for the listed functions.

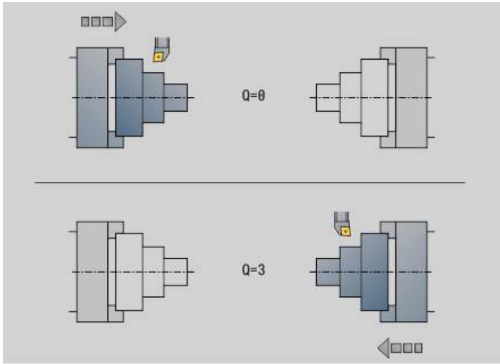
The following table lists the **M** commands used on most machines.

M commands as machine commands

M03	**Main spindle on (cw)**
M04	**Main spindle on (ccw)**
M05	**Main spindle stop**
M12	**Lock main spindle brake**
M13	**Release main spindle brake**
M14	**C axis on**
M15	**C axis off**
M19	**Spindle stop at position C**
M40	**Shift gear to range 0 (neutral)**
M41	**Shift gear to range 1**
M42	**Shift gear to range 2**
M43	**Shift gear to range 3**
M44	**Shift gear to range 4**
Mx03	**Spindle x on (cw)**
Mx04	**Spindle x on (ccw)**
Mx05	**Spindle x stop**

6.36 Assignment, synchronization, workpiece transfer

Converting and mirroring G30



G30 converts **G** codes, **M** functions, and the **No. of spindle**. **G30** mirrors traverse paths and tool dimensions and shifts the machine datum by the datum offset defined for the axis.

Parameters:

- **H**: **Table No.** of the conversion table (possible only if the machine tool builder has configured a conversion table)
- **Q**: **No. of spindle** (default: 0)

Application: For full-surface machining, you describe the complete contour, machine the front face, rechuck the workpiece using an expert program, and then machine the rear face. To enable you to program rear-face machining in the same way as front-face machining (Z-axis orientation, direction of rotation for arcs, etc.), the expert program includes commands for converting and mirroring.

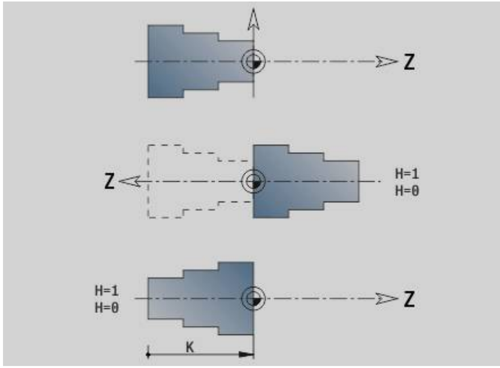
NOTICE

Danger of collision!

Conversions and mirroring will be retained when you switch the operating mode (e.g., between **Machine** operating mode and **Program run** submode). Danger of collision during subsequent machining operations!

- Always intentionally switch off converting or mirroring
- Alternatively, reselect the program

Transformations of contours G99



With the **G99** function you can select a contour group, mirror contours, move them and bring the workpiece to the desired machining position.

Parameters:

- **Q**: number of the **Contour group**
- **D**: **No. of spindle**
- **X**: **Contour pos. in graphic** – move in X (diameter value)
- **Z**: **Contour pos. in graphic** – move in Z
- **V**: **Mirror the Z axis (1)**
  - **V** = 0: Do not mirror
  - **V** = 1: Mirror
- **H**: transformation type – **Move/move + mirror**
  - **H** = 0: Contour shift, not mirroring
  - **H** = 1: Contour shift, mirroring and reversing the direction of the contour description
- **K**: **Length of workpiece shift** – shift coordinate system in Z direction
- **O**: **Omit elements**
  - **O** = 0: All contours are transformed
  - **O** = 1: Auxiliary contours are not transformed
  - **O** = 2: Face contours are not transformed
  - **O** = 4: Lateral contours are not transformed

  You can also add input values in order to combine various settings (e.g. **O3** Do not transform auxiliary contours or face contours)

> Program **G99** again if the workpiece is transferred to another spindle and/or moves its position in the working space.

Setting a synchronizing mark G162

> Refer to your machine manual.
> This function is only available on a machine with multiple channels (option 153).

G162 sets a synchronizing mark. Machining is continued on this slide. Another slide waits for the slide to reach the synchronizing mark.

Parameters:

- **H**: **Sync.mark no.** – number of the synchronizing mark (range: 0 <= **H** <=15)

One-sided synchronization G62

Refer to your machine manual.

This function is only available on a machine with multiple channels (option 153).

G62 enables you to program the synchronization of two slides. The slide programmed with **G62** waits until slide **Q** has reached the synchronizing mark **H** that has been set with **G162**.

If you use parameter **O** for programming **G62**, the slide waits until the synchronizing mark **H** and the programmed coordinate have been reached.

Parameters:

- **H**: **Sync.mark no.** – number of the synchronizing mark (range: 0 <= **H** <=15)
- **Q**: **No. of slide** – slide for which the control will wait
- **O**: **Direction** (default: 0)
  - **O** = –1: The slide waits until slide Q has traversed the synchronizing mark in negative direction in the specified axis direction.
  - **O** = 0: The slide waits until slide Q has reached the synchronizing mark.
  - **O** = 1: The slide waits until slide Q has traversed the synchronizing mark in positive direction in the specified axis direction.
- **X**: **Diameter** – coordinate at which the waiting process ends
- **Z**: **Length** – coordinate at which the waiting process ends
- **Y**: **Length** – coordinate at which the waiting process ends

Please note:

- **G162** and **G62** must be defined in the same main program.
- If you are using coordinates, the control must reach this coordinate. Therefore, instead of synchronizing to the end point of an NC block, synchronize to a coordinate that will reliably be traversed.

Example: G60

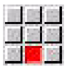
. . .	
$1 N10 G62 Q2 H5	Slide $1 waits until slide $2 has reached the mark 5
...	
$2 N40 G62 Q1 O1 H7 X200	Slide $2 waits until slide $1 has reached the mark 7, and position X > 200
. . .	

Synchronous start of slides G63

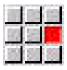
Refer to your machine manual.
This function is only available on a machine with multiple channels (option 153).

G63 starts the programmed slides simultaneously (synchronously).

The slides involved can be programmed as follows:



- Select the **Extras** menu item



- Select the **Slide...** menu item
- Enter the slide numbers

Synchronous function M97

Refer to your machine manual.
This function is only available on a machine with multiple channels (option 153).

M97 synchronizes all programmed slides. Each slide waits until all slides have reached this block—the control does not continue program run until all slides have reached the block.

If you need more than one synchronization point, use parameters to program M97.

Parameters:

- **H**: **Sync.mark no.** – number of synchronizing mark (evaluated only during NC program interpretation)
- **Q**: **No. of slide** – slide for which the control will wait
- **D**: **on/off**
  - D = 0: synchronization during run time of NC program
  - D = 1: synchronization exclusively during interpretation of the NC programs

Example: M97

. . .	
$1$3 N110 M97	Slide $1 and slide $3 wait for each other
...	
$1 N230 M97 H1 Q123	Slide $1, slide $2 and slide $3 wait for each other
. . .	
$1 N340 M97 H1 Q13 D1	The calculations (interpretations) of slide $1 and slide $3 wait for each other
. . .	

Spindle synchronization G720

Refer to your machine manual.
This function must be set up by your machine manufacturer.

G720 controls the workpiece transfer from the **Master spind.** to the **Slave spind.** and synchronizes functions such as polygonal turning jobs. The function remains active until you deactivate **G720** with the setting **H0**.

If you would like to synchronize more than two spindles you can program **G720** several times in succession.

Parameters:

- **S**: Number of the **Master spind.**
- **H**: Number of the **Slave spind.** – no input or **H**=0: Switches off the spindle synchronization
- **C**: **Angle** – offset angle
- **Q**: **Master spindle speed factor** (range: –100 <= **Q** <= 100)
- **F**: **Slave spindle speed factor** (range: –100 <= **F** <= 100)
- **Y**: **Type of cycle** (machine-dependent)

Program the speed of the **Master spind.** with **Gx97 S..** and define the speed ratio between the **Master spind.** and **Slave spind.** with **Q** and **F**. If you enter a negative value for **Q** or **F**, the direction of rotation of the **Slave spind.** will be reversed.

Remember that: **Q** * master speed = **F** * slave speed

. . .	
N.. G397 S1500 M3	Spindle speed and direction of rotation of master spindle
N.. G720 C180 S0 H1 Q2 F-1	Synchronization of master spindle and slave spindle. The slave spindle precedes the master spindle by 180°. Slave spindle: Direction of rotation M4; rotational speed 750
N.. G1 X.. Z..	
. . .	

Offset C angle G905

G905 measures the angular offset during workpiece transfer while the spindle is rotating. The sum of **Angle C** and the angle offset goes into effect as the datum shift of the C axis. If you interrogate the datum shift of the current C-axis in the variable **#a0 (C,1)**, the sum of the programmed datum shift and the measured offset angle will be transferred.

The datum shift is effective internally as a direct datum shift for the respective C axis. The contents of the variables are retained even if the control has been switched off.

You can also examine and reset the respectively active datum shift of the C axis in the **Setup** menu using the **Set C-axis values** function.

Parameters:

- **Q**: **No. axis C**
- **C**: **Angle** – angle of additional datum shift for offset gripping (range: –360° <= **C** <= 360°; default: 0°)

NOTICE

Danger of collision!

When switching off the control and when switching the operating mode (e.g., between **Machine** operating mode and **Program run** submode), datum shifts of the C axis are retained. Danger of collision during subsequent machining operations or workpiece transfers!

- Always intentionally switch off datum shifts of the C axis

NOTICE

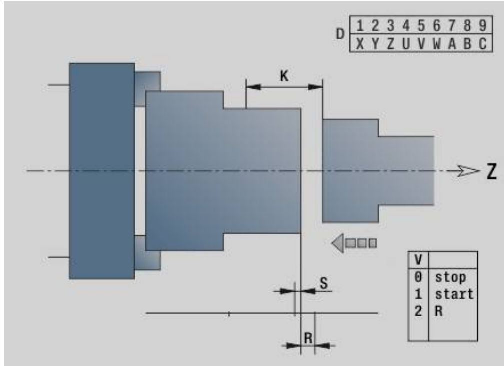
Danger of collision!

The control does not check for jaw collisions during workpiece transfer (e.g. between the main spindle and counter spindle). Danger of collision with short workpieces during transfers!

- Inspect the datum shift of the C axis and reset if required to ensure offset gripping of the jaws

Traverse to a fixed stop G916

Refer to your machine manual.
The machine tool builder determines the scope of function and behavior of this function.



G916 switches on the monitoring of the traverse path and moves to a fixed stop (example: transferring a premachined workpiece to a second traveling spindle if you do not know the exact position of the workpiece).

The control stops the slide and saves the stop position. **G916** generates an interpreter stop.

Parameters:

- **H**: **Contact force** in daN (1 daNewton = 10 newtons)
- **D**: **Axis number** (X = 1, Y = 2, Z = 3, U = 4, V = 5, W = 6, A = 7, B = 8, C = 9)
- **K**: **Incremental distance**
- **R**: **Return path**
- **V**: **Type of departure**
  - **V** = 0: Stay at fixed stop
  - **V** = 1: Return to start position
  - **V** = 2: Retract by return path **R**
- **O**: **Err. evaluation**
  - **O** = 0: Error evaluation in expert program
  - **O** = 1: The control issues an error message

- Lag error monitoring is not activated until the acceleration phase has been completed
- The feed rate override is not effective during cycle execution

When traversing to a fixed stop, the control moves

- up to the fixed stop and stops as soon as the following error has been reached. The remaining path of traverse is deleted
- back to starting position
- back by the return traverse path

Programming:

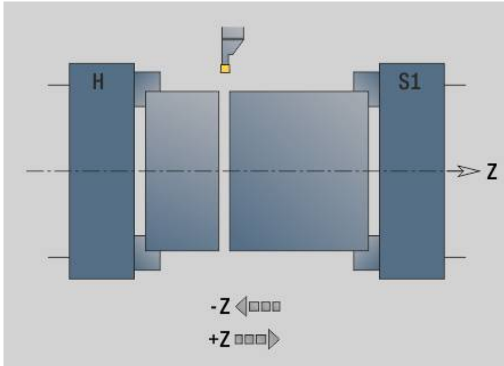
- Position the slide at a sufficient distance before the fixed stop
- Use a moderate feed rate (< 1000 mm/min)

Example: Traversing to a fixed stop

. . .	
N.. G0 Z20	Pre-position slide 2
N.. G916 H100 D6 K-20 V0 O1	Activate monitoring, traverse to a fixed stop
. . .	

Controlled parting using lag error monitoring G917

Refer to your machine manual.
The machine tool builder determines the scope of function and behavior of this function.



G917 monitors the path of traverse. The controlled parting function (cut-off control) prevents collisions caused by incomplete parting processes.

The control stops the slide when the tensile force is too high and generates an interpreter stop.

Parameters:

- **H**: **Tensile force**
- **D**: **Axis number** (X = 1, Y = 2, Z = 3, U = 4, V = 5, W = 6, A = 7, B = 8, C = 9)
- **K**: **Incremental distance**
- **O**: **Err. evaluation**
  - **O** = 0: Error evaluation in expert program
  - **O** = 1: The control issues an error message

During parting control, the parted workpiece moves in the **+Z** direction. If a lag error occurs, the workpiece is defined as not cut off.

The result is saved in the variable **#i99**:

- 0: Workpiece was not correctly cut off (following error detected)
- 1: Workpiece was correctly cut off (no following error detected)

- Lag error monitoring is not activated until the acceleration phase has been completed
- The feed rate override is not effective during cycle execution

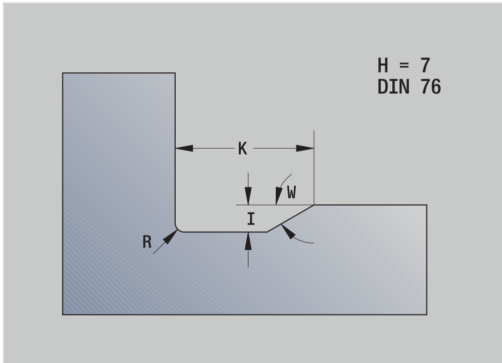
6.37 G codes from previous controls

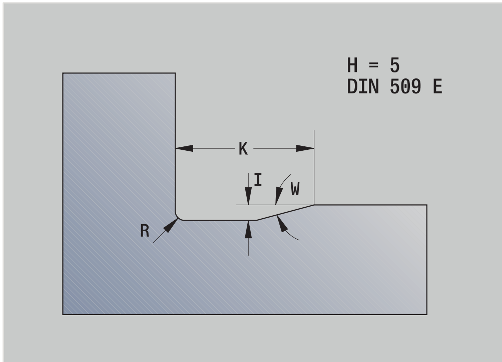
Fundamentals

The commands described in the following are supported to enable you to use NC programs from previous controls. HEIDENHAIN recommends no longer using these commands with new NC programs.

Undercut G25 – contour definitions in the machining section

G25 generates an undercut form element (**DIN 509 E**, **DIN 509 F**, **DIN 76**) that can then be integrated in roughing or finishing cycles. The help graphic illustrates the undercut parameters.





Parameters:

- **H**: **Cut type** (default: 0)
  - 0 or 5: DIN 509 E
  - 6: DIN 509 F
  - 7: DIN 76
- **I**: **Undercut depth** (default: value from standard table)
- **K**: **Undercut width** (default: standard table)
- **R**: **Undercut radius** (default: standard table)
- **P**: **Trav.dpth** (default: value from standard table)
- **W**: **Undercut angle** (default: value from standard table)
- **A**: **Face angle** (default: standard table)
- **FP**: **Thread pitch** (no input: FP is calculated from the thread diameter)
- **U**: **Grind. overs.** (default: 0)
- **E**: **Reduc. Feed** for machining the undercut (default: active feed rate)

If the parameters are not defined, the control determines the following values from the diameter or the thread pitch in the standard table:

- **DIN 509 E**: **I**, **K**, **W**, **R**
- **DIN 509 F**: **I**, **K**, **W**, **R**, **P**, **A**
- **DIN 76**: **I**, **K**, **W**, **R** (determined from the **Thread pitch**)

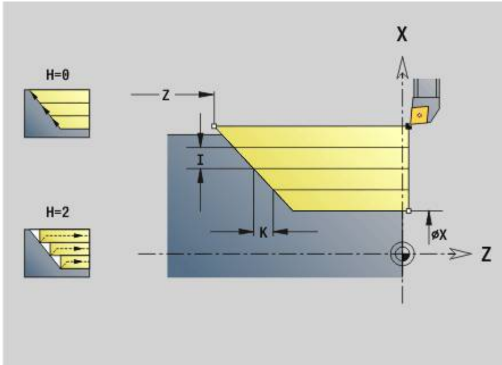
- All parameters that you enter will be accounted for—even if the standard table prescribes other values.
- If you are programming an internal thread, it is advisable to preset the **Thread pitch FP** since the diameter of the longitudinal element is not the thread diameter.
  If you have the control calculate the **Thread pitch** automatically, slight deviations may occur.

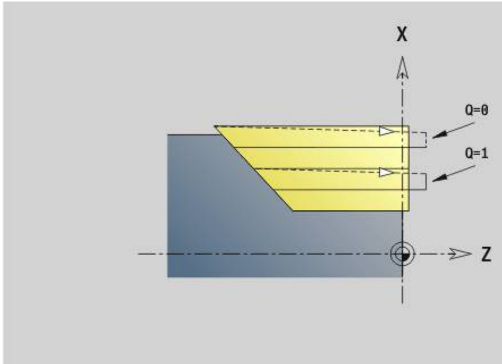
Example: G25

%25.nc	
N1 T1 G95 F0.4 G96 S150 M3	
N2 G0 X62 Z2	
N3 G819 P4 H0 I0.3 K0.1	
N4 G0 X13 Z0	
N5 G1 X16 Z-1.5	
N6 G1 Z-30	
N7 G25 H7 I1.15 K5.2 R0.8 W30 FP1.5	
N8 G1 X20	
N9 G1 X40 Z-35	
N10 G1 Z-55 B4	
N11 G1 X55 B-2	
N12 G1 Z-70	
N13 G1 X60	
N14 G80	
END	

Simple longitudinal roughing G81 – simple turning cycles

G81 roughs the contour area defined by the current tool position and **X**, **Z**. If you wish to machine an oblique cut, you can define the angle with **I** and **K**.





Parameters:

- **X**: Contour **Start point** (diameter value)
- **Z**: **Final point**
- **I**: **Max. approach**
- **K**: **Offset** (in Z; default: 0)
- **Q**: **G-Fct.approach** (default: 0)
  - 0: Infeed with **G0** (rapid traverse)
  - 1: Infeed with **G1** (feed rate)
- **V**: **Type of retraction** (default: 0)
  - 0: Return to cycle starting point in Z and last retraction diameter in X
  - 1: Return to cycle starting point
- **H**: **Contour smoothing**
  - 0: With each cut (machine contour outline after each pass)
  - 2: No smoothing (retracts at 45°; no contour smoothing)

The control uses the position of the target point to distinguish between external and internal machining. The number of cutting passes is calculated in such a way that an abrasive cut is avoided and the calculated **Max. approach** is <= **I**.

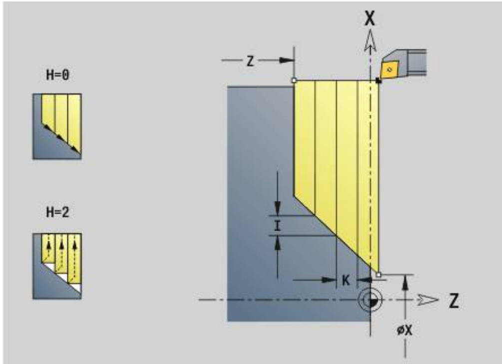
- Programming **X**, **Z**: Absolute, incremental or modal
- Tool radius compensation is inactive.
- Safety clearance after each step: 1 mm
- A **G57** oversize
  - Is calculated with algebraic sign (oversizes are therefore impossible for inside contour machining)
  - Remains effective after cycle end
- A **G58** oversize is not taken into account.

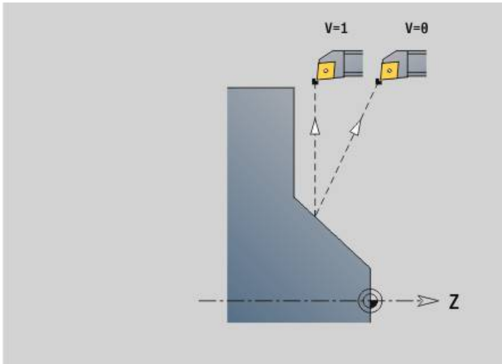
Example: G81

. . .	
N1 T3 G95 F0.25 G96 S200 M3	
N2 G0 X120 Z2	
N3 G81 X100 Z-70 I4 K4 Q0	
N4 G0 X100 Z2	
N5 G81 X80 Z-60 I-4 K2 Q1	
N6 G0 X80 Z2	
N7 G81 X50 Z-45 I4 Q1	
. . .	

Simple face roughing G82 – simple turning cycles

G82 roughs the contour area defined by the current tool position and **X**, **Z**. If you wish to machine an oblique cut, you can define the angle with **I** and **K**.





Parameters:

- **X**: **Final point** (diameter value)
- **Z**: **Start point** Z
- **I**: **Offset** in X (default: 0)
- **K**: **Max. approach**
- **Q**: **G-Fct.approach** (default: 0)
  - 0: Infeed with **G0** (rapid traverse)
  - 1: Infeed with **G1** (feed rate)
- **V**: **Type of retraction** (default: 0)
  - 0: Return to cycle starting point in X and last retraction position in Z
  - 1: Return to cycle starting point
- **H**: **Contour smoothing**
  - 0: With each cut (machine contour outline after each pass)
  - 2: No smoothing (retracts at 45°; no contour smoothing)

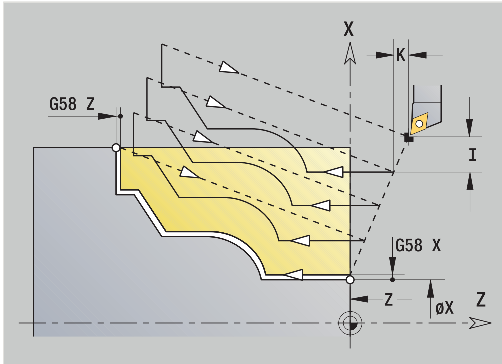
The control uses the position of the target point to distinguish between external and internal machining. The number of cutting passes is calculated in such a way that an abrasive cut is avoided and the calculated **Max. approach** is <= **K**.

- Programming **X**, **Z**: Absolute, incremental or modal
- Tool radius compensation is inactive.
- Safety clearance after each step: 1 mm
- A **G57** oversize
  - Is calculated with algebraic sign (oversizes are therefore impossible for inside contour machining)
  - Remains effective after cycle end
- A **G58** oversize is not taken into account.

Example: G82

```
. . .
N1 T3 G95 F0.25 G96 S200 M3
N2 G0 X120 Z2
N3 G82 X20 Z-15 I4 K4 Q0
N4 G0 X120 Z-15
N5 G82 X50 Z-26 I2 K-4 Q1
N6 G0 X120 Z-26
N7 G82 X80 Z-45 K4 Q1
. . .
```

Simple contour repeat cycle G83 – simple turning cycles



G83 carries out the functions programmed in the following blocks (simple traverses or cycles without a contour definition) more than once. **G80** ends the machining cycle.

Parameters:

- **X**: **Destinat. point** of contour (diameter; default: apply the last X coordinate)
- **Z**: **Destinat. point** of contour (default: apply the last Z coordinate)
- **I**: **Max. approach**
- **K**: **Max. approach**

If the number of infeeds differs for the X and Z axes, the tool first advances in both axes with the programmed values. The infeed is set to zero if the target value for one direction is reached.

Programming:

- **G83** is alone in the block
- **G83** must not be nested, not even by calling subprograms

> - Tool radius compensation is inactive.
> - Safety clearance after each step: 1 mm
> - A **G57** oversize
>   - Is calculated with algebraic sign (oversizes are therefore impossible for inside contour machining)
>   - Remains effective after cycle end
> - A **G58** oversize
>   - Is taken into account if you work with **TRC**
>   - Remains effective after cycle end

NOTICE

Danger of collision!

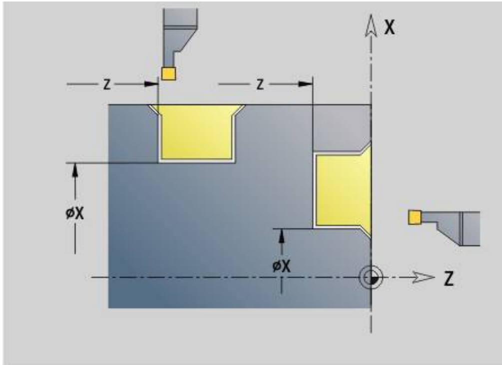
After each cut, the **G83** function pre-positions the tool along the shortest path (diagonally) for the next infeed. Danger of collision during pre-positioning!

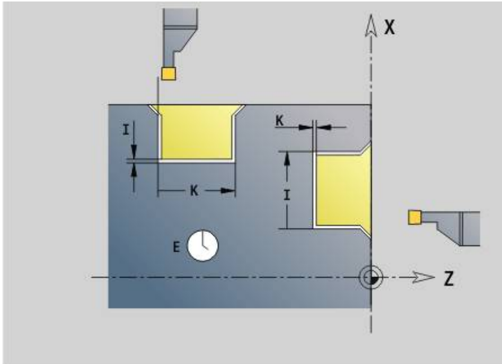
- Test the NC program in **Simulation** submode, using the graphic displayed there.
- Program an additional rapid traverse path to a safe position if required

Example: G83

. . .	
N1 T3 G95 F0.25 G96 S200 M3	
N2 G0 X120 Z2	
N3 G83 X80 Z0 I4 K0.3	
N4 G0 X80 Z0	
N5 G1 Z-15 B-1	
N6 G1 X102 B2	
N7 G1 Z-22	
N8 G1 X90 Zi-12 B1	
N9 G1 Zi-6	
N10 G1 X100 A80 B-1	
N11 G1 Z-47	
N12 G1 X110	
N13 G0 Z2	
N14 G80	

Recessing G86 – simple turning cycle





G86 machines simple radial and axial recesses with chamfers. From the tool position, the control determines whether a radial or axial recess, or an inside or outside recess is to be machined.

Parameters:

- **X**: **Base corner X** (diameter value)
- **Z**: **Base corner Z**
- **I**: Radial recess – **Ov.size** / Axial recess – **Width**
  Radial recess
  - **I** > 0: Oversize (roughing and finishing)
  - **I** = 0: No finishing

  Axial recess:
  - **I** > 0: Recess width
  - No input: Recess width = tool width
- **K**: Radial recess – **Width** / Axial recess – **Ov.size**
  Radial recess
  - **K** > 0: Recess width
  - No input: Recess width = tool width

  Axial recess
  - **K** > 0: Oversize (roughing and finishing)
  - **K** = 0: No finishing
- **E**: **Delay** (default: time for one spindle revolution)
  - With finishing oversize: Only for finishing
  - Without finishing oversize: No finishing

Oversize programmed: First roughing, then finishing

G86 machines chamfers at the sides of the recess. If you do not wish to cut the chamfers, you must position the tool at a sufficient distance from the workpiece.

Calculate the starting position **XS** (diameter) as follows:

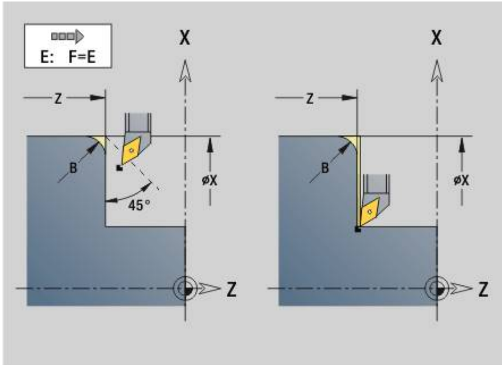
- **XS** = **XK** + 2 * (1.3 – **b**)
- **XK**: Contour diameter
- **b**: Chamfer width

> - The tool radius compensation is active
> - An oversize is not taken into account

Example: G86

. . .	
N1 T3 G95 F0.25 G96 S200 M3	
N2 G0 X62 Z2	
N3 G86 X54 Z-30 I0.2 K7 E2	Radial
N4 G14 Q0	
N5 T38 G95 F0.15 G96 S200 M3	
N6 G0 X120 Z1	
N7 G86 X102 Z-4 I7 K0.2 E1	Axial
. . .	

Radius cycle G87 – simple turning cycles



G87 machines transition radii at orthogonal, paraxial inside and outside corners. The direction is taken from the position of the machining direction of the tool.

Parameters:

- **X**: **Edge** (diameter value)
- **Z**: **Edge**
- **B**: **Radius**
- **E**: **Reduced feed**

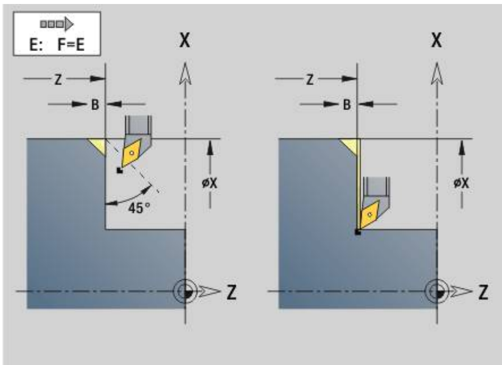
A preceding longitudinal or transverse element is machined if the tool is located at the **X** or **Z** coordinate of the corner before the cycle is executed.

- The tool radius compensation is active
- An oversize is not taken into account

Example: G87

. . .	
N1 T3 G95 F0.25 G96 S200 M3	
N2 G0 X70 Z2	
N3 G1 Z0	
N4 G87 X84 Z0 B2	Radius

Chamfer cycle G88 – simple turning cycles



G88 machines chamfers at orthogonal, paraxial outside corners. The direction is taken from the position of the machining direction of the tool.

Parameters:

- **X**: **Edge** (diameter value)
- **Z**: **Edge**
- **B**: **Cham. width**
- **E**: **Reduced feed**

A preceding longitudinal or transverse element is machined if the tool is located at the **X** or **Z** coordinate of the corner before the cycle is executed.

- The tool radius compensation is active
- An oversize is not taken into account

Example: G88

. . .	
N1 T3 G95 F0.25 G96 S200 M3	
N2 G0 X70 Z2	
N3 G1 Z0	
N4 G88 X84 Z0 B2	Chamfer

Simple longitudinal single-start thread G350 – 4110

G350 cuts a longitudinal thread (internal or external thread). The thread starts at the current tool position and ends at the **Final point Z**.

Parameters:

- **Z**: **Edge** of thread
- **F**: **Thread pitch**
- **U**: **Thread depth**
  - **U** > 0: Internal thread
  - **U** <= 0: External thread (lateral surface or front face)
  - **U** = +999 or –999: Thread depth is calculated
- **I**: **Max. approach** (no input: **I** is calculated from the thread pitch and the thread depth)

Internal or external threads: See algebraic sign of **U**

Handwheel superposition (provided that your machine is equipped accordingly): The superposition is limited to the following range:

- X direction: Depending on the current cutting depth—without exceeding the starting and end points of the thread
- Z direction: Maximal 1 thread groove—without exceeding the starting and end points of the thread

> - **NC stop** becomes effective at the end of a thread cut.
> - The feed rate and spindle speed overrides are not effective during cycle run.
> - Handwheel superimpositioning can be activated with a switch located on the machine operating panel if your machine is equipped accordingly.
> - Feedforward control is switched off.

Simple longitudinal multi-start thread G351 – 4110

G351 machines a single or multi-start longitudinal thread (internal or external thread) with variable pitch. The thread starts at the current tool position and ends at the **Final point Z**.

Parameters:

- **Z**: **Edge** of thread
- **F**: **Thread pitch**
- **U**: **Thread depth**
  - **U** > 0: Internal thread
  - **U** <= 0: External thread (lateral surface or front face)
  - **U** = +999 or –999: Thread depth is calculated
- **I**: **Max. approach** (no input: **I** is calculated from the thread pitch and the thread depth)
- **A**: **Approach ang.** (range: –60° < **A** < 60°; default: 30°)
  - **A** < 0: Infeed on left thread flank
  - **A** > 0: Infeed on right thread flank
- **D**: **No.gears** (default: 1 single-start thread)
- **J**: **Remaining cut depth** (default: 1/100 mm)
- **E**: **Variable gr.** (default: 0)
  Increases/decreases the pitch per revolution by **E**.

Internal or external threads: See algebraic sign of **U**

Number of cutting passes: **I** is used for the first pass. The cutting depth is reduced with each further pass until the remaining cutting depth **J** is reached.

Handwheel superposition (provided that your machine is equipped accordingly): The superposition is limited to the following range:

- X direction: Depending on the current cutting depth—without exceeding the starting and end points of the thread
- Z direction: Maximal 1 thread groove—without exceeding the starting and end points of the thread

- **NC stop** becomes effective at the end of a thread cut.
- The feed rate and spindle speed overrides are not effective during cycle run.
- Handwheel superimpositioning can be activated with a switch located on the machine operating panel if your machine is equipped accordingly.
- Feedforward control is switched off.

6.38 DIN PLUS program example

Example of a subprogram with contour repetitions

Contour repetitions, including saving of the contour

PROGRAM HEAD	
#SLIDE $1	
TURRET 1	
T2 ID "121-55-040.1"	
T3 ID "111-55.080.1"	
T4 ID "161-400.2"	
T8 ID "342-18.0-70"	
T12 ID "112-12-050.1"	
BLANK	
N1 G20 X100 Z120 K1	
FINISHED PART	
N2 G0 X19.2 Z-10	
N3 G1 Z-8.5 BR0.35	
N4 G1 X38 BR3	
N5 G1 Z-3.05 BR0.2	
N6 G1 X42 BR0.5	
N7 G1 Z0 BR0.2	
N8 G1 X66 BR0.5	
N9 G1 Z-10 BR0.5	
N10 G1 X19.2 BR0.5	
MACHINING	
N11 G26 S2500	
N12 G14 Q0	
N13 G702 Q0 H1	Save contour
N14 L"1" V0 Q2	Qx = number of repetitions
N15 M30	
SUBROUTINE “1“	
N16 M108N17 G702 Q1 H1	Load saved contour
N18 G14 Q0	
N19 T8	
N20 G97 S2000 M3	
N21 G95 F0.2	
N22 G0 X0 Z4	
N23 G147 K1	
N24 G74 Z-15 P72 I8 B20 J36 E0.1 K0	
N25 G14 Q0	

N26 T3	
N27 G96 S300 G95 F0.35 M4	
N28 G0 X72 Z2	
N29 G820 NS8 NE8 P2 K0.2 W270 V3	
N30 G14 Q0	
N31 T12	
N32 G96 S250 G95 F0.22	
N33 G810 NS7 NE3 P2 I0.2 K0.1 Z-12 H0 W180 Q0	
N34 G14 Q2	
N35 T2	
N36 G96 S300 G95 F0.08	
N37 G0 X69 Z2	
N38 G47 P1	
N39 G890 NS8 V3 H3 Z-40 D3	
N40 G47 P1	
N41 G890 NS9 V1 H0 Z-40 D1 I74 K0	
N42 G14 Q0	
N43 T12	
N44 G0 X44 Z2	
N45 G890 NS7 NE3	
N46 G14 Q2	
N47 T4	Insert parting tool
N48 G96 S160 G95 F0.18 M4	
N49 G0 X72 Z-14	
N50 G150	Shift reference point to the right of the cutting edge
N51 G1 X60	
N52 G1 X72	
N53 G0 Z-9	
N54 G1 X66 G95 F0.18	
N55 G42	Activate TRC
N56 G1 Z-10 B0.5	
N57 G1 X17	
N58 G0 X72	
N59 G0 X80 Z-10 G40	Deactivate TRC
N60 G14 Q0	
N61 G56 Z-14.4	Incremental datum shift
Return	
END	

6.39 Connection between geometry and machining commands

Turning

Function	Geometry	Machining
Individual elements	■ **G0**..**G3** ■ **G12**/**G13**	■ **Roughing longitudinal G810** ■ **Roughing transversal G820** ■ **Parallel contour G830** ■ **bidirectional G835** (contour-parallel roughing with neutral tool) ■ **Cutting-in universal G860** ■ **Recess turning G869** ■ **Finishing G890**
Recess	■ **G22** (standard)	■ **Cutting-in universal G860** ■ **Recessing cycle G870** ■ **Recess turning G869**
Recess	■ **G23**	■ **Cutting-in universal G860** ■ **Recess turning G869**
Thread with undercut	■ **G24**	■ **Roughing longitudinal G810** ■ **Roughing transversal G820** ■ **Parallel contour G830** ■ **Finishing G890** ■ **Thread turning G31**
Undercut	■ **G25**	■ **Roughing longitudinal G810** ■ **Finishing G890**
Thread	■ **G34** (standard) ■ **G37** (general)	■ **Thread turning G31**
Hole	■ **G49** (turning center)	■ **Simple G71** ■ **G72 Boring, cntrsnkg.** ■ **Tapping G73** ■ **Drilling deep G74**

C-axis machining – front/rear face

Function	Geometry	Machining
Individual elements	■ **G100..G103**	■ **Mill. contour G840** ■ **Pock. - rough milling G845** ■ **Pock. - finish milling G846**
Figures	■ **Linear groove G301** ■ **Circular slot G302/G303** ■ **Complete circle G304** ■ **Rectangle G305** ■ **Polygon G307**	■ **Mill. contour G840** ■ **Pock. - rough milling G845** ■ **Pock. - finish milling G846**
Hole	■ **Bore hole G300**	■ **Simple G71** ■ **G72 Boring, cntrsnkg.** ■ **Tapping G73** ■ **Drilling deep G74**
Engraving	■ **Text for front face C G306**	■ **G801 Engraving**

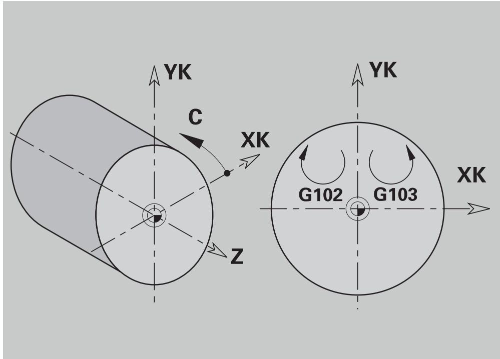
C-axis machining – lateral surface

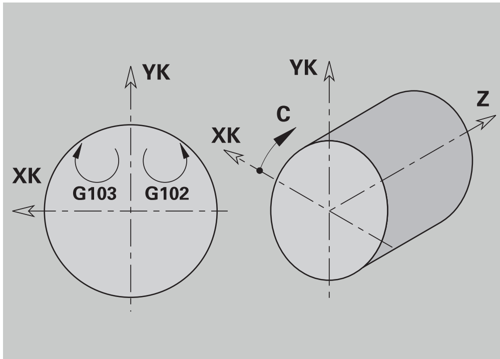
Function	Geometry	Machining
Individual elements	■ **G110..G113**	■ **Mill. contour G840** ■ **Pock. - rough milling G845** ■ **Pock. - finish milling G846**
Figures	■ **Linear groove G311** ■ **Circular slot G312/G313** ■ **Complete circle G314** ■ **Rectangle G315** ■ **Polygon G317**	■ **Mill. contour G840** ■ **Pock. - rough milling G845** ■ **Pock. - finish milling G846**
Hole	■ **Boring G310**	■ **Simple G71** ■ **G72 Boring, cntrsnkg.** ■ **Tapping G73** ■ **Drilling deep G74**
Engraving	■ **Text for lateral face C G316**	■ **G802 Engraving**

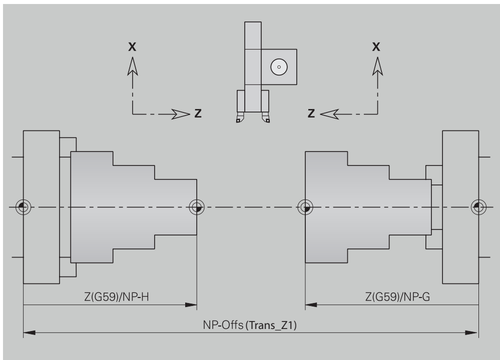
6.40 Full-surface machining

Fundamentals of full-surface machining

In "full-surface machining," the front and rear ends can be machined in one NC program. The control supports full-surface machining for all common machine designs. The features include angle-synchronous part transfer with rotating spindle, traversing to a stop, controlled parting, and coordinate transformation. This ensures efficient full-surface machining and simple programming.







You describe the turning contour, the contours for the C axis as well as full-surface machining functions in one NC program. Expert programs are available for configuring the lathe.

You can also enjoy the benefits of full-surface machining on lathes with only one spindle.

Rear-face contours with C axis: The XK axis and therefore also the C axis are oriented with respect to the workpiece, not to the spindle.

Therefore, for the rear face:

- Orientation of the XK axis: To the left (front face: to the right)
- Orientation of the C axis: Clockwise
- Direction of rotation for circular arcs **G102**: Counterclockwise
- Direction of rotation for circular arcs **G103**: Clockwise

Turning: The control supports full-surface machining with conversion and mirroring functions.

This makes it possible to keep the usual directions of movement for rear-side machining as well.

- Movements in a positive (+) direction depart the workpiece.
- Movements in a negative (–) direction approach the workpiece

The machine manufacturer can provide your lathe with suitable expert programs for workpiece transfer.

Reference points and coordinate system: The position of the machine and workpiece datums as well as the coordinate systems for the spindle and opposing spindle are illustrated in the figure at bottom. With this design of lathe it is recommended to mirror only the Z axis. Then, for either spindle, moving in positive direction will stand for motion away from the workpiece.

Usually the expert program contains the mirroring of the Z axis and the datum shift by the dimension **NP-Offs**.

Programming of full-surface machining

When programming a contour on the rear face, be sure to consider the orientation of the XK axis (or X axis) and rotational direction of arcs.

Insofar as you use drilling and milling cycles, there are no special aspects to rear-face machining, since these cycles refer to predefined contours.

For rear-face machining with the basic commands **G100** to **G103** the same conditions apply as for rear-face contours.

Turning operations: The expert programs for reclamping contain converting and mirroring functions.

The following principle applies for rear-face machining (2nd setup):

- + direction: Goes away from the workpiece
- – direction: Goes toward the workpiece
- **G2** and **G12**: Circular arc clockwise
- **G3** and **G13**: Circular arc counterclockwise

Working without expert programs: If you do not use the converting and mirroring functions, the following principle applies:

- + direction: Goes away from the main spindle
- – direction: Goes toward the main spindle
- **G2** and **G12**: Circular arc clockwise
- **G3** and **G13**: Circular arc counterclockwise

Full-surface machining with counter spindle

G30: The expert program switches the kinematics of the counterspindle. In addition, **G30** activates the mirroring of the Z axis and converts other functions (e.g. circular arcs **G2**, **G3**).

G99: The expert program shifts the contour and mirrors the coordinate system (Z axis). Further programming of **G99** is normally not required for machining the rear face after rechucking.

Example: The workpiece is machined on the front face, transferred to the counter spindle through an expert program and machined on the rear face.

The expert program is used for:

- Angle-synchronous workpiece transfer to the counter spindle
- Mirroring traverse paths in the Z axis
- Activating a conversion list
- Mirroring the contour description and shifting for the 2nd setup

Full-surface machining on machines with counter spindles

PROGRAM HEAD		
#MATERIAL	STEEL	
#UNIT	METRIC	
TURRET		
T1 ID "512-600.10"		
T2 ID "111-80-080.1"		
T102 ID "115-80-080.1"		
BLANK		
N1 G20 X100 Z100 K1		
FINISHED PART		
. . .		
FRONT Z0		
N13 G308 ID"Line" P-1		
N14 G100 XK-15 YK10		
N15 G101 XK-10 YK12 BR2		
N16 G101 XK-4.0725 YK-12.6555 BR4		
N18 G101 XK10		
N19 G309		
REAR SIDE Z-98		
. . .		
MACHINING		
N27 G59 Z233		Datum shift for 1st setup
N28 G0 W#iS18		Counterspindle to machining position
N30 G14 Q0		
N31 G26 S2500		
N32 T2		
. . .		

N63 M5	
N64 T1	
N65 G197 S1485 G193 F0.05 M103	C-axis machining in the main spindle
N66 M14	
N67 M107	
N68 G0 X36.0555 Z3	
N69 G110 C146.31	
N70 G147 I2 K2	
N71 G840 Q0 NS15 NE18 I0.5 R0 P1	
N72 G0 X31.241 Z3	
N73 G14 Q0	
N74 M105 M109	
N76 M15	Deactivate C axis
N80 L"RECHUCK" V1 LA.. LB.. LC..	Expert prog. for part transfer with following functions: G720 Spindle synchronization G916 Traversing to a fixed stop G30 Switch the kinematics G99 Mirroring and shifting of the workpiece contour
N90 G59 Z222	Datum shift for 2nd setup
. . .	
N91 G14 Q0	
N92 T102	
N93 G396 S220 G395 F0.2 M304	Technology data for opposing spindle
N94 M107	Turning in the counterspindle
N95 G0 X120 Z3	
N96 G810	Machining cycles
N97 G30 Q0	Switch off rear-face machining
. . .	
N129 M30	
END	

Full-surface machining with single spindle

G30: Normally not required.

G99: The expert program mirrors the contour. Further programming of **G99** is normally not required for machining the rear face after rechucking.

Example: Describes the machining of the front and rear face, using one NC program. The workpiece is first machined on the front face; then it is rechucked manually. The rear face is machined subsequently.

The expert program mirrors and shifts the contour for the 2nd setup.

Full-surface machining on machine with one spindle

PROGRAM HEAD	
#MATERIAL STEEL	
#UNIT METRIC	
TURRET	
T1 ID "512-600.10"	
T2 ID "111-80-080.1"	
T102 ID "115-80-080.1"	
BLANK	
N1 G20 X100 Z100 K1	
FINISHED PART	
. . .	
FRONT Z0	
. . .	
REAR SIDE Z-98	
. . .	
N20 G308 ID"R" P-1	
N21 G100 XK5 YK-10	
N22 G101 YK15	
N23 G101 XK-5	
N24 G103 XK-8 YK3.8038 R6 I-5	
N25 G101 XK-12 YK-10	
N26 G309	
MACHINING	
N27 G59 Z233	Datum shift for 1st setup
. . .	
N82 M15	Prepare the rechucking
N86 G99 H1 V0 K-98	Contour mirroring and shifting for manual rechucking
N87 M0	Stop for rechucking
N88 G59 Z222	Datum shift for 2nd setup
. . .	

N125 M5	Milling – rear face
N126 T1	
N127 G197 S1485 G193 F0.05 M103	
N128 M14	
N130 M107	
N131 G0 X22.3607 Z3	
N132 G110 C-116.565	
N134 G147 I2 K2	
N135 G840 Q0 NS22 NE25 I0.5 R0 P1	
N136 G0 X154 Z-95	
N137 G0 X154 Z3	
N138 G14 Q0	
N139 M105 M109	
N142 M15	
N143 G30 Q0	Switch off rear-face machining
N144 M30	
END	

6.41 Program templates

Fundamentals

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

A program template is a predefined NC program. It defines the structure for complex programming, for example. This reduces the programming efforts.

Your machine tool builder can provide you with up to nine program templates.

Opening a program template

In order to use the program templates defined by the machine tool builder, create a new NC program from the template in **smart.Turn** operating mode.

Proceed as follows:

- Select the **Prog** menu item
- Select the **New** menu item
- Select the **New program from template** menu item
- Select the desired template

7

Touch Probe Cycles

7.1 General information on touch probe cycles (option 17)

Fundamentals

Refer to your machine manual.

The control must be specially prepared by the machine tool builder for the use of a 3-D touch probe.

HEIDENHAIN gives a warranty for the function of the touch probe cycles only if HEIDENHAIN touch probes are used!

Principle of function of touch probe cycles

When you run a touch probe cycle, the 3-D touch probe is pre-positioned at positioning feed rate. The actual probing movement is then executed from there at probing feed rate. The machine tool builder determines the positioning feed rate for the touch probe in a machine parameter. You define the probing feed rate in the respective touch probe cycle.

When the probe stylus contacts the workpiece,

- the 3-D touch probe transmits a signal to the control: the coordinates of the probed position are stored,
- the touch probe stops moving, and
- returns to the starting position of the probing procedure at positioning feed rate.

If the stylus is not deflected within a defined distance, the control displays an error message.

Touch probe cycles for automatic mode

The control provides numerous touch probe cycles for various applications:

- 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
- Datum setting in the Z or C axis
- Automatic tool measurement

Touch probe cycles are programmed in the **smart.Turn** operating mode using the **G** codes. Just like the machining cycles, the touch probe cycles also use transfer parameters.

To simplify programming, the control shows a graphic during cycle definition. The appropriate input parameters are displayed in the help graphic.

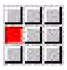
The touch probe cycles save status information and measuring results in the variable **#i99**.

Depending on the input parameters in the touch probe cycle you can interrogate various values.

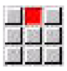
Result #i99	Meaning
< 999997	Measuring result
999999	Touch probe not deflected
-999999	Invalid measuring axis programmed
999998	**Max. deviation WE** exceeded
999997	**Maximum perm. correction E** exceeded

Programming the touch probe cycle in **ISO Mode**:

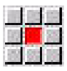
- Select **ISO Mode** programming and place the cursor in the **MACHINING** program section



- Select the **Mach»** menu item



- Select the **G-menu** menu item



- Select the **Touch probe cycl.** menu item
- Select measuring cycle group
- Select the cycle

Example: Touch probe cycle in the DIN PLUS program

PROGRAM HEAD		
#MATERIAL	STEEL	
#UNIT	METRIC	
TURRET		
1T1 ID"342-300.1"		
T2 ID"111-80-080.1"		
. . .		
BLANK		
N1 G20 X120 Z120 K2		
FINISHED PART		
N2 G0 X60 Z-115		
N3 G1 Z-105		
. . .		
MACHINING		
N18 T1		
N19 G0 X0 Z5		
N20 G771 R1 D0 K-30 AC0 BD2 Q0 P0 H0		
N21 T2 G97 S1000 G95 F0.2 M3		
N22 G0 X0 Z5		
N23 G71 Z-25 A5 V2		Drilling
. . .		
END		

Measuring cycle group	Page
Single-point meas.	Page 581
Double-point meas.	Page 589
Calibration	Page 597
Probing	Page 601
Search cycles	Page 608
Circular measurement	Page 616
Angular measurement	Page 620
In-process measrmnt.	Page 623

7.2 Touch probe cycles for single-point measurement

Single-point meas. for tool comp. G770

Cycle **G770** measures with the programmed measuring axis in the specified direction. If the tolerance value defined in the cycle is exceeded, the cycle saves the measured deviation either as tool compensation or as an additive compensation. The result of the measurement is saved additionally in the variable **#i99**.

Further information: "Touch probe cycles for automatic mode", Page 579

Cycle run: From the current position the touch probe moves along the defined measuring axis toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point.

The control displays an error message if the touch probe does not reach any touch point within the defined measuring path. If a **Max. deviation WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the **Max. deviation WE**, the program run is interrupted and an error message is displayed.

Parameters:

- **R**: **Type of correction**
  - 1: Tool compensation **DX**/**DZ** for turning tool or additive compensation
  - 2: Recessing tool **Dx**/**DS**
  - 4: Milling tool **DD**
- **D**: **Measuring axis** – axis in which the measurement is to be made
- **K**: **Incr. meas path w/ Ri.** (the algebraic sign determines the probing direction) – maximum measuring path for probing
- **AC**: **Target pos. nominal value** – touch point coordinate
- **BD**: **Tolerance position +/-** – measurement result range in which no compensation will be applied
- **WT**: **Correction no. T or G149**
  - **T**: Tool at turret position **T** to compensate the difference to the nominal value
  - **G149**: Additive compensation **D9xx** to correct the difference to the nominal value (only possible with compensation type **R** = 1)
- **E**: **Maximum perm. correction** for the tool compensation
- **WE**: **Max. deviation** – probe twice and monitor the dispersion of the measured values
- **V**: **Retraction type**
  - 0: Without—only position touch probe back to the starting point if the touch probe was deflected
  - 1: Automatic—always position touch probe back to the starting point

- **O**: **Err. evaluation**
  - 0: Program—do not interrupt program run, no error message
  - 1: Automatic—interrupt program run and output error message if touch probe is not deflected within measuring path
- **F**: **Measuring feed** – feed rate for probing (if nothing is entered, the measuring feed rate from the touch probe table will be used)
  If you enter a measuring feed rate **F** that is higher than the one in the touch probe table, the feed rate will be reduced to the value from the touch probe table.
- **Q**: **Tool orientation** (machine-dependent)
  Orient the touch probe in the programmed probing direction before each probing operation.
- **P**: **PRINT outputs**
  - **0: OFF** – do not display measuring results
  - **1: ON** – display measuring results on the screen
- **H**: **INPUT instead of measure**
  - **0: Standard** – obtain measured values by probing
  - **1: PC test** – simulate the touch probe cycle at the programming station
- **AN**: **Log no.** – save measurement results in the **TNC:\table\messpro.mep** table (line numbers: 0 to 99)
  The table can be extended if necessary.

Example: G770 Single-point meas. for tool comp.

. . .	
MACHINING	
N3 G770 R1 D0 K20 AC0 BD0.2 WT3 V1 O1 Q0P0 H0	
. . .	

Single-point measurmnt., datum G771

Cycle **G771** measures with the programmed measuring axis in the specified direction. If the tolerance value defined in the cycle is exceeded, the cycle saves the measured deviation as datum shift. The result of the measurement is saved additionally in the variable **#i99**.

Further information: "Touch probe cycles for automatic mode", Page 579

Cycle run: From the current position the touch probe moves along the defined measuring axis toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point.

The control displays an error message if the touch probe does not reach any touch point within the defined measuring path. If a **Max. deviation WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the **Max. deviation WE**, the program run is interrupted and an error message is displayed.

Parameters:

- **R**: **Type of datum shift**
  - 1: Table and **G59** – activate datum shift and additionally save in datum table (the datum shift also remains active after the program run)
  - 2: Activate datum shift with **G59** for the further program run (datum shift no longer active after program run)
- **D**: **Measuring axis** – axis in which the measurement is to be made
- **K**: **Incr. meas path w/ Ri.** (the algebraic sign determines the probing direction) – maximum measuring path for probing
- **AC**: **Target pos. nominal value** – touch point coordinate
- **BD**: **Tolerance position +/-** – measurement result range in which no compensation will be applied
- **WE**: **Max. deviation** – probe twice and monitor the dispersion of the measured values
- **F**: **Measuring feed** – feed rate for probing (if nothing is entered, the measuring feed rate from the touch probe table will be used) If you enter a measuring feed rate **F** that is higher than the one in the touch probe table, the feed rate will be reduced to the value from the touch probe table.
- **Q**: **Tool orientation** (machine-dependent) Orient the touch probe in the programmed probing direction before each probing operation.
- **P**: **PRINT outputs**
  - **0: OFF** – do not display measuring results
  - **1: ON** – display measuring results on the screen

- **H**: **INPUT instead of measure**
  - **0: Standard** – obtain measured values by probing
  - **1: PC test** – simulate the touch probe cycle at the programming station
- **AN**: **Log no.** – save measurement results in the **TNC:\table\messpro.mep** table (line numbers: 0 to 99) The table can be extended if necessary.

Example: G771 Single-point measurmnt., datum

. . .	
MACHINING	
N3 G771 R1 D0 K20 AC0 BD0.2 Q0 P0 H0	
. . .	

Datum, single point w/ C axis G772

Cycle **G772** measures with the C axis in the specified direction. If the tolerance value defined in the cycle is exceeded, the cycle saves the measured deviation as datum shift. The result of the measurement is saved additionally in the variable **#i99**.

Further information: "Touch probe cycles for automatic mode", Page 579

Cycle run: From the current position, the element to be probed is moved toward the touch probe by a rotation of the C axis. When the workpiece touches the stylus, the measured value is saved and the workpiece is returned.

The control displays an error message if the touch probe does not reach any touch point within the defined measuring path. If a **Max. deviation WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the **Max. deviation WE**, the program run is interrupted and an error message is displayed.

Parameters:

- **R**: **Type of datum shift**
  - 1: Table and **G152** – activate datum shift and additionally save in datum table (the datum shift also remains active after the program run)
  - 2: Activate datum shift with **G152** for the further program run (datum shift no longer active after program run)
- **C**: **Incr. meas path w/ Ri.** (the algebraic sign determines the probing direction) – C-axis measuring path (in degrees) starting from the current position
- **AC**: **Target pos. nominal value** – absolute coordinate of the touch point in degrees
- **BD**: **Tolerance position +/-** – measurement result range in which no compensation will be applied
- **WE**: **Max. deviation** – probe twice and monitor the dispersion of the measured values
- **F**: **Measuring feed** – feed rate for probing (if nothing is entered, the measuring feed rate from the touch probe table will be used) If you enter a measuring feed rate **F** that is higher than the one in the touch probe table, the feed rate will be reduced to the value from the touch probe table.
- **Q**: **Tool orientation** (machine-dependent) Orient the touch probe in the programmed probing direction before each probing operation.
- **P**: **PRINT outputs**
  - **0: OFF** – do not display measuring results
  - **1: ON** – display measuring results on the screen

- **H**: **INPUT instead of measure**
  - **0: Standard** – obtain measured values by probing
  - **1: PC test** – simulate the touch probe cycle at the programming station
- **AN**: **Log no.** – save measurement results in the **TNC:\table\messpro.mep** table (line numbers: 0 to 99) The table can be extended if necessary.

Example: G772 Single-point measurement datum C axis

. . .	
MACHINING	
N3 G772 R1 C20 AC0 BD0.2 Q0 P0 H0	
. . .	

Datum, C-axis object center G773

Cycle **G773** measures an element with the C axis from two opposite sides and places the center of the element to a defined position. The result of the measurement is saved additionally in the variable **#i99**.

Further information: "Touch probe cycles for automatic mode", Page 579

Cycle run: From the current position, the element to be probed is moved toward the touch probe by a rotation of the C axis. When the workpiece touches the stylus, the measured value is saved and the workpiece is returned. Then the touch probe is pre-positioned for the opposite probing procedure. When the second measured value has been determined, the cycle computes the mean value of the two measurements and applies a datum shift in the C axis. The **Target pos. nominal value AC** defined in the cycle is then in the center of the probed element.

The control displays an error message if the touch probe does not reach any touch point within the defined measuring path. If a **Max. deviation WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the **Max. deviation WE**, the program run is interrupted and an error message is displayed.

Parameters:

- **R**: **Type of datum shift**
  - 1: Table and **G152** – activate datum shift and additionally save in datum table (the datum shift also remains active after the program run)
  - 2: Activate datum shift with **G152** for the further program run (datum shift no longer active after program run)
- **C**: **Incr. meas path w/ Ri.** (the algebraic sign determines the probing direction) – C-axis measuring path (in degrees) starting from the current position
- **E**: **Circumnavigation axis** – axis that will be retracted by **RB** in order to circumnavigate the element
- **RB**: **Circumnav. dir. offset** – retraction value in the circumnavigation axis **E** for pre-positioning for the next probing position
- **RC**: **Offset C angle** – difference in the C axis between the first and the second measuring positions
- **AC**: **Target pos. nominal value** – absolute coordinate of the touch point in degrees
- **BD**: **Tolerance position +/-** – measurement result range in which no compensation will be applied
- **KC**: **Compensation offset** – additional compensation value that is applied to the resulting datum
- **WE**: **Max. deviation** – probe twice and monitor the dispersion of the measured values

- **F**: **Measuring feed** – feed rate for probing (if nothing is entered, the measuring feed rate from the touch probe table will be used)
  If you enter a measuring feed rate **F** that is higher than the one in the touch probe table, the feed rate will be reduced to the value from the touch probe table.
- **Q**: **Tool orientation** (machine-dependent)
  Orient the touch probe in the programmed probing direction before each probing operation.
- **P**: **PRINT outputs**
  - **0: OFF** – do not display measuring results
  - **1: ON** – display measuring results on the screen
- **H**: **INPUT instead of measure**
  - **0: Standard** – obtain measured values by probing
  - **1: PC test** – simulate the touch probe cycle at the programming station
- **AN**: **Log no.** – save measurement results in the **TNC:\table\messpro.mep** table (line numbers: 0 to 99)
  The table can be extended if necessary.

Example: G773 Single-point measurement C-axis object center

. . .	
MACHINING	
N3 G773 R1 C20 E0 RB20 RC45 AC30 BD0.2 Q0P0 H0	
. . .	

7.3 Touch probe cycles for two-point measurement

Two-point meas. G18 transverse G775

Cycle **G775** measures two opposite points in the X/Z plane with the measuring axis X. If the tolerance values defined in the cycle are exceeded, the cycle saves the measured deviation either as tool compensation or as an additive compensation. The result of the measurement is saved additionally in the variable **#i99**.

Further information: "Touch probe cycles for automatic mode", Page 579

Cycle run: From the current position, the touch probe moves along the defined measuring axis toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe returns to the starting point. For the pre-positioning for the second measurement, the cycle first moves the touch probe by the **Circumnav. dir. offset RB** and then by the **Measuring direction offset RC**. The cycle executes the second probing operation in the opposite direction, saves the result and retracts the touch probe in the circumnavigation axis by the circumnavigation value.

The control displays an error message if the touch probe does not reach any touch point within the defined measuring path. If a **Max. deviation WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the **Max. deviation WE**, the program run is interrupted and an error message is displayed.

Parameters:

- **R**: **Type of correction**
  - 1: Tool compensation **DX**/**DZ** for turning tool or additive compensation
  - 2: Recessing tool **DX**/**DS**
  - 3: Milling tool **DX**/**DD**
  - 4: Milling tool **DD**
- **K**: **Incr. meas path w/ Ri.** (the algebraic sign determines the probing direction) – maximum measuring path for probing
- **E**: **Circumnavigation axis** – selection of axis for retraction movement between the probing positions
  - 0: Z axis
  - 2: Y axis
- **RB**: **Circumnav. dir. offset** – distance
- **RC**: **Offset in X** – distance for pre-positioning before the second measurement
- **XE**: **Target pos. noml value X** – absolute coordinate of the touch point
- **BD**: **Tolerance position +/-** – measurement result range in which no compensation will be applied
- **X**: **Nominal width in X** – coordinate of the second probing position
- **BE**: **Tolerance width +/-** – range for the second measurement result in which no compensation will be applied
- **WT**: **Compens. no.1, measd. edge**

- **T**: Tool at turret position **T** to compensate the difference to the nominal value
- **G149**: Additive compensation **D9xx** to correct the difference to the nominal value (only possible with compensation type **R** = 1)

- **AT**: **Compens. no.2, measd. edge**
  - **T**: Tool at turret position **T** to compensate the difference to the nominal value
  - **G149**: Additive compensation **D9xx** to correct the difference to the nominal value (only possible with compensation type **R** = 1)
- **FP**: **Maximum perm. correction**
- **WE**: **Max. deviation** – probe twice and monitor the dispersion of the measured values
- **F**: **Measuring feed** – feed rate for probing (if nothing is entered, the measuring feed rate from the touch probe table will be used)
  If you enter a measuring feed rate **F** that is higher than the one in the touch probe table, the feed rate will be reduced to the value from the touch probe table.
- **Q**: **Tool orientation** (machine-dependent)
  Orient the touch probe in the programmed probing direction before each probing operation.
- **P**: **PRINT outputs**
  - **0: OFF** – do not display measuring results
  - **1: ON** – display measuring results on the screen
- **H**: **INPUT instead of measure**
  - **0: Standard** – obtain measured values by probing
  - **1: PC test** – simulate the touch probe cycle at the programming station
- **AN**: **Log no.** – save measurement results in the **TNC:\table\messpro.mep** table (line numbers: 0 to 99)
  The table can be extended if necessary.

> The cycle computes the **Compens. no.1, measd. edge WT** from the result of the first measurement and the **Compens. no.2, measd. edge AT** from the result of the second measurement.

Example: G775 Two-point measurement for tool compensation

...	
MACHINING	
N3 G775 R1 K20 E1 XE30 BD0.2 X40 BE0.3WT5 Q0 P0 H0	
...	

Two-pt. meas. G18 longitudinal G776

Cycle **G776** measures two opposite points in the X/Z plane with the measuring axis Z. If the tolerance values defined in the cycle are exceeded, the cycle saves the measured deviation either as tool compensation or as an additive compensation. The result of the measurement is saved additionally in the variable **#i99**.

Further information: "Touch probe cycles for automatic mode", Page 579

Cycle run: From the current position, the touch probe moves along the defined measuring axis toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe returns to the starting point. For the pre-positioning for the second measurement, the control first moves the touch probe by the **Circumnav. dir. offset RB** and then by the **Offset in Z RC**. The cycle executes the second probing operation in the opposite direction, saves the result and retracts the touch probe in the circumnavigation axis by the circumnavigation value.

The control displays an error message if the touch probe does not reach any touch point within the defined measuring path. If a **Max. deviation WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the **Max. deviation WE**, the program run is interrupted and an error message is displayed.

Parameters:

- **R**: **Type of correction**
  - 1: Tool compensation **DX**/**DZ** for turning tool or additive compensation
  - 2: Recessing tool **DX**/**DS**
  - 3: Milling tool **DX**/**DD**
  - 4: Milling tool **DD**
- **K**: **Incr. meas path w/ Ri.** (the algebraic sign determines the probing direction) – maximum measuring path for probing
- **E**: **Circumnavigation axis** – selection of axis for retraction movement between the probing positions
  - 0: X axis
  - 2: Y axis
- **RB**: **Circumnav. dir. offset** – distance
- **RC**: **Offset in Z** – distance for pre-positioning before the second measurement
- **ZE**: **Target pos. noml value Z** – absolute coordinate of touch point
- **BD**: **Tolerance position +/-** – measurement result range in which no compensation will be applied
- **Z**: **Nominal width in Z** – coordinate of the second probing position
- **BE**: **Tolerance width +/-** – range for the second measurement result in which no compensation will be applied

- **WT**: **Compens. no.1, measd. edge**
  - **T**: Tool at turret position **T** to compensate the difference to the nominal value
  - **G149**: Additive compensation **D9xx** to correct the difference to the nominal value (only possible with compensation type **R** = 1)
- **AT**: **Compens. no.2, measd. edge**
  - **T**: Tool at turret position **T** to compensate the difference to the nominal value
  - **G149**: Additive compensation **D9xx** to correct the difference to the nominal value (only possible with compensation type **R** = 1)
- **FP**: **Maximum perm. correction**
- **WE**: **Max. deviation** – probe twice and monitor the dispersion of the measured values
- **F**: **Measuring feed** – feed rate for probing (if nothing is entered, the measuring feed rate from the touch probe table will be used) If you enter a measuring feed rate **F** that is higher than the one in the touch probe table, the feed rate will be reduced to the value from the touch probe table.
- **Q**: **Tool orientation** (machine-dependent) Orient the touch probe in the programmed probing direction before each probing operation.
- **P**: **PRINT outputs**
  - **0: OFF** – do not display measuring results
  - **1: ON** – display measuring results on the screen
- **H**: **INPUT instead of measure**
  - **0: Standard** – obtain measured values by probing
  - **1: PC test** – simulate the touch probe cycle at the programming station
- **AN**: **Log no.** – save measurement results in the **TNC:\table\messpro.mep** table (line numbers: 0 to 99) The table can be extended if necessary.

> The cycle computes the **Compens. no.1, measd. edge WT** from the result of the first measurement and the **Compens. no.2, measd. edge AT** from the result of the second measurement.

Example: G776 Two-point measurement for tool compensation

...	
MACHINING	
N3 G776 R1 K20 E1 ZE30 BD0.2 Z40 BE0.3WT5 Q0 P0 H0	
...	

Two-point measurement G17 G777

Cycle **G777** measures two opposite points in the X/Y plane with the measuring axis Y. If the tolerance values defined in the cycle are exceeded, the cycle saves the measured deviation either as tool compensation or as an additive compensation. The result of the measurement is saved additionally in the variable **#i99**.

Further information: "Touch probe cycles for automatic mode", Page 579

Cycle run: From the current position, the touch probe moves along the defined measuring axis toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe returns to the starting point. For the pre-positioning for the second measurement, the cycle first moves the touch probe by the **Circumnav. dir. offset Zi RB** and then by the **Offset in Yi RC**. The cycle executes the second probing operation in the opposite direction, saves the result and retracts the touch probe in the circumnavigation axis by the circumnavigation value.

The control displays an error message if the touch probe does not reach any touch point within the defined measuring path. If a **Max. deviation WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the **Max. deviation WE**, the program run is interrupted and an error message is displayed.

Parameters:

- **R**: **Type of correction**
  - 1: Tool compensation **DX**/**DZ** for turning tool or additive compensation
  - 2: Recessing tool **DX**/**DS**
  - 3: Milling tool **DX**/**DD**
  - 4: Milling tool **DD**
- **K**: **Incr. meas path w/ Ri.** (the algebraic sign determines the probing direction) – maximum measuring path for probing
- **RB**: **Circumnav. dir. offset Zi** – distance
- **RC**: **Offset in Yi** – distance for pre-positioning before the second measurement
- **YE**: **Target pos. noml value Y** – absolute coordinate of touch point
- **BD**: **Tolerance position +/-** – measurement result range in which no compensation will be applied
- **Y**: **Nominal width in Y** – coordinate of the second probing position
- **BE**: **Tolerance width +/-** – range for the second measurement result in which no compensation will be applied
- **WT**: **Compens. no.1, measd. edge**
  - **T**: Tool at turret position **T** to compensate the difference to the nominal value
  - **G149**: Additive compensation **D9xx** to correct the difference to the nominal value (only possible with compensation type **R** = 1)

- **AT**: **Compens. no.2, measd. edge**
  - **T**: Tool at turret position **T** to compensate the difference to the nominal value
  - **G149**: Additive compensation **D9xx** to correct the difference to the nominal value (only possible with compensation type **R** = 1)
- **FP**: **Maximum perm. correction**
- **WE**: **Max. deviation** – probe twice and monitor the dispersion of the measured values
- **F**: **Measuring feed** – feed rate for probing (if nothing is entered, the measuring feed rate from the touch probe table will be used)
  If you enter a measuring feed rate **F** that is higher than the one in the touch probe table, the feed rate will be reduced to the value from the touch probe table.
- **Q**: **Tool orientation** (machine-dependent)
  Orient the touch probe in the programmed probing direction before each probing operation.
- **P**: **PRINT outputs**
  - **0: OFF** – do not display measuring results
  - **1: ON** – display measuring results on the screen
- **H**: **INPUT instead of measure**
  - **0: Standard** – obtain measured values by probing
  - **1: PC test** – simulate the touch probe cycle at the programming station
- **AN**: **Log no.** – save measurement results in the **TNC:\table\messpro.mep** table (line numbers: 0 to 99)
  The table can be extended if necessary.

> The cycle computes the **Compens. no.1, measd. edge WT** from the result of the first measurement and the **Compens. no.2, measd. edge AT** from the result of the second measurement.

Example: G777 Two-point measurement for tool compensation

...	
MACHINING	
N3 G777 R1 K20 YE10 BD0.2 Y40 BE0.3 WT5Q0 P0 H0	
...	

Two-point measurement G19 G778

Cycle **G778** measures two opposite points in the Y/Z plane with the measuring axis Y. If the tolerance values defined in the cycle are exceeded, the cycle saves the measured deviation either as tool compensation or as an additive compensation. The result of the measurement is saved additionally in the variable **#i99**.

Further information: "Touch probe cycles for automatic mode", Page 579

Cycle run: From the current position, the touch probe moves along the defined measuring axis toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe returns to the starting point. For the pre-positioning for the second measurement, the cycle first moves the touch probe by the **Circumnav. dir. offset Xi RB** and then by the **Offset in Yi RC**. The cycle executes the second probing operation in the opposite direction, saves the result and retracts the touch probe in the circumnavigation axis by the circumnavigation value.

The control displays an error message if the touch probe does not reach any touch point within the defined measuring path. If a **Max. deviation WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the **Max. deviation WE**, the program run is interrupted and an error message is displayed.

Parameters:

- **R**: **Type of correction**
  - 1: Tool compensation **DX**/**DZ** for turning tool or additive compensation
  - 2: Recessing tool **DX**/**DS**
  - 3: Milling tool **DX**/**DD**
  - 4: Milling tool **DD**
- **K**: **Incr. meas path w/ Ri.** (the algebraic sign determines the probing direction) – maximum measuring path for probing
- **RB**: **Circumnav. dir. offset Xi** – distance
- **RC**: **Offset in Yi** – distance for pre-positioning before the second measurement
- **YE**: **Target pos. noml value Y** – absolute coordinate of touch point
- **BD**: **Tolerance position +/-** – measurement result range in which no compensation will be applied
- **Y**: **Nominal width in Y** – coordinate of the second probing position
- **BE**: **Tolerance width +/-** – range for the second measurement result in which no compensation will be applied
- **WT**: **Compens. no.1, measd. edge**
  - **T**: Tool at turret position **T** to compensate the difference to the nominal value
  - **G149**: Additive compensation **D9xx** to correct the difference to the nominal value (only possible with compensation type **R** = 1)

- **AT**: **Compens. no.2, measd. edge**
  - **T**: Tool at turret position **T** to compensate the difference to the nominal value
  - **G149**: Additive compensation **D9xx** to correct the difference to the nominal value (only possible with compensation type **R** = 1)
- **FP**: **Maximum perm. correction**
- **WE**: **Max. deviation** – probe twice and monitor the dispersion of the measured values
- **F**: **Measuring feed** – feed rate for probing (if nothing is entered, the measuring feed rate from the touch probe table will be used)
  If you enter a measuring feed rate **F** that is higher than the one in the touch probe table, the feed rate will be reduced to the value from the touch probe table.
- **Q**: **Tool orientation** (machine-dependent)
  Orient the touch probe in the programmed probing direction before each probing operation.
- **P**: **PRINT outputs**
  - **0: OFF** – do not display measuring results
  - **1: ON** – display measuring results on the screen
- **H**: **INPUT instead of measure**
  - **0: Standard** – obtain measured values by probing
  - **1: PC test** – simulate the touch probe cycle at the programming station
- **AN**: **Log no.** – save measurement results in the **TNC:\table\messpro.mep** table (line numbers: 0 to 99)
  The table can be extended if necessary.

> The cycle computes the **Compens. no.1, measd. edge WT** from the result of the first measurement and the **Compens. no.2, measd. edge AT** from the result of the second measurement.

Example: G778 Two-point measurement for tool compensation

. . .	
MACHINING	
N3 G778 R1 K20 YE30 BD0.2 Y40 BE0.3 WT5Q0 P0 H0	
. . .	

7.4 Calibrating touch probes

Calibrate touch probe standard G747

Cycle **G747** measures in the programmed axis and, depending on the selected calibration method, calculates the touch probe adjustment dimension or the ball diameter. If the tolerance values defined in the cycle are exceeded, the cycle corrects the touch probe data. In addition, the measurement result is saved additionally in the variable **#i99**.

Further information: "Touch probe cycles for automatic mode", Page 579

Cycle run: From the current position the touch probe moves along the defined measuring axis toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point.

The control displays an error message if the touch probe does not reach any touch point within the defined measuring path. If a **Max. deviation WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the **Max. deviation WE**, the program run is interrupted and an error message is displayed.

Parameters:

- **R**: **Calibration method**
  - 0: change CAx
  - 1: change ball diameter
  - 2: change setup dimension
- **D**: **Measuring axis** – axis in which the measurement is to be made
- **K**: **Incr. meas path w/ Ri.** (the algebraic sign determines the probing direction) – maximum measuring path for probing
- **AC**: **Target pos. nominal value** – touch point coordinate
- **BD**: **Tolerance position +/-** – measurement result range in which no compensation will be applied
- **WE**: **Max. deviation** – probe twice and monitor the dispersion of the measured values
- **F**: **Measuring feed** – feed rate for probing (if nothing is entered, the measuring feed rate from the touch probe table will be used)
  If you enter a measuring feed rate **F** that is higher than the one in the touch probe table, the feed rate will be reduced to the value from the touch probe table.
- **Q**: **Tool orientation** (machine-dependent)
  Orient the touch probe in the programmed probing direction before each probing operation.
- **P**: **PRINT outputs**
  - **0: OFF** – do not display measuring results
  - **1: ON** – display measuring results on the screen

- **H**: **INPUT instead of measure**
  - **0: Standard** – obtain measured values by probing
  - **1: PC test** – simulate the touch probe cycle at the programming station
- **AN**: **Log no.** – save measurement results in the **TNC:\table\messpro.mep** table (line numbers: 0 to 99) The table can be extended if necessary.

Example: G747 Calibrate touch probe

. . .	
MACHINING	
N3 G747 R1 K20 AC10 BD0.2 Q0 P0 H0	
. . .	

Calibrate touch probe via two points G748

Cycle **G748** measures two opposite points and computes the touch probe adjustment dimension and the ball diameter. If the tolerance values defined in the cycle are exceeded, the cycle corrects the touch probe data. The result of the measurement is saved additionally in the variable **#i99**.

Further information: "Touch probe cycles for automatic mode", Page 579

Cycle run: From the current position the touch probe moves along the defined measuring axis toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point.

The control displays an error message if the touch probe does not reach any touch point within the defined measuring path. If a **Max. deviation WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the **Max. deviation WE**, the program run is interrupted and an error message is displayed.

Parameters:

- **K**: **Incr. meas path w/ Ri.** (the algebraic sign determines the probing direction) – maximum measuring path for probing
- **RB**: **Circumnav. dir. offset** – distance
- **RC**: **Measuring direction offset** – distance for pre-positioning before the second measurement
- **AC**: **Target pos. nominal value** – touch point coordinate
- **EC**: **Nominal width** – coordinate of the second probing position
- **BE**: **Tolerance width +/-** – range for the second measurement result in which no compensation will be applied
- **WE**: **Max. deviation** – probe twice and monitor the dispersion of the measured values
- **F**: **Measuring feed** – feed rate for probing (if nothing is entered, the measuring feed rate from the touch probe table will be used)
  If you enter a measuring feed rate **F** that is higher than the one in the touch probe table, the feed rate will be reduced to the value from the touch probe table.
- **Q**: **Tool orientation** (machine-dependent)
  Orient the touch probe in the programmed probing direction before each probing operation.
- **P**: **PRINT outputs**
  - **0: OFF** – do not display measuring results
  - **1: ON** – display measuring results on the screen

- **H**: **INPUT instead of measure**
  - **0: Standard** – obtain measured values by probing
  - **1: PC test** – simulate the touch probe cycle at the programming station
- **AN**: **Log no.** – save measurement results in the **TNC:\table\messpro.mep** table (line numbers: 0 to 99) The table can be extended if necessary.

Example: G748 Calibrate touch probe via two points

. . .	
MACHINING	
N3 G748 K20 AC10 EC33 Q0 P0 H0	
. . .	

7.5 Measuring with touch probe cycles

Paraxial probing G764

Cycle **G764** measures with the programmed axis and displays the measured values on the control screen. The result of the measurement is saved additionally in the variable **#i99**.

Further information: "Touch probe cycles for automatic mode", Page 579

Cycle run: From the current position the touch probe moves along the defined measuring axis toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path.

Parameters:

- **D**: **Measuring axis** – axis in which the measurement is to be made
- **K**: **Incr. meas path w/ Ri.** (the algebraic sign determines the probing direction) – maximum measuring path for probing
- **V**: **Retraction type**
  - 0: Without—only position touch probe back to the starting point if the touch probe was deflected
  - 1: Automatic—always position touch probe back to the starting point
- **O**: **Err. evaluation**
  - 0: Program—do not interrupt program run, no error message
  - 1: Automatic—interrupt program run and output error message if touch probe is not deflected within measuring path
- **F**: **Measuring feed** – feed rate for probing (if nothing is entered, the measuring feed rate from the touch probe table will be used) If you enter a measuring feed rate **F** that is higher than the one in the touch probe table, the feed rate will be reduced to the value from the touch probe table.
- **Q**: **Tool orientation** (machine-dependent) Orient the touch probe in the programmed probing direction before each probing operation.
- **P**: **PRINT outputs**
  - **0: OFF** – do not display measuring results
  - **1: ON** – display measuring results on the screen

- **H**: **INPUT instead of measure**
  - **0: Standard** – obtain measured values by probing
  - **1: PC test** – simulate the touch probe cycle at the programming station

Example: G764 Paraxial probing

. . .	
MACHINING	
N3 G764 D0 K20 V1 O1 Q0 P0 H0	
. . .	

Probing in C axis G765

Cycle **G765** measures with the C axis and displays the measured values on the control screen. The result of the measurement is saved additionally in the variable **#i99**.

Further information: "Touch probe cycles for automatic mode", Page 579

Cycle run: From the current position, the element to be probed is moved toward the touch probe by a rotation of the C axis. When the workpiece touches the stylus, the measured value is saved and the workpiece is returned.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path.

Parameters:

- **C**: **Incr. meas path w/ Ri.** (the algebraic sign determines the probing direction) – C-axis measuring path (in degrees) starting from the current position
- **V**: **Retraction type**
  - 0: Without—only position touch probe back to the starting point if the touch probe was deflected
  - 1: Automatic—always position touch probe back to the starting point
- **O**: **Err. evaluation**
  - 0: Program—do not interrupt program run, no error message
  - 1: Automatic—interrupt program run and output error message if touch probe is not deflected within measuring path
- **F**: **Measuring feed** – feed rate for probing (if nothing is entered, the measuring feed rate from the touch probe table will be used)
  If you enter a measuring feed rate **F** that is higher than the one in the touch probe table, the feed rate will be reduced to the value from the touch probe table.
- **Q**: **Tool orientation** (machine-dependent)
  Orient the touch probe in the programmed probing direction before each probing operation.
- **P**: **PRINT outputs**
  - **0: OFF** – do not display measuring results
  - **1: ON** – display measuring results on the screen
- **H**: **INPUT instead of measure**
  - **0: Standard** – obtain measured values by probing
  - **1: PC test** – simulate the touch probe cycle at the programming station

Example: G765 Probing in C axis

. . .	
MACHINING	
N3 G765 C20 V1 O1 AC0 BD0.2 Q0 P0 H0	
. . .	

Probing w/ 2 axes in ZX plane G766

Cycle **G766** measures the position programmed in the cycle in the X/Z plane and displays the measured values on the control screen. In parameter **NF** you can additionally define the variables in which the measurement results should be saved.

Cycle run: The touch probe moves from the current position toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path.

Parameters:

- **Z**: **Target pt Z** – Z coordinate of the measuring point
- **X**: **Target pt X** – X coordinate of the measuring point
- **V**: **Retraction type**
  - 0: Without—only position touch probe back to the starting point if the touch probe was deflected
  - 1: Automatic—always position touch probe back to the starting point
- **O**: **Err. evaluation**
  - 0: Program—do not interrupt program run, no error message
  - 1: Automatic—interrupt program run and output error message if touch probe is not deflected within measuring path
- **F**: **Measuring feed** – feed rate for probing (if nothing is entered, the measuring feed rate from the touch probe table will be used)
  If you enter a measuring feed rate **F** that is higher than the one in the touch probe table, the feed rate will be reduced to the value from the touch probe table.
- **Q**: **Tool orientation** (machine-dependent)
  Orient the touch probe in the programmed probing direction before each probing operation.
- **P**: **PRINT outputs**
  - **0: OFF** – do not display measuring results
  - **1: ON** – display measuring results on the screen
- **H**: **INPUT instead of measure**
  - **0: Standard** – obtain measured values by probing
  - **1: PC test** – simulate the touch probe cycle at the programming station

Example: G766 Probing w/ 2 axes in ZX plane

...	
MACHINING	
N3 G766 Z-5 X30 V1 O1 AC0 BD0.2 Q0 P0 H0	
...	

Probing w/ 2 axes in ZX plane G767

Cycle **G767** measures the position programmed in the cycle in the X/C plane and displays the measured values on the control screen. In parameter **NF** you can additionally define the variables in which the measurement results should be saved.

Cycle run: The touch probe moves from the current position toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path.

Parameters:

- **XK**: **Target point** (in Cartesian coordinates)
- **YK**: **Target point** (in Cartesian coordinates)
- **V**: **Retraction type**
  - 0: Without—only position touch probe back to the starting point if the touch probe was deflected
  - 1: Automatic—always position touch probe back to the starting point
- **O**: **Err. evaluation**
  - 0: Program—do not interrupt program run, no error message
  - 1: Automatic—interrupt program run and output error message if touch probe is not deflected within measuring path
- **F**: **Measuring feed** – feed rate for probing (if nothing is entered, the measuring feed rate from the touch probe table will be used)
  If you enter a measuring feed rate **F** that is higher than the one in the touch probe table, the feed rate will be reduced to the value from the touch probe table.
- **Q**: **Tool orientation** (machine-dependent)
  Orient the touch probe in the programmed probing direction before each probing operation.
- **P**: **PRINT outputs**
  - **0: OFF** – do not display measuring results
  - **1: ON** – display measuring results on the screen
- **H**: **INPUT instead of measure**
  - **0: Standard** – obtain measured values by probing
  - **1: PC test** – simulate the touch probe cycle at the programming station

Example: G767 Probing w/ 2 axes in XC plane

. . .	
MACHINING	
N3 G767 XK30 YK5 V1 O1 Q0 P0 H0	
. . .	

Probing w/ 2 axes in ZY plane G768

Cycle **G768** measures the position programmed in the cycle in the Z/Y plane and displays the measured values on the control screen. In parameter **NF** you can additionally define the variables in which the measurement results should be saved.

Cycle run: The touch probe moves from the current position toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path.

Parameters:

- **Z**: **Target pt Z** – Z coordinate of the measuring point
- **Y**: **Target point Y** – Y coordinate of the measuring point
- **V**: **Retraction type**
  - 0: Without—only position touch probe back to the starting point if the touch probe was deflected
  - 1: Automatic—always position touch probe back to the starting point
- **O**: **Err. evaluation**
  - 0: Program—do not interrupt program run, no error message
  - 1: Automatic—interrupt program run and output error message if touch probe is not deflected within measuring path
- **F**: **Measuring feed** – feed rate for probing (if nothing is entered, the measuring feed rate from the touch probe table will be used) If you enter a measuring feed rate **F** that is higher than the one in the touch probe table, the feed rate will be reduced to the value from the touch probe table.
- **Q**: **Tool orientation** (machine-dependent) Orient the touch probe in the programmed probing direction before each probing operation.
- **P**: **PRINT outputs**
  - **0: OFF** – do not display measuring results
  - **1: ON** – display measuring results on the screen
- **H**: **INPUT instead of measure**
  - **0: Standard** – obtain measured values by probing
  - **1: PC test** – simulate the touch probe cycle at the programming station

Example: G768 Probing w/ 2 axes in ZY plane

...	
MACHINING	
N3 G768 Z-5 Y10 V1 O1 AC0 BD0.2 Q0 P0 H0	
...	

Probing w/ 2 axes in XY plane G769

Cycle **G769** measures the position programmed in the cycle in the X/Y plane and displays the measured values on the control screen. In parameter **NF** you can additionally define the variables in which the measurement results should be saved.

Cycle run: The touch probe moves from the current position toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is positioned back to the starting point.

The control outputs an error message if the touch probe does not reach any touch point within the defined measuring path.

Parameters:

- **X**: **Target pt X** – X coordinate of the measuring point
- **Y**: **Target point Y** – Y coordinate of the measuring point
- **V**: **Retraction type**
  - 0: Without—only position touch probe back to the starting point if the touch probe was deflected
  - 1: Automatic—always position touch probe back to the starting point
- **O**: **Err. evaluation**
  - 0: Program—do not interrupt program run, no error message
  - 1: Automatic—interrupt program run and output error message if touch probe is not deflected within measuring path
- **F**: **Measuring feed** – feed rate for probing (if nothing is entered, the measuring feed rate from the touch probe table will be used)
  If you enter a measuring feed rate **F** that is higher than the one in the touch probe table, the feed rate will be reduced to the value from the touch probe table.
- **Q**: **Tool orientation** (machine-dependent)
  Orient the touch probe in the programmed probing direction before each probing operation.
- **P**: **PRINT outputs**
  - **0: OFF** – do not display measuring results
  - **1: ON** – display measuring results on the screen
- **H**: **INPUT instead of measure**
  - **0: Standard** – obtain measured values by probing
  - **1: PC test** – simulate the touch probe cycle at the programming station

Example: G769 Probing w/ 2 axes in XY plane

. . .	
MACHINING	
N3 G769 X25 Y10 V1 O1 AC0 BD0.2 Q0 P0 H0	
. . .	

7.6 Search cycles

Find hole in C face G780

Cycle **G780** probes the workpiece face several times with the Z axis. Prior to each probing, the touch probe is shifted by a distance defined in the cycle until a hole is found. Optionally, the cycle determines the mean value by two probing operations in the hole.

If the tolerance value defined in the cycle is exceeded, the cycle saves the measured deviation as datum shift. The result of the measurement is saved additionally in the variable **#i99**.

Result #i99	Meaning
< 999997	Result of first measurement
999999	Deviation of probing operations was higher than programmed in parameter **Max. deviation WE**
–999999	Hole was not found

Cycle run: From the current position, the touch probe moves along the measuring axis Z toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe returns to the starting point. Then, the cycle rotates the C axis by the angle defined in parameter **Search grid Ci RC** and probes again in the Z axis. This process is repeated until a hole is found. Inside the hole, the cycle performs two probing operations in the C axis, calculates the center of the hole, and places the datum in the C axis.

The control displays an error message if the touch probe does not reach any touch point within the defined measuring path. If a **Max. deviation WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the **Max. deviation WE**, the program run is interrupted and an error message is displayed.

Parameters:

- **R**: **Type of datum shift**
  - 1: Table and **G152** – activate datum shift and additionally save in datum table (the datum shift also remains active after the program run)
  - 2: Activate datum shift with **G152** for the further program run (datum shift no longer active after program run)
- **D**: **Result:**
  - 1: Position—set datum without determining the hole center. No probing operation in the hole.
  - 2: Object center—before the datum is set, determine hole center in two probing operations with the C axis.
- **K**: **Incr. meas path Z with Ri.** (the algebraic sign determines the probing direction) – maximum measuring path for probing
- **C**: **Starting position C** – position of the C axis for the first probing operation
- **RC**: **Search grid Ci** – stepping angle of the C axis for the subsequent probing operations
- **A**: **Number of points** – maximum number of probing operations

- **IC**: **Measuring path in C** – measuring path of the C axis (in degrees), starting from the current position (the algebraic sign determines the probing direction)
- **AC**: **Target pos. nominal value** – absolute coordinate of the touch point in degrees
- **BD**: **Tolerance position +/-** – measurement result range in which no compensation will be applied
- **KC**: **Compensation offset** – additional compensation value that is applied to the resulting datum
- **WE**: **Max. deviation** – probe twice and monitor the dispersion of the measured values
- **F**: **Measuring feed** – feed rate for probing (if nothing is entered, the measuring feed rate from the touch probe table will be used)
  If you enter a measuring feed rate **F** that is higher than the one in the touch probe table, the feed rate will be reduced to the value from the touch probe table.
- **Q**: **Tool orientation** (machine-dependent)
  Orient the touch probe in the programmed probing direction before each probing operation.
- **P**: **PRINT outputs**
  - **0: OFF** – do not display measuring results
  - **1: ON** – display measuring results on the screen
- **H**: **INPUT instead of measure**
  - **0: Standard** – obtain measured values by probing
  - **1: PC test** – simulate the touch probe cycle at the programming station
- **AN**: **Log no.** – save measurement results in the **TNC:\table\messpro.mep** table (line numbers: 0 to 99)
  The table can be extended if necessary.

Example: G780 Find hole in C face G780

...	
MACHINING	
N3 G780 R1 D1 K2 C0 RC10 IC20 AC0 BD0.2 Q0P0 H0	
...	

Find hole C lateral surface G781

Cycle **G780** probes the lateral surface of a workpiece several times with the X axis. Prior to each probing, the C axis is rotated by a distance defined in the cycle until a hole is found. Optionally, the cycle determines the mean value by two probing operations in the hole.

If the tolerance value defined in the cycle is exceeded, the cycle saves the measured deviation as datum shift. The result of the measurement is saved additionally in the variable **#i99**.

Result #i99	Meaning
< 999997	Result of first measurement
999999	Deviation of probing operations was higher than programmed in parameter **Max. deviation WE**
–999999	Hole was not found

Cycle run: From the current position, the touch probe moves along the measuring axis X toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe returns to the starting point. Then, the cycle rotates the C axis by the angle defined in the parameter **Search grid Ci RC** and probes again in the X axis. This process is repeated until a hole is found. Inside the hole, the cycle performs two probing operations in the C axis, calculates the center of the hole, and places the datum in the C axis.

The control displays an error message if the touch probe does not reach any touch point within the defined measuring path. If a **Max. deviation WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the **Max. deviation WE**, the program run is interrupted and an error message is displayed.

Parameters:

- **R**: **Type of datum shift**
  - 1: Table and **G152** – activate datum shift and additionally save in datum table (the datum shift also remains active after the program run)
  - 2: Activate datum shift with **G152** for the further program run (datum shift no longer active after program run)
- **D**: **Result:**
  - 1: Position—set datum without determining the hole center. No probing operation in the hole.
  - 2: Object center—before the datum is set, determine hole center in two probing operations with the C axis.
- **K**: **Incr. meas path X with Ri.** (the algebraic sign determines the probing direction) – maximum measuring path for probing
- **C**: **Starting position C** – position of the C axis for the first probing operation
- **RC**: **Search grid Ci** – stepping angle of the C axis for the subsequent probing operations
- **A**: **Number of points** – maximum number of probing operations
- **IC**: **Measuring path in C** – measuring path of the C axis (in degrees), starting from the current position (the algebraic sign determines the probing direction)

- **AC**: **Target pos. nominal value** – absolute coordinate of the touch point in degrees
- **BD**: **Tolerance position +/-** – measurement result range in which no compensation will be applied
- **KC**: **Compensation offset** – additional compensation value that is applied to the resulting datum
- **WE**: **Max. deviation** – probe twice and monitor the dispersion of the measured values
- **F**: **Measuring feed** – feed rate for probing (if nothing is entered, the measuring feed rate from the touch probe table will be used)
  If you enter a measuring feed rate **F** that is higher than the one in the touch probe table, the feed rate will be reduced to the value from the touch probe table.
- **Q**: **Tool orientation** (machine-dependent)
  Orient the touch probe in the programmed probing direction before each probing operation.
- **P**: **PRINT outputs**
  - **0: OFF** – do not display measuring results
  - **1: ON** – display measuring results on the screen
- **H**: **INPUT instead of measure**
  - **0: Standard** – obtain measured values by probing
  - **1: PC test** – simulate the touch probe cycle at the programming station
- **AN**: **Log no.** – save measurement results in the **TNC:\table\messpro.mep** table (line numbers: 0 to 99)
  The table can be extended if necessary.

Example: G781 Find hole in C lateral surface

. . .	
MACHINING	
N3 G781 R1 D1 K2 C0 RC10 IC20 AC0 BD0.2 Q0P0 H0	
. . .	

Find stud in C face G782

Cycle **G782** probes the workpiece face several times with the Z axis. Prior to each probing, the C axis is rotated by a distance defined in the cycle until a stud is found. Optionally, the cycle determines the mean value by two probing operations on the stud diameter.

If the tolerance value defined in the cycle is exceeded, the cycle saves the measured deviation as datum shift. The result of the measurement is saved additionally in the variable **#i99**.

Result #i99	Meaning
< 999997	Result of first measurement
999999	Deviation of probing operations was higher than programmed in parameter **Max. deviation WE**
–999999	Stud was not found.

Cycle run: From the current position, the touch probe moves along the measuring axis X toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe returns to the starting point. Then, the cycle rotates the C axis by the angle defined in the parameter **Search grid Ci RC** and probes again in the X axis. This process is repeated until a stud is found. The cycle performs two probing operations on the stud diameter with the C axis, calculates the center of the stud, and places the datum in the C axis.

The control displays an error message if the touch probe does not reach any touch point within the defined measuring path. If a **Max. deviation WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the **Max. deviation WE**, the program run is interrupted and an error message is displayed.

Parameters:

- **R**: **Type of datum shift**
  - 1: Table and **G152** – activate datum shift and additionally save in datum table (the datum shift also remains active after the program run)
  - 2: Activate datum shift with **G152** for the further program run (datum shift no longer active after program run)
- **D**: **Result:**
  - 1: Position—set datum without determining the stud center. The stud diameter is not probed.
  - 2: Object center—before the datum is set, determine stud center in two probing operations with the C axis.
- **K**: **Incr. meas path Z with Ri.** (the algebraic sign determines the probing direction) – maximum measuring path for probing
- **C**: **Starting position C** – position of the C axis for the first probing operation
- **RC**: **Search grid Ci** – stepping angle of the C axis for the subsequent probing operations
- **A**: **Number of points** – maximum number of probing operations
- **IC**: **Measuring path in C** – measuring path of the C axis (in degrees), starting from the current position (the algebraic sign determines the probing direction)

- **AC**: **Target pos. nominal value** – absolute coordinate of the touch point in degrees
- **BD**: **Tolerance position +/-** – measurement result range in which no compensation will be applied
- **KC**: **Compensation offset** – additional compensation value that is applied to the resulting datum
- **WE**: **Max. deviation** – probe twice and monitor the dispersion of the measured values
- **F**: **Measuring feed** – feed rate for probing (if nothing is entered, the measuring feed rate from the touch probe table will be used)
  If you enter a measuring feed rate **F** that is higher than the one in the touch probe table, the feed rate will be reduced to the value from the touch probe table.
- **Q**: **Tool orientation** (machine-dependent)
  Orient the touch probe in the programmed probing direction before each probing operation.
- **P**: **PRINT outputs**
  - **0: OFF** – do not display measuring results
  - **1: ON** – display measuring results on the screen
- **H**: **INPUT instead of measure**
  - **0: Standard** – obtain measured values by probing
  - **1: PC test** – simulate the touch probe cycle at the programming station
- **AN**: **Log no.** – save measurement results in the **TNC:\table\messpro.mep** table (line numbers: 0 to 99)
  The table can be extended if necessary.

Example: G782 Find stud in C face

. . .	
MACHINING	
N3 G782 R1 D1 K2 C0 RC10 IC20 AC0 BD0.2 Q0P0 H0	
. . .	

Find stud C lateral surface G783

Cycle **G783** probes the workpiece face several times with the X axis. Prior to each probing, the touch probe is shifted by a distance defined in the cycle until a stud is found. Optionally, the cycle determines the mean value by two probing operations on the stud diameter.

If the tolerance value defined in the cycle is exceeded, the cycle saves the measured deviation as datum shift. The result of the measurement is saved additionally in the variable **#i99**.

Result #i99	Meaning
< 999997	Result of first measurement
999999	Deviation of probing operations was higher than programmed in parameter **Max. deviation WE**
–999999	Stud was not found.

Cycle run: From the current position, the touch probe moves along the measuring axis Z toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe returns to the starting point. Then, the cycle rotates the C axis by the angle defined in parameter **Search grid Ci RC** and probes again in the Z axis. This process is repeated until a stud is found. The cycle performs two probing operations on the stud diameter with the C axis, calculates the center of the stud, and places the datum in the C axis.

The control displays an error message if the touch probe does not reach any touch point within the defined measuring path. If a **Max. deviation WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the **Max. deviation WE**, the program run is interrupted and an error message is displayed.

Parameters:

- **R**: **Type of datum shift**
  - 1: Table and **G152** – activate datum shift and additionally save in datum table (the datum shift also remains active after the program run)
  - 2: Activate datum shift with **G152** for the further program run (datum shift no longer active after program run)
- **D**: **Result:**
  - 1: Position—set datum without determining the stud center. The stud diameter is not probed.
  - 2: Object center—before the datum is set, determine stud center in two probing operations with the C axis.
- **K**: **Incr. meas path X with Ri.** (the algebraic sign determines the probing direction) – maximum measuring path for probing
- **C**: **Starting position C** – position of the C axis for the first probing operation
- **RC**: **Search grid Ci** – stepping angle of the C axis for the subsequent probing operations
- **A**: **Number of points** – maximum number of probing operations
- **IC**: **Measuring path in C** – measuring path of the C axis (in degrees), starting from the current position (the algebraic sign determines the probing direction)

- **AC**: **Target pos. nominal value** – absolute coordinate of the touch point in degrees
- **BD**: **Tolerance position +/-** – measurement result range in which no compensation will be applied
- **KC**: **Compensation offset** – additional compensation value that is applied to the resulting datum
- **WE**: **Max. deviation** – probe twice and monitor the dispersion of the measured values
- **F**: **Measuring feed** – feed rate for probing (if nothing is entered, the measuring feed rate from the touch probe table will be used)
  If you enter a measuring feed rate **F** that is higher than the one in the touch probe table, the feed rate will be reduced to the value from the touch probe table.
- **Q**: **Tool orientation** (machine-dependent)
  Orient the touch probe in the programmed probing direction before each probing operation.
- **P**: **PRINT outputs**
  - **0: OFF** – do not display measuring results
  - **1: ON** – display measuring results on the screen
- **H**: **INPUT instead of measure**
  - **0: Standard** – obtain measured values by probing
  - **1: PC test** – simulate the touch probe cycle at the programming station
- **AN**: **Log no.** – save measurement results in the **TNC:\table\messpro.mep** table (line numbers: 0 to 99)
  The table can be extended if necessary.

Example: G783 Find stud in C lateral surface

. . .	
MACHINING	
N3 G783 R1 D1 K2 C0 RC10 IC20 AC0 BD0.2 Q0P0 H0	
. . .	

7.7 Circular measurement

Circular measurmnt G785

Cycle **G785** determines the circle center and diameter by probing three times in the programmed plane and shows the measured values on the control screen. The result of the measurement is saved additionally in the variable **#i99**.

Further information: "Touch probe cycles for automatic mode", Page 579

Cycle run: From the current position, the touch probe moves in the defined measuring plane toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe returns to the starting point. Another two probing operations are carried out with the defined stepping angle. If a **Starting diameter D** was programmed, the cycle positions the touch probe along a circular path before the respective measuring process.

The control displays an error message if the touch probe does not reach any touch point within the defined measuring path. If a **Max. deviation WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the **Max. deviation WE**, the program run is interrupted and an error message is displayed.

Parameters:

- **R**: **Measuring plane**
  - 0: X/Y plane **G17** – probe circle in X/Y plane
  - 1: Z/X plane **G18** – probe circle in Z/X plane
  - 2: Y/Z plane **G19** – probe circle in Y/Z plane
  - 3: X/C plane on face C
- **BR**: **Inside/outside**
  - 0: Inside: Probe inside diameter
  - 1: Outside: Probe outside diameter
- **K**: **Measuring path** (the algebraic sign determines the probing direction) – maximum measuring path for probing
- **C**: **Angle of 1st measurement** – angle for the first probing operation
- **RC**: **Incremental angle** – stepping angle for the subsequent probing operations
- **D**: **Starting diameter** – diameter on which the touch probe is pre-positioned before the measurements
- **WB**: **Position in infeed direct.** – measuring height to which the touch probe is positioned before the measuring process (no input: the circle is probed from the current position)
- **I**: **Circle center in axis 1** – nominal position of the circle center in first axis
- **J**: **Circle center in axis 2** – nominal position of the circle center in second axis
- **WE**: **Max. deviation** – probe twice and monitor the dispersion of the measured values

- **F**: **Measuring feed** – feed rate for probing (if nothing is entered, the measuring feed rate from the touch probe table will be used)
  If you enter a measuring feed rate **F** that is higher than the one in the touch probe table, the feed rate will be reduced to the value from the touch probe table.
- **Q**: **Tool orientation** (machine-dependent)
  Orient the touch probe in the programmed probing direction before each probing operation.
- **NF**: **Variable no. result** – number of the first global variable in which the result is saved (no input: variable **810**)
  The second measurement result is saved automatically to the next consecutive number.
- **P**: **PRINT outputs**
  - **0: OFF** – do not display measuring results
  - **1: ON** – display measuring results on the screen
- **H**: **INPUT instead of measure**
  - **0: Standard** – obtain measured values by probing
  - **1: PC test** – simulate the touch probe cycle at the programming station
- **AN**: **Log no.** – save measurement results in the **TNC:\table\messpro.mep** table (line numbers: 0 to 99)
  The table can be extended if necessary.

Example: G785 Circular measurement

. . .	
MACHINING	
N3 G785 R0 BR0 K2 C0 RC60 I0 J0 Q0 P0 H0	
. . .	

Calc. of pitch circle G786

Cycle **G786** determines the center and diameter of a pitch circle by measuring three holes and shows the measured values on the control screen. The result of the measurement is saved additionally in the variable **#i99**.

Further information: "Touch probe cycles for automatic mode", Page 579

Cycle run: From the current position, the touch probe moves in the defined measuring plane toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe returns to the starting point. Another two probing operations are carried out with the defined stepping angle. If a **Starting diameter D** was programmed, the cycle positions the touch probe along a circular path before the respective measuring process.

The control displays an error message if the touch probe does not reach any touch point within the defined measuring path. If a **Max. deviation WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the **Max. deviation WE**, the program run is interrupted and an error message is displayed.

Parameters:

- **R**: **Measuring plane**
  - 0: X/Y plane **G17** – probe circle in X/Y plane
  - 1: Z/X plane **G18** – probe circle in Z/X plane
  - 2: Y/Z plane **G19** – probe circle in Y/Z plane
  - 3: X/C plane on face C
- **K**: **Measuring path** (the algebraic sign determines the probing direction) – maximum measuring path for probing
- **C**: **Angle of 1st hole** – angle for the first probing operation
- **AC**: **Angle of 2nd hole** – angle for the second probing operation
- **RC**: **Angle of 3rd hole** – angle for the third probing operation
- **WB**: **Position in infeed direct.** – measuring height to which the touch probe is positioned before the measuring process (no input: the circle is probed from the current position)
- **I**: **Circ. scale center in axis 1** – nominal position of the circle center in first axis
- **J**: **Circ. scale center in axis 2** – nominal position of the circle center in second axis
- **D**: **Nominal diameter** – diameter on which the touch probe is pre-positioned before the measurements
- **WS**: **Upper tol., calcultd. dia.** of pitch circle
- **WC**: **Lower tol., calcultd. dia.** of pitch circle
- **BD**: **Tol. of center in axis 1**
- **BE**: **Tol. of center in axis 2**
- **WE**: **Max. deviation** – probe twice and monitor the dispersion of the measured values

- **F**: **Measuring feed** – feed rate for probing (if nothing is entered, the measuring feed rate from the touch probe table will be used)
  If you enter a measuring feed rate **F** that is higher than the one in the touch probe table, the feed rate will be reduced to the value from the touch probe table.
- **Q**: **Tool orientation** (machine-dependent)
  Orient the touch probe in the programmed probing direction before each probing operation.
- **NF**: **Variable no. result** – number of the first global variable in which the result is saved (no input: variable **810**)
  The second measurement result is saved automatically to the next consecutive number.
- **P**: **PRINT outputs**
  - **0: OFF** – do not display measuring results
  - **1: ON** – display measuring results on the screen
- **H**: **INPUT instead of measure**
  - **0: Standard** – obtain measured values by probing
  - **1: PC test** – simulate the touch probe cycle at the programming station
- **AN**: **Log no.** – save measurement results in the **TNC:\table\messpro.mep** table (line numbers: 0 to 99)
  The table can be extended if necessary.

Example: G786 Calculation of pitch circle

...	
MACHINING	
N3 G786 R0 K8 I0 J0 D50 WS50.1 WC49.9BD0.1 BE0.1 P0 H0	
...	

7.8 Angular measurement

Angular measurmnt G787

Cycle **G787** probes twice in the programmed direction and computes the angle. If the tolerance value defined in the cycle is exceeded, the cycle saves the measured deviation for a subsequent misalignment compensation. Program Cycle **G788** next in order to activate the misalignment compensation. The result of the measurement is saved additionally in the variable **#i99**.

Further information: "Touch probe cycles for automatic mode", Page 579

Cycle run: From the current position the touch probe moves along the defined measuring axis toward the measuring point. When the stylus touches the workpiece, the measured value is saved and the touch probe is retracted. Then the touch probe is pre-positioned for the second measurement and the workpiece is probed.

The control displays an error message if the touch probe does not reach any touch point within the defined measuring path. If a **Max. deviation WE** was programmed, the measuring point is approached twice and the mean value is saved as result. If the difference of the measurements is greater than the **Max. deviation WE**, the program run is interrupted and an error message is displayed.

Parameters:

- **R**: **Evaluation**
  - 1: Prepare tool compensation and misalignment compensation
  - 2: Prepare misalignment compensation
  - 3: Angle output
- **D**: **Directions**
  - 0: X measurement, Z offset
  - 1: Y measurement Z offset
  - 2: Z measurement, X offset
  - 3: Y measurement, X offset
  - 4: Z measurement, Y offset
  - 5: X measurement, Y offset
- **K**: **Measuring path** (the algebraic sign determines the probing direction) – maximum measuring path for probing
- **WS**: **Pos. of 1st measurement**
- **WC**: **Pos. of 2nd measurement**
- **AC**: **Nominal angle** of measured surface
- **BE**: **Tolerance of angle +/-** – measurement result range (in degrees) in which no compensation will be applied
- **RC**: **Target pos., 1st measmnt.** – nominal value of first measuring point
- **BD**: **Tol., 1st measurement +/-** – measurement result range in which no compensation will be applied

- **WT**: **Correction no. T or G149**
  - **T**: Tool at turret position **T** to compensate the difference to the nominal value
  - **G149**: Additive compensation **D9xx** to correct the difference to the nominal value (only possible with compensation type **R** = 1)
- **FP**: **Maximum perm. correction**
- **WE**: **Max. deviation** – probe twice and monitor the dispersion of the measured values
- **F**: **Measuring feed** – feed rate for probing (if nothing is entered, the measuring feed rate from the touch probe table will be used)
  If you enter a measuring feed rate **F** that is higher than the one in the touch probe table, the feed rate will be reduced to the value from the touch probe table.
- **Q**: **Tool orientation** (machine-dependent)
  Orient the touch probe in the programmed probing direction before each probing operation.
- **NF**: **Variable no. result** – number of the first global variable in which the result is saved (no input: variable **810**)
  The second measurement result is saved automatically to the next consecutive number.
- **P**: **PRINT outputs**
  - **0: OFF** – do not display measuring results
  - **1: ON** – display measuring results on the screen
- **H**: **INPUT instead of measure**
  - **0: Standard** – obtain measured values by probing
  - **1: PC test** – simulate the touch probe cycle at the programming station
- **AN**: **Log no.** – save measurement results in the **TNC:\table\messpro.mep** table (line numbers: 0 to 99)
  The table can be extended if necessary.

Example: G787 Angular measurement

...	
MACHINING	
N3 G787 R1 D0 BR0 K2 WS-2 WC15 AC170 BE1RC0 BD0.2 WT3 Q0 P0 H0	
...	

Misalignment compensation after angle measurement G788

Cycle **G788** activates a misalignment compensation determined with Cycle **G787** Angle Measurement.

Parameters:

- **NF**: **Variable no. result** – number of the first global variable in which the result is saved (no input: variable **810**)
  The second measurement result is saved automatically to the next consecutive number.
- **P**: **Compensation:**
  - **0: OFF** – do not perform misalignment compensation
  - **1: ON** – perform misalignment compensation

G788 Misalignment compensation after angle measurement

. . .	
MACHINING	
N3 G788 NF1 P0	
. . .	

7.9 In-process measrmnt.

Workpiece measurement

In-process measrmnt. is measurement at the workpiece with a touch probe located in a tool holder of the machine. In the tool list, define your touch probe as a new tool. Use the **Measure probe** tool type. The following cycles for **In-process measrmnt.** are basic cycles for probing functions that you can use to program individually adapted probing sequences.

Switch on measurement G910

G910 activates the selected **Measure probe**.

Parameters:

- **V**: **Tool probe(1)/work probe(0)**
  - 0: Touch probe (for workpiece measurement)
  - 1: Table-mounted touch probe (for tool measurement)
- **D**: **Axis number**

Example: In-process measrmnt.

. . .	
N1 G0 X105 Z-20	
N2 G94 F500	
N3 G910 V0 D1	
N4 G911 V0	
N5 G1 Xi-10	
N6 G914	
N7 G912 Q1	
N8 G913	
N9 G0 X115	
N10 #l1=#a9(X,0)	
N11 IF NDEF(#l1)	
N12 THEN	
N13 PRINT("Probe not reached")	
N14 ELSE	
N4 PRINT ("Result of measurement:",#l1)	
N4 ENDIF	
. . .	

Activating measuring path monitoring G911

G911 activates the measuring path monitoring. Then only a single feed path is permissible.

Parameters:

- **V**: **Type of departure**
  - 0: Axes stay stationary with deflected touch probe
  - 1: Axes automatically retract after deflection of the touch probe
- **R**: **Return path**

Actual-value determination G912

G912 puts the positions at which the touch probe was deflected into the result variables.

Parameters:

- **Q**: **Err. evaluation** when the touch probe is not reached
  - 0: Error evaluation in the NC program, measuring results = **NDEF**
  - 1: Error message of NC, program stops

  The measurement results are available in the following variables: **#a9** (axis,channel)
  - Axis = axis name
  - Channel = channel number, 0 = current channel

Example: Measurement results

...	
N1 #l1=#a9(X,0)	X value of current channel
N2 #l2=#a9(Z,1)	Z value of channel 1
N3 #l3=#a9(Y,0)	Y value of current channel
N4 #l4=#a9(C,0)	C value of current channel
...	

End measuring G913

G913 ends the measuring process.

Deactivating measuring path monitoring G914

G914 deactivates the measuring-path monitoring.

Example: Measuring and compensating workpieces

The control provides subprograms for the measurement of workpieces:

- **measure_pos.ncs** (German dialog texts)
- **measure_pos_e.ncs** (English dialog texts)

The programs require a touch probe as a tool. Beginning from the current position or the defined starting position, the control moves along a measuring path in the entered axis direction. At the end it returns again to the previous position. The result of measurement can be included in error compensation.

The following subprograms are used:

- **measure_pos_move.ncs**
- **_Print_txt_lang.ncs**

Parameters:

- **LA**: **Measurement starting point in X** (diameter value; no input: Current position)
- **LB**: **Measurement starting point in Z** (no input: Current position)
- **LC**: **Type of approach** to the starting point of measurement
  - 0: Diagonal
  - 1: First X, then Z
  - 2: First Z, then X
- **LD**: **Measuring axis**
  - 0: X axis
  - 1: Z axis
  - 2: Y axis
- **LE**: Incremental **Measuring path** – the algebraic sign defines the direction
- **LF**: **Measuring feed** in mm/min (no input: the measuring feed rate from the touch probe table will be used)
- **LH**: Nominal value of the **Target position**
- **LI**: **Tolerance +/-** – if the measured deviation lies within this tolerance, the entered compensation value will not be changed
- **LJ**: 1: The **measurement result** is output as **PRINT**
- **LK**: **Number of the compensation value** to be changed
  - 1-xx Turret pocket number of the tool to be compensated
  - 901-916 Additive compensation number
  - Current tool number for touch probe calibration
- **LO**: **Number** of measurements
  - **LO** > 0: The measurements are evenly distributed on the circumference with **M19**
  - **LO** < 0: The measurements are made at the same position
- **LP**: Maximum permissible **difference** between the measurement results at a position
  The program stops if the limit is violated.
- **LR**: Maximum permissible **compensation value** (range: < 10)
- **LS**: 1: For test purposes, when the program runs on the PC, measurement results are interrogated through **INPUT**

8

ISO Programming for the Y Axis (Option 70)

8.1 Y-axis contours – fundamentals

Position of milling contours

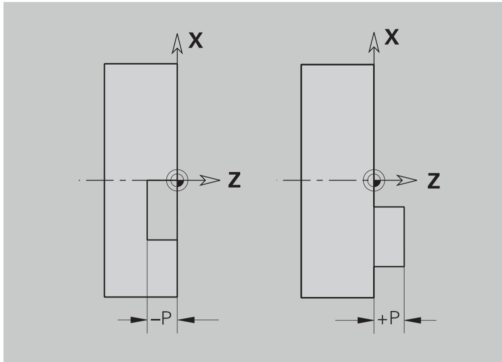
Define the reference plane and the reference diameter in the section code.

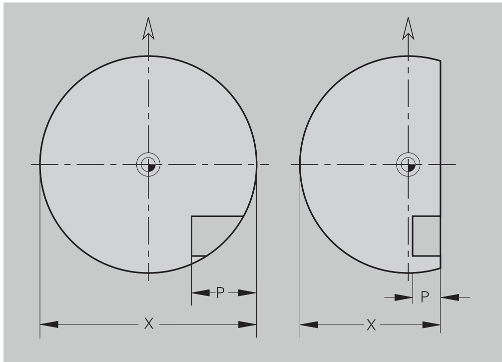
Specify the depth and position of a milling contour (pocket, island) in the contour definition:

- With **Depth P** in the previously programmed **G308** cycle
- Alternatively on figures: Cycle parameter **Depth P**

The **algebraic sign of P** defines the position of the milling contour:

- **P** < 0: Pocket
- **P** > 0: Island





Position of milling contour

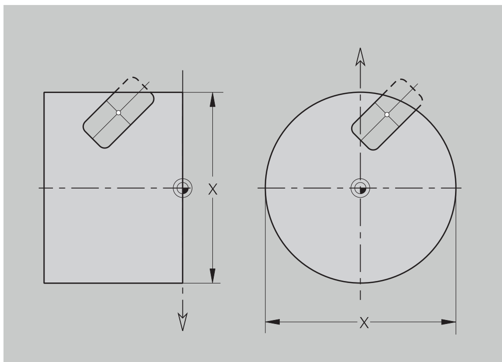
Section	P	Surface	Milling floor
FRONT	**P** < 0	**Z**	**Z** + **P**
	P > 0	**Z** + **P**	**Z**
REAR SIDE	**P** < 0	**Z**	**Z** – **P**
	P > 0	**Z** – **P**	**Z**
LATERAL	**P** < 0	**X**	**X** + (**P** * 2)
	P > 0	**X** + (**P** * 2)	**X**

- **X**: Reference diameter from the section code
- **Z**: Reference plane from the section code
- **P**: Depth from **G308** or from the figure definition

> The area milling cycles mill the surface specified in the contour definition. **Islands** within this surface are not taken into consideration.

Cutting limit

If parts of the milling contour lie outside of the turning contour, you must limit the machining area with the **area diameter X** / **reference diameter X** (parameters of the section code or of the figure definition).



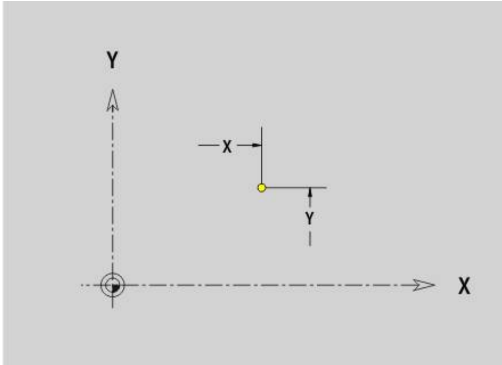
8.2 Contours in the XY plane

Starting point of contour in XY plane G170-Geo

G170 defines the **Start point** of a contour in the XY plane.

Parameters:

- **X**: **Start point** of contour (radius value)
- **Y**: **Start point** of contour
- **PZ**: **Start point** (polar radius)
- **W**: **Start point** (polar angle)

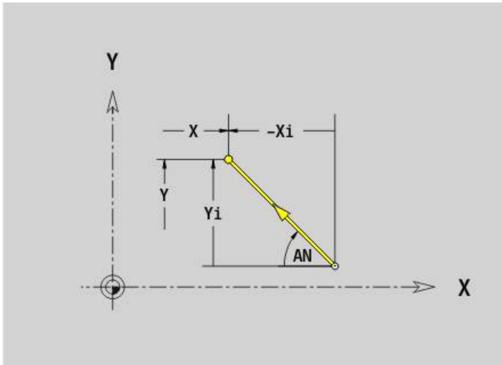


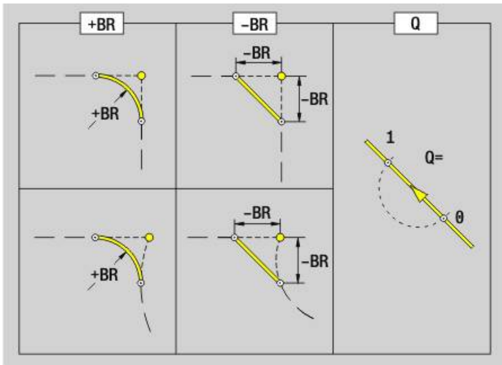
Line segment in XY plane G171-Geo

G171 defines a line segment in a contour of the XY plane.

Parameters:

- **X**: **Final point** (radius value)
- **Y**: **Final point**
- **AN**: **Angle** to X axis
- **Q**: **Intersect. pt.** or **Final point** if the line segment intersects a circular arc (default: 0)
  - 0: Near point of intersection
  - 1: Far point of intersection
- **BR**: **Chamf./round.** – defines the transition to the next contour element
  When entering a **Chamf./round.**, program the theoretical end point.
  - No entry: Tangential transition
  - **BR** = 0: No tangential transition
  - **BR** > 0: Rounding radius
  - **BR** < 0: Width of chamfer
- **PZ**: **Final point** (polar radius; reference: workpiece datum)
- **W**: **Final point** (polar angle; reference: workpiece datum)
- **AR**: **Incr.angle to foregoer ARi** (**AR** corresponds to **AN**)
- **R**: **Line length**





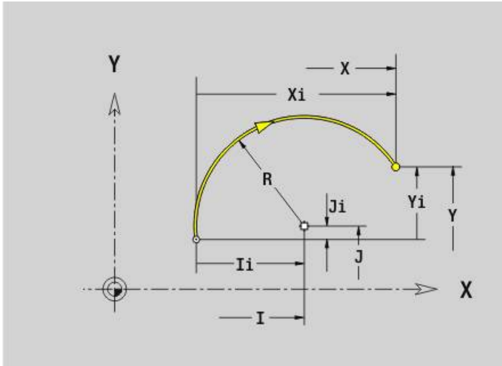
> Programming:
> - **X**, **Y**: Absolute, incremental, modal or **?**
> - **ANi**: Angle to the subsequent element
> - **ARi**: Angle to the previous element

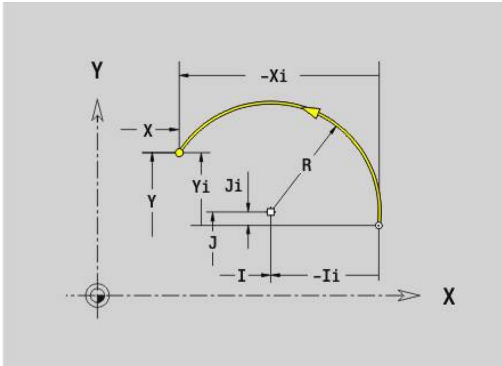
Circular arc in XY plane G172-/G173-Geo

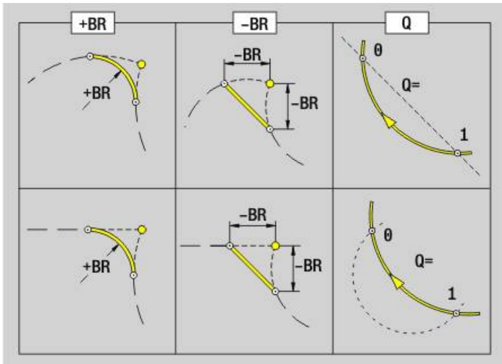
G172 and **G173** define a circular arc in a contour of the XY plane.

Parameters:

- **X**: **Final point** (radius value)
- **Y**: **Final point**
- **R**: **Radius**
- **I**: **Center** in X direction (radius value)
- **J**: **Center** (in Y)
- **Q**: **Intersect. pt.** or **Final point** if the line segment intersects a circular arc (default: 0)
  - 0: Near point of intersection
  - 1: Far point of intersection
- **BR**: **Chamf./round.** – defines the transition to the next contour element
  When entering a **Chamf./round.**, program the theoretical end point.
  - No entry: Tangential transition
  - **BR** = 0: No tangential transition
  - **BR** > 0: Rounding radius
  - **BR** < 0: Width of chamfer
- **PZ**: **Final point** (polar radius; reference: workpiece datum)
- **W**: **Final point** (polar angle; reference: workpiece datum)
- **PM**: **Center** (polar radius; reference: workpiece datum)
- **WM**: **Center** (polar angle; reference: workpiece datum)
- **AR**: **Start angle** – tangential angle to rotary axis
- **AN**: **Final angle** – tangential angle to rotary axis







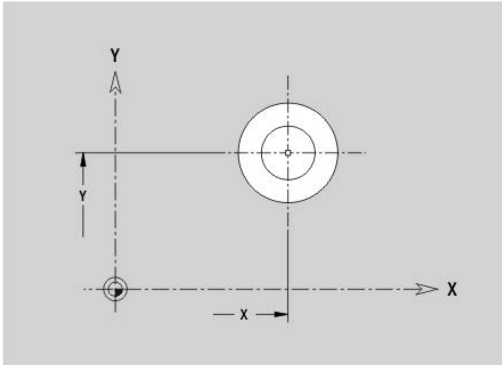
> **i** Programming:
> - **X**, **Y**: Absolute, incremental, modal or **?**
> - **I**, **J**: Absolute or incremental
> - **PZ**, **W**, **PM**, **WM**: Absolute or incremental
> - **ANi**: Angle to the subsequent element
> - **ARi**: Angle to the previous element
> - End point must not be the starting point (**no full circle**)

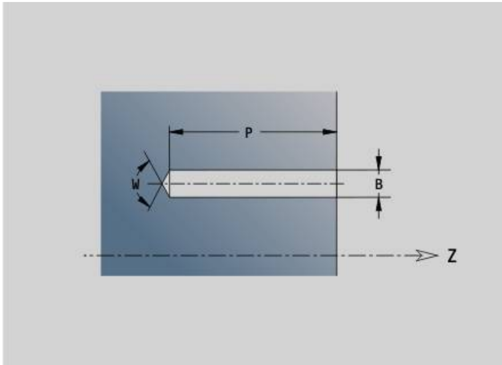
Hole in XY plane G370-Geo

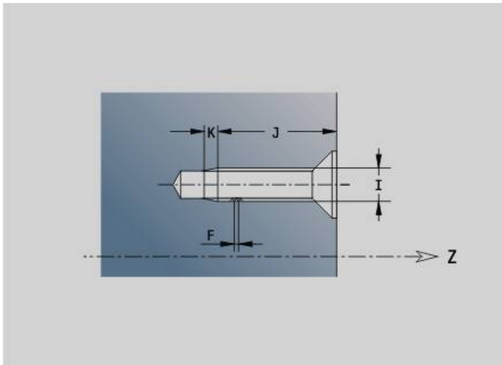
G370 defines a hole with countersinking and thread in the XY plane.

Parameters:

- **X**: **Center** of hole (radius value)
- **Y**: **Center** of hole
- **B**: **Diameter**
- **P**: **Depth** excluding point
- **W**: **Point angle** (default: 180°)
- **R**: **Sink diam.**
- **U**: **Sink depth**
- **E**: **Sink angle**
- **I**: **Thread diameter**
- **J**: **Thread depth**
- **K**: **Start of thread** – run-out length
- **F**: **Thread pitch**
- **V**: **Direction of thread:** (default: 0)
  - **0: Right-hand thread**
  - **1: Left-hand thread**
- **A**: **Angle** to Z axis – angle of the hole
  - Front face (range: –90° < **A** < 90°; default: 0°)
  - Rear face (range: 90° < **A** < 270°; default: 180°)
- **O**: **Center. diam.**





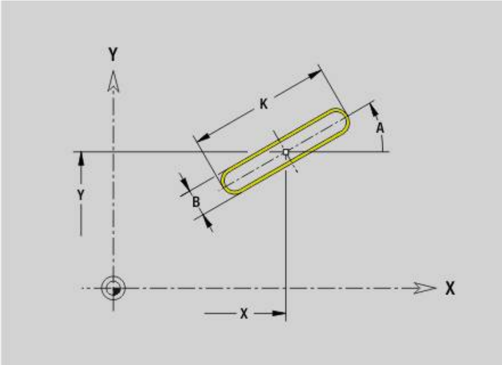


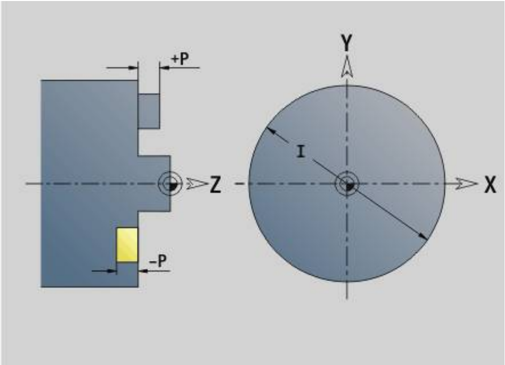
Linear slot in XY plane G371-Geo

G371 defines the contour of a linear slot in the XY plane.

Parameters:

- **X**: **Center** of slot (radius value)
- **Y**: **Center** of slot
- **A**: **Position angle** (reference: positive X axis; default: 0°)
- **K**: **Length**
- **B**: **Width**
- **P**: **Depth/Height** (default: **P** from **G308**)
  - **P** < 0: Pocket
  - **P** > 0: Island
- **I**: **Limit diameter** (as cutting limit)
  - No input: **X** from section code
  - **I** overwrites **X** from section code





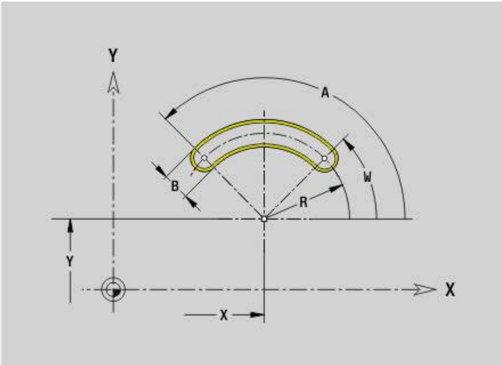
Circular slot in XY plane G372/G373-Geo

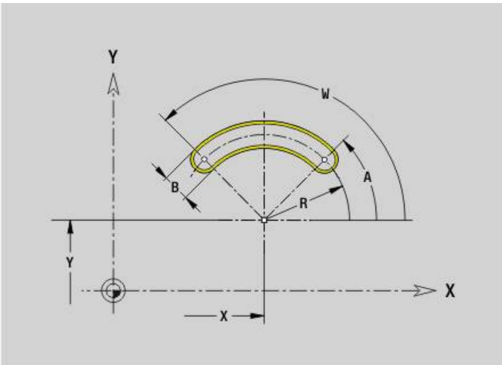
G372 and **G373** define a circular slot in the XY plane.

- **G372**: Circular slot clockwise
- **G373**: Circular slot counterclockwise

Parameters:

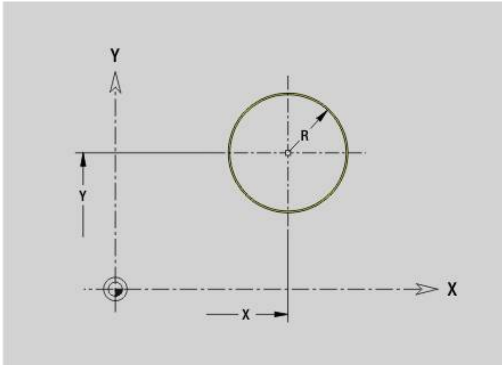
- **X**: **Center** of slot (radius value)
- **Y**: **Center** of slot
- **R**: **Radius** – Curvature radius (reference: center point path of the slot)
- **A**: **Start angle** (reference: positive X axis; default: 0°)
- **W**: **Final angle** (reference: positive X axis; default: 0°)
- **B**: **Width**
- **P**: **Depth/Height** (default: **P** from **G308**)
  - **P** < 0: Pocket
  - **P** > 0: Island
- **I**: **Limit diameter** (as cutting limit)
  - No input: **X** from section code
  - **I** overwrites **X** from section code





Full circle in XY plane G374-Geo

G374 defines a **Full circle** in the XY plane.

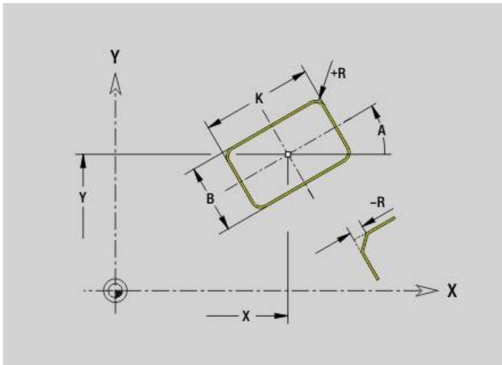


Parameters:

- **X**: **Center** (radius value)
- **Y**: **Center**
- **R**: **Radius**
- **P**: **Depth/Height** (default: **P** from **G308**)
  - **P** < 0: Pocket
  - **P** > 0: Island
- **I**: **Limit diameter** (as cutting limit)
  - No input: **X** from section code
  - **I** overwrites **X** from section code

Rectangle in XY plane G375-Geo

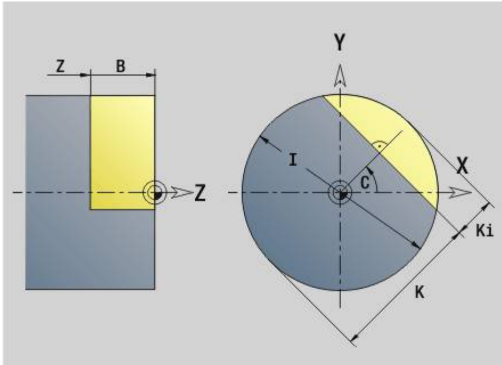
G375 defines a rectangle in the XY plane.



Parameters:

- **X**: **Center** of rectangle (radius value)
- **Y**: **Center** of rectangle
- **A**: **Position angle** (reference: positive X axis; default: 0°)
- **K**: **Length** of rectangle
- **B**: **Width** of rectangle
- **R**: **Chamf./round.** (default: 0)
  - **R** > 0: Radius of rounding arc
  - **R** < 0: Chamfer width
- **P**: **Depth/Height** (default: **P** from **G308**)
  - **P** < 0: Pocket
  - **P** > 0: Island
- **I**: **Limit diameter** (as cutting limit)
  - No input: **X** from section code
  - **I** overwrites **X** from section code

Single surface in XY plane G376-Geo



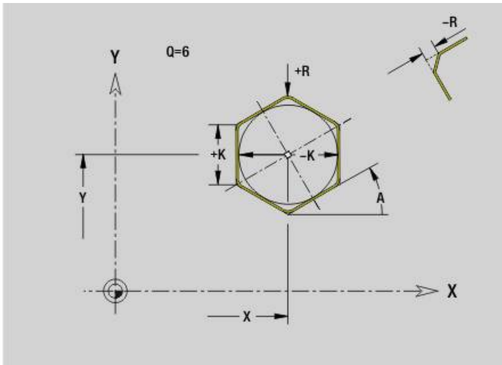
G376 defines a surface in the XY plane.

Parameters:

- **Z**: **Reference edge** (default: **Z** from section code)
- **K**: **Residual depth**
- **Ki**: **Depth**
- **B**: **Width** (reference: **Reference edge Z**)
  - **B** < 0: Surface in negative Z direction
  - **B** > 0: Surface in positive Z direction
- **I**: **Limit diameter** (**as cutting limit** and as reference for **K** and **Ki**)
  - No input: **X** from section code
  - **I** overwrites **X** from section code
- **C**: **Spindle angle** of surface normal (default: **C** from section code)

> Whether the surface lies on the front or rear face has no effect on the evaluation of the algebraic sign for **Width B**.

Polygon in XY plane G377-Geo



G377 defines the contour of an eccentric polygon in the XY plane.

Parameters:

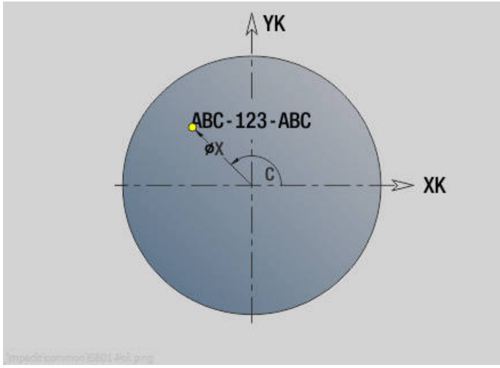
- **X**: **Center** of polygon (radius value)
- **Y**: **Center** of polygon
- **Q**: **Number of corners** (**Q** >= 3)
- **A**: **Position angle** (reference: positive X axis; default: 0°)
- **K**: **+edge lngth/-width a. flats**
  - **K** > 0: Edge length
  - **K** < 0: Width across (Inside diameter)
- **R**: **Chamf./round.** (default: 0)
  - **R** > 0: Radius of rounding arc
  - **R** < 0: Chamfer width
- **P**: **Depth/Height** (default: **P** from **G308**)
  - **P** < 0: Pocket
  - **P** > 0: Island
- **I**: **Limit diameter** (as cutting limit)
  - No input: **X** from section code
  - **I** overwrites **X** from section code

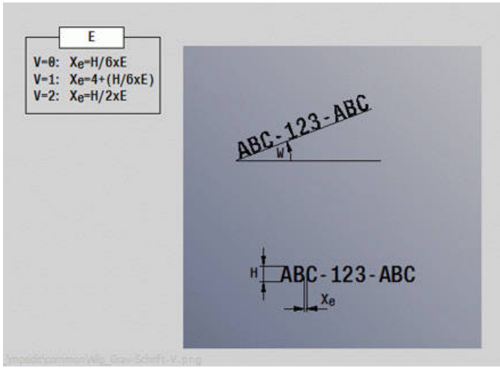
Text for front face Y G378-Geo

G378 defines a text in the XY plane.

Parameters:

- **X**: **Start point** X
- **Y**: **Start point** Y
- **ID**: **Text** to be engraved
- **NF**: **Char. no.** – ASCII code of the character to be engraved
- **P**: **Depth**
- **W**: **Inclinat. ang.** of the character string
- **H**: **Font height**
- **E**: **Distance factor**
  The spacing between the characters is calculated according to the following formula: **H** / 6 * **E**
- **F**: **Plunging feed rate factor** (plunging feed rate = current feed rate * **F**)
- **O**: **Mirror writing**
  - **0** (**No**): Engraving is not mirrored
  - **1** (**Yes**): Engraving is mirrored (mirror writing)





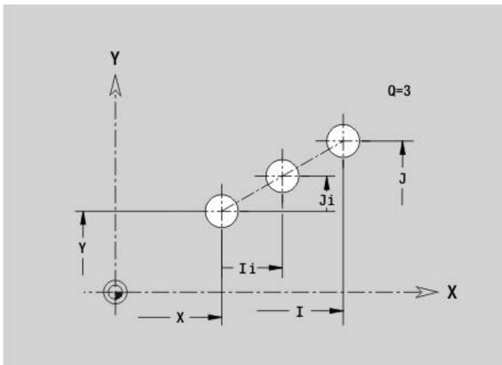
Linear pattern in XY plane G471-Geo

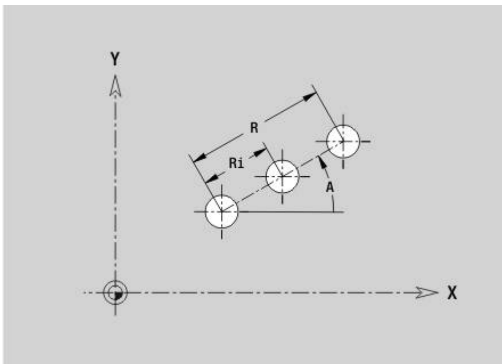
G471 defines a linear pattern in the XY plane.

G471 affects the hole or figure defined in the following block (**G370**-**G375**, **G377**).

Parameters:

- **Q**: **Number** of figures
- **X**: **1st point of pattern** (radius)
- **Y**: **1st point of pattern**
- **I**: **Final point** of pattern (in X; radius value)
- **J**: **Final point** of pattern (in Y)
- **Ii**: **Final point** – distance between two figures (in X)
- **Ji**: **Final point** – distance between two figures (in Y)
- **A**: **Position angle** of longitudinal axis of pattern (reference: positive X axis)
- **R**: **Length** – total length of pattern
- **Ri**: **Length** – distance between two figures





Programming notes:

- Program the hole or figure in the following block without a center
- In the **MACHINING** section, the drilling or milling cycle calls the hole or figure in the following block—not the pattern definition

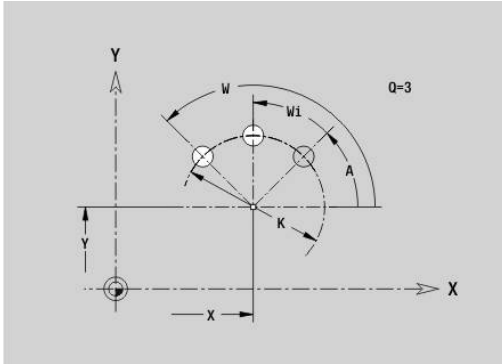
Circular pattern in XY plane G472-Geo

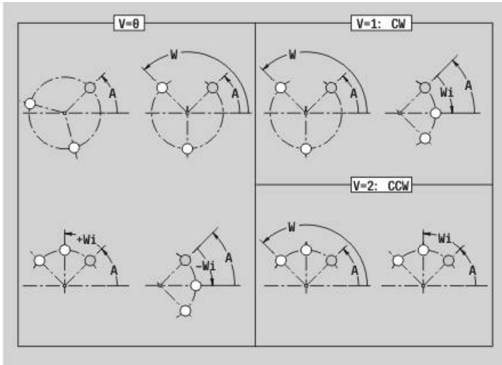
G472 defines a circular pattern in the XY plane.

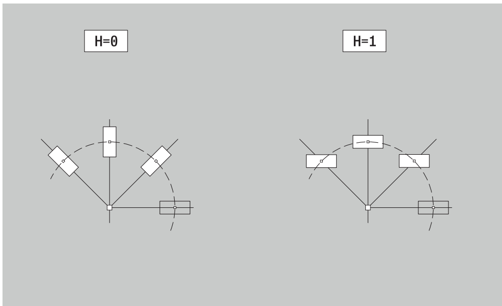
G472 is effective for the figure defined in the following block (**G370**-**G375**, **G377**).

Parameters:

- **Q**: **Number** of figures
- **K**: **Diameter** – pattern diameter
- **A**: **Start angle** – position of the first figure (reference: positive X axis; default: 0°)
- **W**: **Final angle** – position of the last figure (reference: positive X axis; default: 360°)
- **Wi**: **Final angle** – **Angle** between two figures
- **V**: **Direction** – orientation (default: 0)
  - **V** = 0, without **W**: Figures are arranged on a full circle
  - **V** = 0, with **W**: Figures are arranged on the longer circular arc
  - **V** = 0, with **W**: The algebraic sign of **Wi** defines the direction (**W** < 0: clockwise)
  - **V** = 1, with **W**: Clockwise
  - **V** = 1, with **W**: Clockwise (algebraic sign of **W** has no effect)
  - **V** = 2, with **W**: Counterclockwise
  - **V** = 2, with **W**: Counterclockwise (algebraic sign of **W** has no effect)
- **X**: **Center** of pattern (radius value)
- **Y**: **Center** of pattern
- **H**: **0=Normal position** – position of the figures (default: 0)
  - **0**: Normal position – 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)







> Programming notes:
>
> - Program the hole or figure in the following block without a center. Exception: circular slot
>   **Further information:** "Circular pattern with circular slots", Page 312
> - The drilling or milling cycle (**MACHINING** section) calls the hole or figure in the following block—not the pattern definition

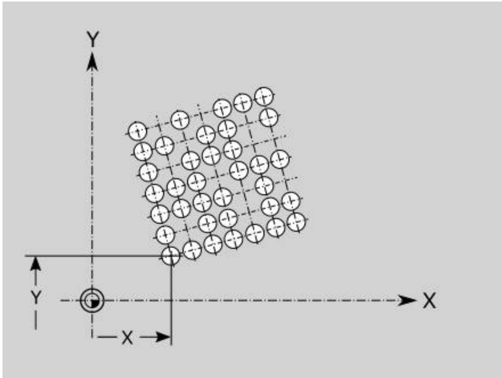
DataMatrix pattern in XY plane G475-Geo

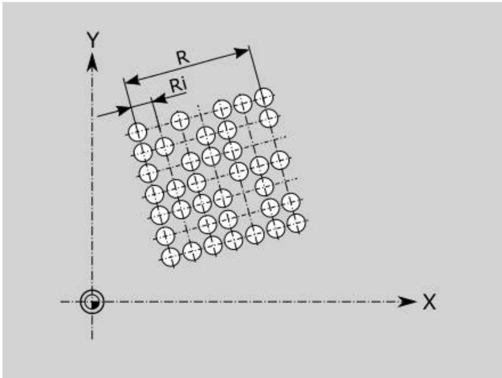
G475 defines a pattern in DataMatrix code in the XY plane.

G475 is effective for the hole or figure defined in the following block (**G370**, **G374**, **G375**, or **G377**).

Parameters:

- **ID**: **Text** to be converted into DataMatrix code
- **X**: **1st point of pattern** (radius)
- **Y**: **1st point of pattern**
- **A**: **Position angle** of longitudinal axis of pattern (reference: positive X axis)
- **R**: **Length** – total length of pattern
- **Ri**: **Length** – distance to next hole or figure





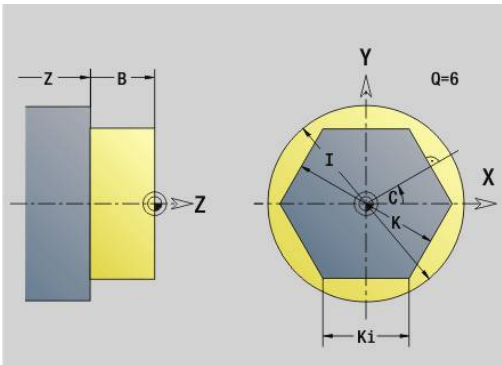
> Programming notes
> - If you do not enter the length, the control calculates the pattern so that the holes or figures touch each other.
> - Program the hole or figure in the following block without a center
> - In the **MACHINING** section, the drilling or milling cycle calls the hole or figure in the following block—not the pattern definition
> - A maximum of 80 ASCII characters are allowed per DataMatrix code
> - The G codes for rectangle and polygon are restricted to a square shape

Centric polygon in XY plane G477-Geo

G477 defines polygonal surfaces in the XY plane.

Parameters:

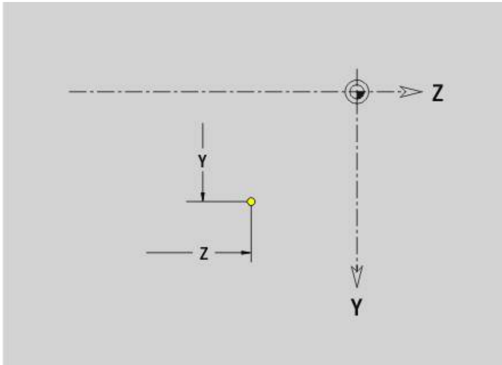
- **Z**: **Reference edge** (default: **Z** from section code)
- **K**: **Width across** – Inscribed circle diameter
- **Ki**: **Edge length**
- **B**: **Width** (reference: **Reference edge Z**)
  - **B** < 0: Surface in negative Z direction
  - **B** > 0: Surface in positive Z direction
- **C**: **Spindle angle** of surface normal (default: **C** from section code)
- **Q**: **No. of surfaces** (**Q** >= 2)
- **I**: **Limit diameter** (as cutting limit)
  - No input: **X** from section code
  - **I** overwrites **X** from section code



> Whether the surface lies on the front or rear face has no effect on the evaluation of the algebraic sign for **Width B**.

8.3 Contours in the YZ plane

Starting point of contour in YZ plane G180-Geo

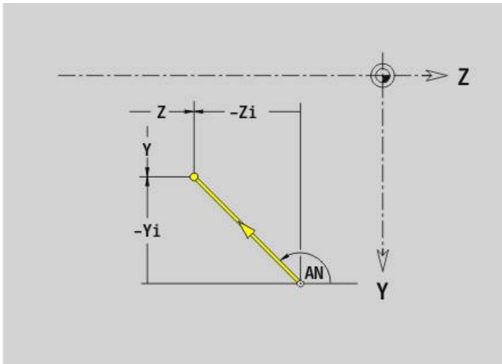


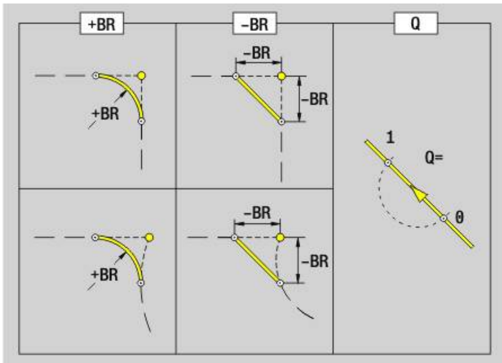
G180 defines the **Start point** of a contour in the YZ plane.

Parameters:

- **Y**: **Start point** of contour
- **Z**: **Start point** of contour
- **PZ**: **Start point** (polar radius)
- **W**: **Start point** (polar angle)

Line segment in YZ plane G181-Geo





G181 defines a line segment in a contour of the YZ plane.

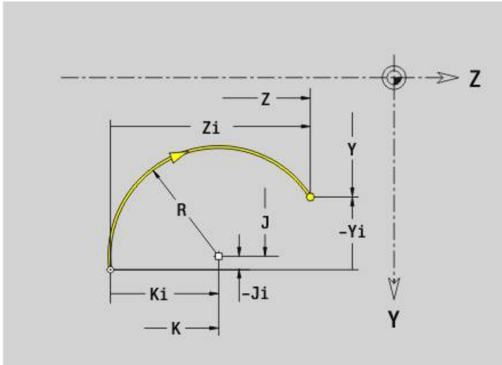
Parameters:

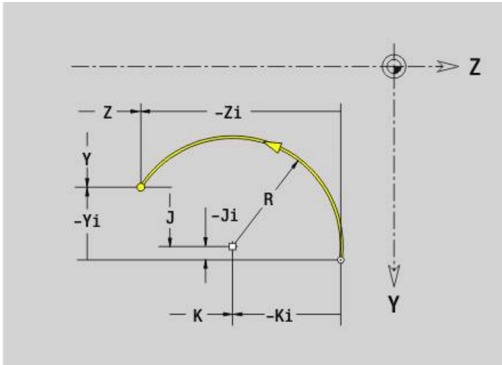
- **Y**: **Final point**
- **Z**: **Final point**
- **AN**: **Angle** to positive Z axis
- **Q**: **Intersect. pt.** or **Final point** if the line segment intersects a circular arc (default: 0)
  - 0: Near point of intersection
  - 1: Far point of intersection
- **BR**: **Chamf./round.** – defines the transition to the next contour element
  When entering a **Chamf./round.**, program the theoretical end point.
  - No entry: Tangential transition
  - **BR** = 0: No tangential transition
  - **BR** > 0: Rounding radius
  - **BR** < 0: Width of chamfer
- **PZ**: **Final point** (polar radius; reference: workpiece datum)
- **W**: **Final point** (polar angle; reference: workpiece datum)
- **AR**: **Incr.angle to foregoer ARi** (**AR** corresponds to **AN**)
- **R**: **Line length**

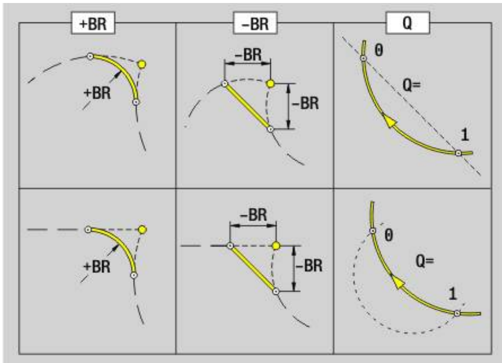
> Programming:
> - **Y**, **Z**: Absolute, incremental, modal or **?**
> - **ANi**: Angle to the subsequent element
> - **ARi**: Angle to the previous element

Circular arc in YZ plane G182/G183-Geo

G182 and **G183** define a circular arc in a contour of the YZ plane.







Parameters:

- **Y**: **Final point**
- **Z**: **Final point**
- **R**: **Radius**
- **J**: **Center** (in Y)
- **K**: **Center** (in Z)
- **Q**: **Intersect. pt.** or **Final point** if the line segment intersects a circular arc (default: 0)
  - 0: Near point of intersection
  - 1: Far point of intersection
- **BR**: **Chamf./round.** – defines the transition to the next contour element
  When entering a **Chamf./round.**, program the theoretical end point.
  - No entry: Tangential transition
  - **BR** = 0: No tangential transition
  - **BR** > 0: Rounding radius
  - **BR** < 0: Width of chamfer
- **PZ**: **Final point** (polar radius; reference: workpiece datum)
- **W**: **Final point** (polar angle; reference: workpiece datum)
- **PM**: **Center** (polar radius; reference: workpiece datum)
- **WM**: **Center** (polar angle; reference: workpiece datum)
- **AR**: **Start angle** – tangential angle to rotary axis
- **AN**: **Final angle** – tangential angle to rotary axis

Programming:

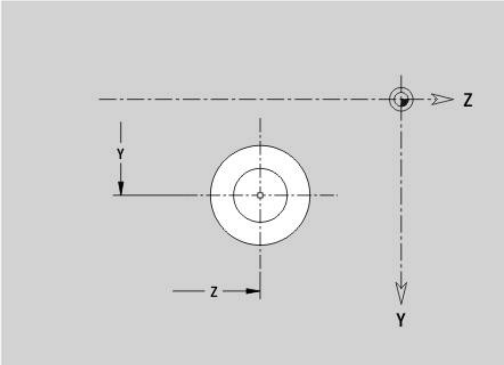
- **Y**, **Z**: Absolute, incremental, modal or **?**
- **J**, **K**: Absolute or incremental
- **PZ**, **W**, **PM**, **WM**: Absolute or incremental
- **ANi**: Angle to the subsequent element
- **ARi**: Angle to the previous element
- End point must not be the starting point (**no full circle**)

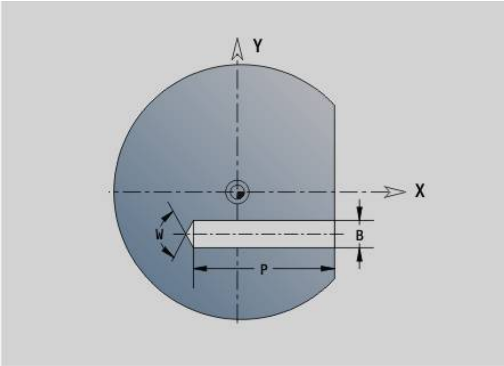
Hole in YZ plane G380-Geo

G380 defines a hole with countersinking and thread in the YZ plane.

Parameters:

- **Y**: **Center** of hole
- **Z**: **Center** hole
- **B**: **Diameter**
- **P**: **Depth** excluding point
- **W**: **Point angle** (default: 180°)
- **R**: **Sink diam.**
- **U**: **Sink depth**
- **E**: **Sink angle**
- **I**: **Thread diameter**
- **J**: **Thread depth**
- **K**: **Start of thread** – run-out length
- **F**: **Thread pitch**
- **V**: **Direction of thread:** (default: 0)
  - **0: Right-hand thread**
  - **1: Left-hand thread**
- **A**: **Angle** to X axis (range: –90° < **A** < 90°)
- **O**: **Center. diam.**



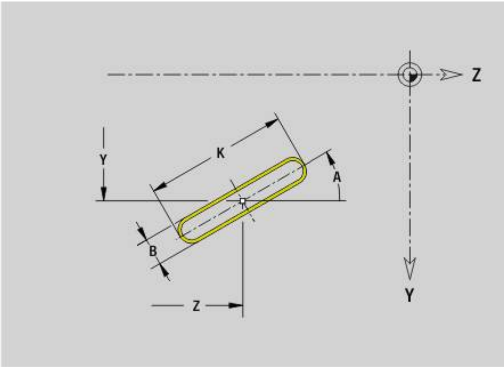


Linear slot in YZ plane G381-Geo

G381 defines the contour of a linear slot in the YZ plane.

Parameters:

- **Y**: **Center** of slot
- **Z**: **Center** of slot
- **X**: **Reference diameter**
  - No input: **X** from section code
  - **X** overwrites **X** from section code
- **A**: **Position angle** (reference: positive Z axis; default: 0°)
- **K**: **Length**
- **B**: **Width**
- **P**: **Depth/Height** (default: **P** from **G308**)



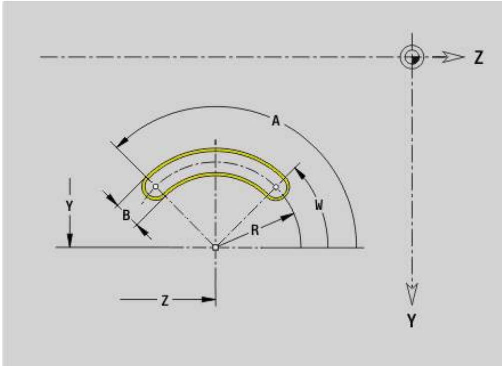
Circular slot in YZ plane G382/G383-Geo

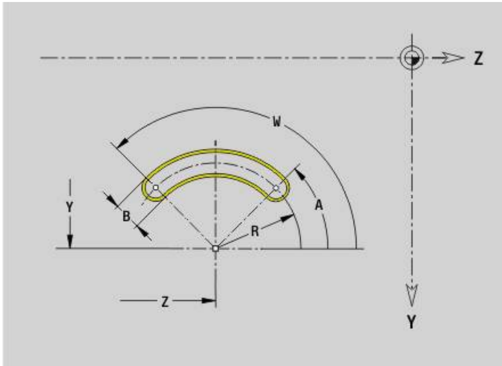
G382 and **G383** define a circular slot in the YZ plane.

- **G382**: Circular slot clockwise
- **G383**: Circular slot counterclockwise

Parameters:

- **Z**: **Center** of slot
- **Y**: **Center** of slot
- **X**: **Reference diameter**
  - No input: **X** from section code
  - **X** overwrites **X** from section code
- **R**: **Radius**
- **A**: **Start angle** (reference: positive X axis; default: 0°)
- **W**: **Final angle** (reference: positive X axis; default: 0°)
- **B**: **Width**
- **P**: **Depth/Height** (default: **P** from **G308**)



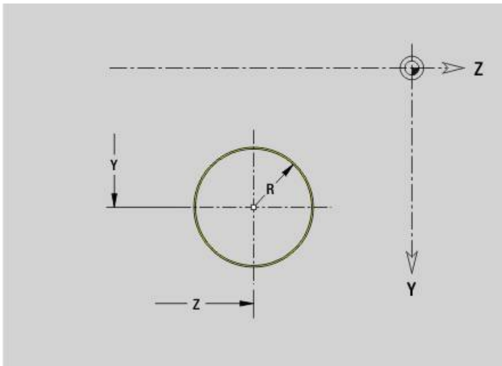


Full circle in YZ plane G384-Geo

G384 defines a full circle in the YZ plane.

Parameters:

- **Z**: **Center**
- **Y**: **Center**
- **X**: **Reference diameter**
  - No input: **X** from section code
  - **X** overwrites **X** from section code
- **R**: **Radius**
- **P**: **Depth/Height** (default: **P** from **G308**)

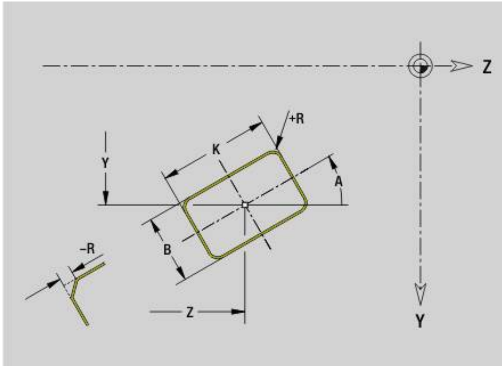


Rectangle in YZ plane G385-Geo

G385 defines a rectangle in the YZ plane.

Parameters:

- **Z**: **Center** of rectangle
- **Y**: **Center** of rectangle
- **X**: **Reference diameter**
  - No input: **X** from section code
  - **X** overwrites **X** from section code
- **A**: **Position angle** (reference: positive Z axis; default: 0°)
- **K**: **Length** of rectangle
- **B**: **Width** of rectangle
- **R**: **Chamf./round.** (default: 0)
  - **R** > 0: Radius of rounding arc
  - **R** < 0: Chamfer width
- **P**: **Depth/Height** (default: **P** from **G308**)

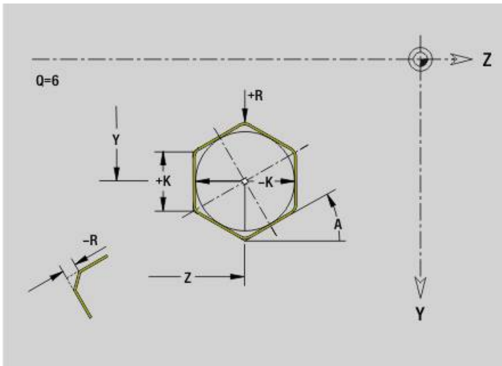


Polygon in YZ plane G387-Geo

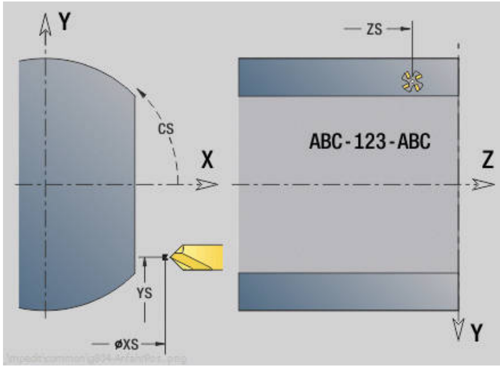
G387 defines the contour of an eccentric polygon in the YZ plane.

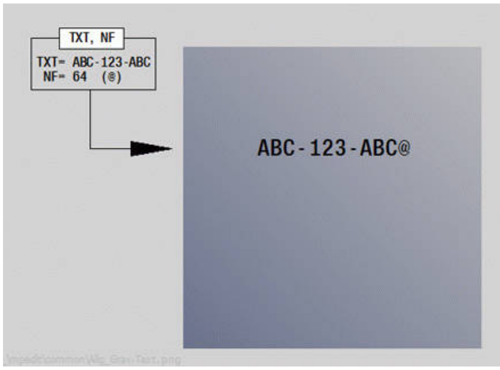
Parameters:

- **Z**: **Center** of polygon
- **Y**: **Center** of polygon
- **X**: **Reference diameter**
  - No input: **X** from section code
  - **X** overwrites **X** from section code
- **Q**: **Number of corners** (**Q** >= 3)
- **A**: **Position angle** (reference: positive Z axis; default: 0°)
- **K**: **+edge lngth/-width a. flats**
  - **K** > 0: Edge length
  - **K** < 0: Width across (Inside diameter)
- **R**: **Chamf./round.** (default: 0)
  - **R** > 0: Radius of rounding arc
  - **R** < 0: Chamfer width
- **P**: **Depth/Height** (default: **P** from **G308**)



Text for lateral surface Y G388-Geo



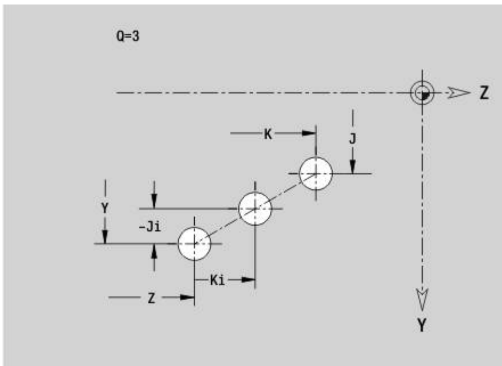


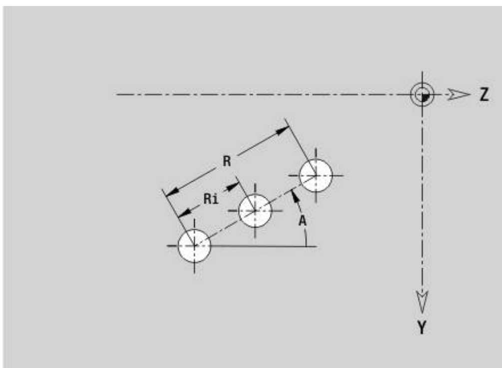
G388 defines a text in the YZ plane.

Parameters:

- **Y**: **Start point** Y
- **Z**: **Start point** Z
- **ID**: **Text** to be engraved
- **NF**: **Char. no.** – ASCII code of the character to be engraved
- **P**: **Depth**
- **W**: **Inclinat. ang.** of the character string
- **H**: **Font height**
- **E**: **Distance factor**
  The spacing between the characters is calculated according to the following formula: **H** / 6 * **E**
- **F**: **Plunging feed rate factor** (plunging feed rate = current feed rate * **F**)
- **O**: **Mirror writing**
  - **0** (**No**): Engraving is not mirrored
  - **1** (**Yes**): Engraving is mirrored (mirror writing)

Linear pattern in YZ plane G481-Geo





G481 defines a linear pattern in the YZ plane.

G481 affects the hole or figure defined in the following block (**G380**-**G385**, **G387**).

Parameters:

- **Q**: **Number** of figures
- **Z**: **1st pattern point**
- **Y**: **1st point of pattern**
- **K**: **Final point** of pattern (Z direction)
- **J**: **Final point** of pattern (in Y)
- **Ki**: **Final point** – distance between two figures (in Z)
- **Ji**: **Final point** – distance between two figures (in Y)
- **A**: **Position angle** (reference: positive Z axis; default: 0°)
- **R**: **Length** – total length of pattern
- **Ri**: **Length** – distance between two figures

Programming notes:

- Program the hole or figure in the following block without a center
- In the **MACHINING** section, the drilling or milling cycle calls the hole or figure in the following block—not the pattern definition

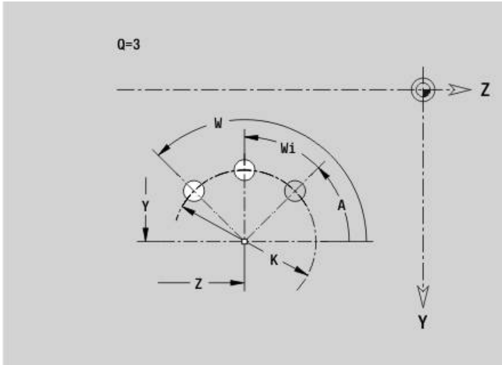
Circular pattern in YZ plane G482-Geo

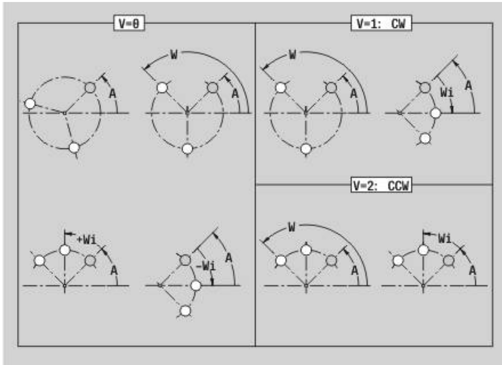
G482 defines a circular pattern in the YZ plane.

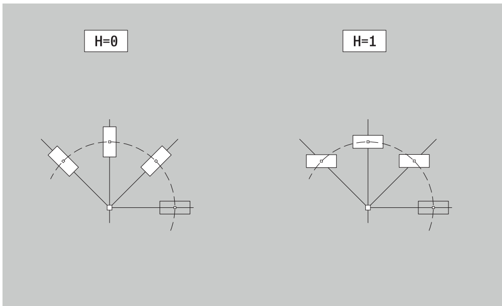
G482 is effective for the figure defined in the following block (**G380**-**G385**, **G387**).

Parameters:

- **Q**: **Number** of figures
- **K**: **Diameter** – pattern diameter
- **A**: **Position angle** (reference: positive Z axis; default: 0°)
- **W**: **Final angle** – position of the last figure (reference: positive Z axis; (default: 360°)
- **Wi**: **Final angle** – **Angle** between two figures
- **V**: **Direction** – orientation (default: 0)
  - **V** = 0, without **W**: Figures are arranged on a full circle
  - **V** = 0, with **W**: Figures are arranged on the longer circular arc
  - **V** = 0, with **W**: The algebraic sign of **Wi** defines the direction (**W** < 0: clockwise)
  - **V** = 1, with **W**: Clockwise
  - **V** = 1, with **W**: Clockwise (algebraic sign of **W** has no effect)
  - **V** = 2, with **W**: Counterclockwise
  - **V** = 2, with **W**: Counterclockwise (algebraic sign of **W** has no effect)
- **Z**: **Center** of pattern
- **Y**: **Center** of pattern
- **H**: **0=Normal position** – position of the figures (default: 0)
  - **0**: Normal position – 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)







Programming notes:

- Program the hole or figure in the following block without a center. Exception: circular slot
  Further information: "Circular pattern with circular slots", Page 312
- The drilling or milling cycle (**MACHINING** section) calls the hole or figure in the following block—not the pattern definition

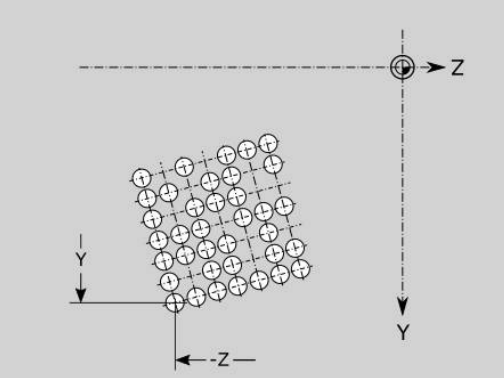
DataMatrix pattern in YZ plane G485-Geo

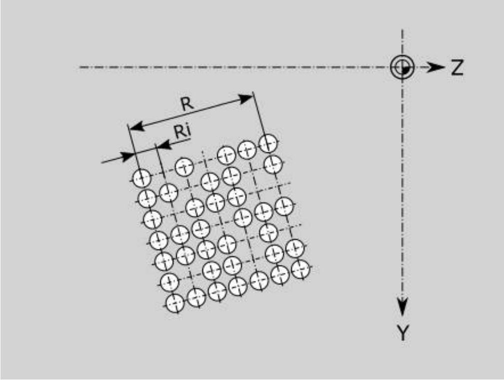
G485 defines a pattern in DataMatrix code in the YZ plane.

G485 is effective for the hole or figure defined in the following block (**G380**, **G384**, **G385**, or **G387**).

Parameters:

- **ID**: **Text** to be converted into DataMatrix code
- **Z**: **1st pattern point**
- **Y**: **1st point of pattern**
- **A**: **Position angle** (reference: positive Z axis; default: 0°)
- **R**: **Length** – total length of pattern
- **Ri**: **Length** – distance to next hole or figure





Programming notes

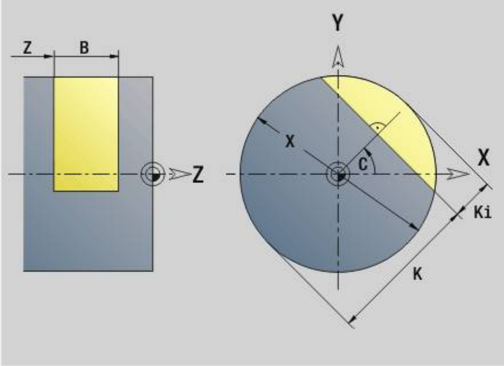
- If you do not enter the length, the control calculates the pattern so that the holes or figures touch each other.
- Program the hole or figure in the following block without a center
- In the **MACHINING** section, the drilling or milling cycle calls the hole or figure in the following block—not the pattern definition
- A maximum of 80 ASCII characters are allowed per DataMatrix code
- The G codes for rectangle and polygon are restricted to a square shape

Single surface in YZ plane G386-Geo

G386 defines a surface in the YZ plane.

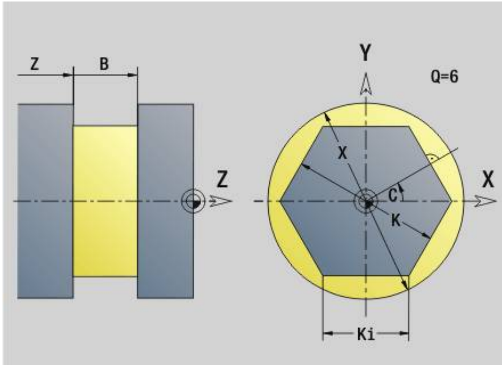
Parameters:

- **Z**: **Reference edge** (default: **Z** from section code)
- **K**: **Residual depth**
- **Ki**: **Depth**
- **B**: **Width** (reference: **Reference edge Z**)
  - **B** < 0: Surface in negative Z direction
  - **B** > 0: Surface in positive Z direction
- **X**: **Reference diameter**
  - No input: **X** from section code
  - **X** overwrites **X** from section code
- **C**: **Spindle angle** of surface normal (default: **C** from section code)



The **Reference diameter X** limits the surface to be machined.

Centric polygon in YZ plane G487-Geo



G487 defines polygonal surfaces in the YZ plane.

Parameters:

- **Z**: **Reference edge** (default: **Z** from section code)
- **K**: **Width across** – Inscribed circle diameter
- **Ki**: **Edge length**
- **B**: **Width** (reference: **Reference edge Z**)
  - **B** < 0: Surface in negative Z direction
  - **B** > 0: Surface in positive Z direction
- **X**: **Reference diameter**
  - No input: **X** from section code
  - **X** overwrites **X** from section code
- **C**: **Spindle angle** of surface normal (default: **C** from section code)
- **Q**: **No. of surfaces** (**Q** >= 2)

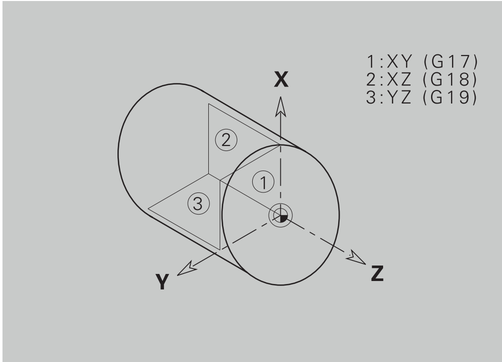
The **Reference diameter X** limits the surface to be machined.

8.4 Working planes

Y-axis machining

When programming drilling or milling operations with the Y axis, you need to define the working plane.

If no working plane is programmed, the control assumes a turning operation or a milling operation with the C axis (**G18** XZ plane).



G17 XY plane (front or rear face)

Milling cycles are executed in the XY plane, with the depth feed for milling and drilling cycles in the Z direction.

G18 XZ plane (turning)

In the XZ plane, normal turning operations as well as drilling and milling operations are executed with the C axis.

G19 YZ plane (lateral view / lateral surface)

Milling cycles are executed in the YZ plane, with the depth feed for milling and drilling cycles in the X direction.

Tilt working plane G16

G16 executes the following transformations and rotations:

- Shifts the coordinate system to the position **I**, **K**
- Rotates the coordinate system by the **Angle B**;
  Reference pt.: **I**, **K**
- Shifts, if programmed, the coordinate system by **U** and **W** in the rotated coordinate system

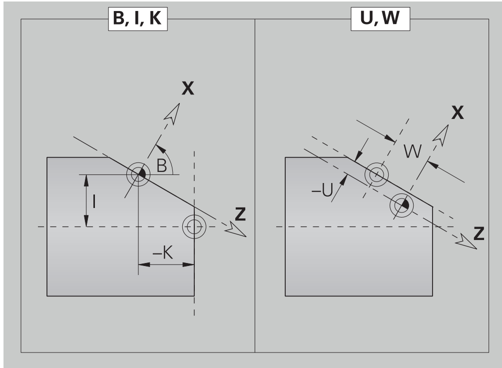
Parameters:

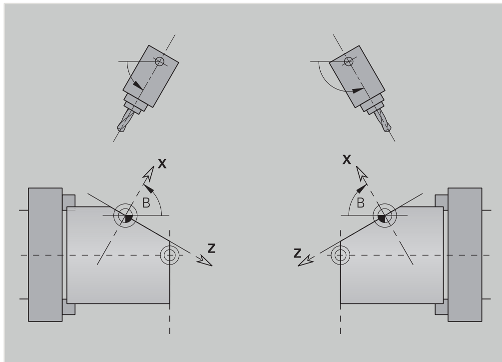
- **B**: **Plane angle** (reference: positive Z axis)
- **I**: **Plane ref. in X** (radius value)
- **K**: **Plane ref. in Z**
- **U**: **Shift in X** (radius value)
- **W**: **Shift in Z**
- **Q**: **On/Off** – enable/disable tilting the working plane
  - 0: Disable tilted working plane function
  - 1: Tilt working plane
  - 2: Restore previous **G16** plane

G16 Q0 resets the working plane. The datum and coordinate system defined before **G16** are then in effect again.

G16 Q2 restores the previous **G16** plane.

The positive Z axis is the reference axis for the **Plane angle B**. This also applies to a mirrored coordinate system.



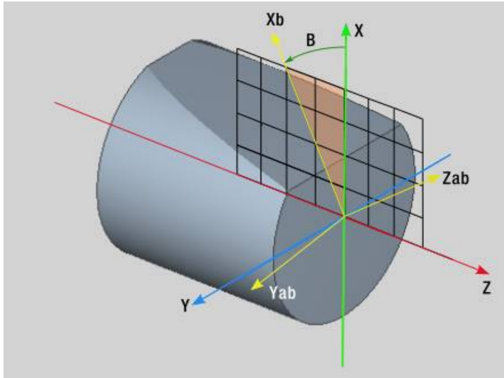


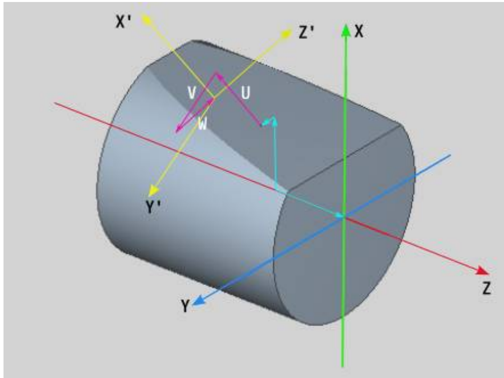
> Please note:
> - **X** is the infeed axis in a tilted coordinate system. X coordinates are entered as diameter coordinates.
> - Mirroring the coordinate system has no effect on the reference axis of the tilt angle (**B axis angle** of the tool call)
> - Other datum shifts are not permitted as long as **G16** is active

Example: G16

. . .	
MACHINING	
. . .	
N.. G19	
N.. G15 B130	
N.. G16 B130 I59 K0 Q1	
N.. G1 X.. Z.. Y..	
N.. G16 Q0	
. . .	

Tilt working plane G16





Use **G160** to tilt the coordinate system conveniently for machining.

G16 executes the following transformations:

- Shifts the coordinate system to the positions **I**, **J** and **K** prior to tilting
- Tilts the coordinate system to the defined **Spatial angle A**, **B** and **C**
  Reference pt.: **I**, **J**, **K**
- Shifts the coordinate system by **U**, **V** and **W** after tilting

Parameters:

- **A**: **Spatial angle**
- **B**: **Spatial angle**
- **C**: **Spatial angle**
- **I**: **Tilting position in X** (radius value)
- **J**: **Tilting position in Y**
- **K**: **Tilting position in Z**
- **U**: **Shift in X** (radius value)
- **V**: **Shift Y**
- **W**: **Shift in Z**

G160 without inputs resets tilting. The control activates the datum and the coordinate system that were active before **G160**.

Before you program further transformations, you have the reset any active tilting function with **G160** or **G16**.

- The control considers mirroring with **G30** for the tilting process.
- The spatial angles **A**, **B** and **C** refer to the axes **X**, **Y** and **Z** in the machine coordinate system.
- If you do not define the **Spatial angle A**, **B** or **C**, the control will use the value 0 for calculating.
- The machine manufacturer defines whether the control will show the current values of the machining plane in the machine data display.
  Further information: User's Manual

8.5 Tool positioning in Y axis

Rapid traverse G0

G0 moves the tool at rapid traverse along the shortest path to the **Target point X**, **Y**, **Z**.

Parameters:

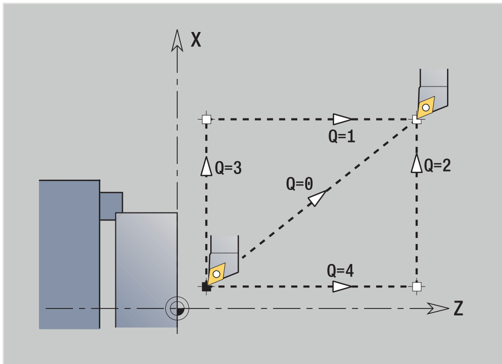
- **X**: **Diameter** – target point
- **Y**: **Length** – target point
- **Z**: **Length** – target point

> Programming:
> - **X**, **Y** and **Z** absolute, incremental or modal

> If more axes are available on your machine, additional input parameters will be displayed, e.g. parameter **B** for the B axis.

Approach the Tool change point G14

G14 moves the tool at rapid traverse to the **Tool change point**. Coordinates for the tool change position can be defined in setup mode.



Parameters:

- **Q**: **Order** (default: 0)
  - **0: Simultaneously**
  - **1: First X, then Z**
  - **2: First Y, then Z, then X**
  - **3: Only X**
  - **4: Only Z**
  - **5: Only Y** (machine-dependent)
  - **6: Simultaneous w/ Y** (machine-dependent)

> If **Q** = 0-4, the Y axis does not move.

Rapid traverse in machine coordinates G701

G701 moves the tool at rapid traverse along the shortest path to the **Destinat. point X**, **Y**, **Z**.

Parameters:

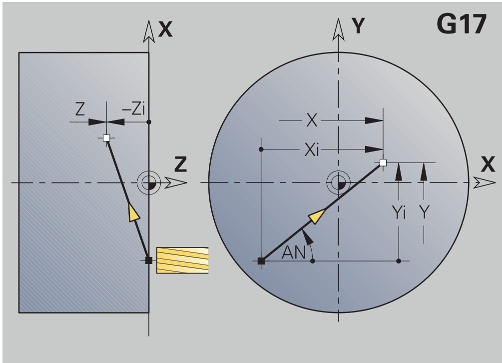
- **X**: **Final point** (diameter value)
- **Y**: **Final point**
- **Z**: **Final point**

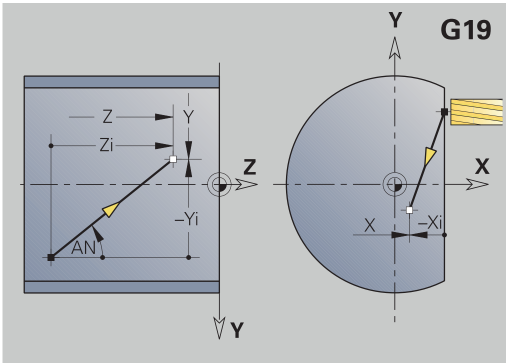
> **X**, **Y** and **Z** refer to the machine datum and the slide reference point.

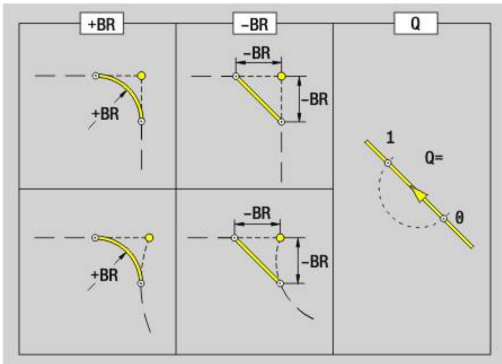
> If more axes are available on your machine, additional input parameters will be displayed, e.g. parameter **B** for the B axis.

8.6 Linear and circular movements in the Y axis

Milling: Linear movement G1







G1 moves the tool on a linear path at the feed rate to the **Final point**.

The execution of **G1** varies depending on the working plane:

- **G17** Interpolation in the XY plane
  - Infeed in Z direction
  - Angle A—reference: positive X axis
- **G18** Interpolation in the XZ plane
  - Infeed in Y direction
  - Angle A—reference: negative Z axis
- **G19** Interpolation in the YZ plane
  - Infeed in X direction
  - Angle A—reference: positive Z axis

Parameters:

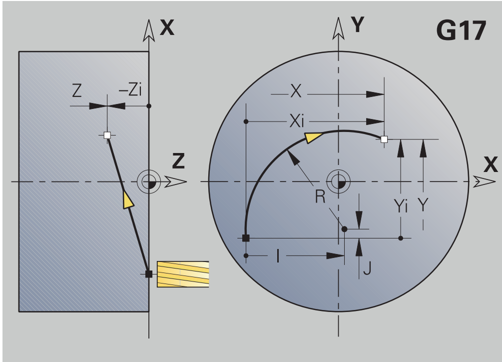
- **X**: **Diameter** – target point
- **Y**: **Length** – target point
- **Z**: **Length** – target point
- **AN**: **Angle** (reference: depends on the working plane)
- **Q**: **Intersect. pt.** or **Final point** if the line segment intersects a circular arc (default: 0)
  - 0: Near point of intersection
  - 1: Far point of intersection
- **BR**: **Chamf./round.** – defines the transition to the next contour element
  When entering a **Chamf./round.**, program the theoretical end point.
  - No entry: Tangential transition
  - **BR** = 0: No tangential transition
  - **BR** > 0: Rounding radius
  - **BR** < 0: Width of chamfer
- **BE**: **Special feed rate factor** for **Chamf./round.** (default: 1)
  Special feed rate = active feed rate * **BE** (range: 0 < **BE** <= 1)

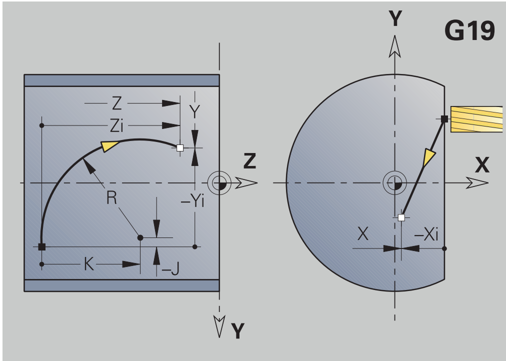
> Programming:
> - **X**, **Y** and **Z** absolute, incremental, modal or **?**

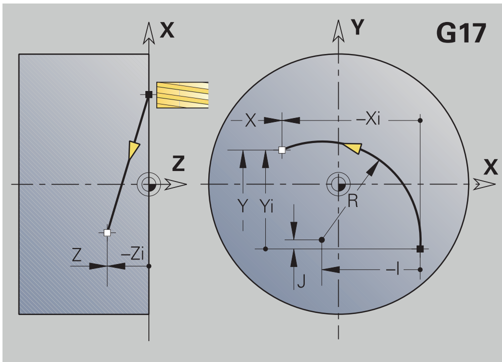
> If more axes are available on your machine, additional input parameters will be displayed, e.g. parameter **B** for the B axis.

Milling: Circular arc cw G2, G3 – incremental center coordinates

G2 and **G3** move the tool on a circular arc to the **Final point** at the specified feed rate.







The execution of **G2** and **G3** varies depending on the working plane:

- **G17** Interpolation in the XY plane
  - Infeed in Z direction
  - Center definition: with **I**, **J**
- **G18** Interpolation in the XZ plane
  - Infeed in Y direction
  - Center definition: with **I**, **K**
- **G19** Interpolation in the YZ plane
  - Infeed in X direction
  - Center definition: with **J**, **K**

Parameters:

- **X**: **Diameter** – target point
- **Y**: **Length** – target point
- **Z**: **Length** – target point
- **I**: **Center, incremental** (radius value)
- **J**: **Center, incremental**
- **K**: **Center, incremental**
- **Q**: **Intersect. pt.** or **Final point** if the line segment intersects a circular arc (default: 0)
  - 0: Near point of intersection
  - 1: Far point of intersection
- **BR**: **Chamf./round.** – defines the transition to the next contour element
  When entering a **Chamf./round.**, program the theoretical end point.
  - No entry: Tangential transition
  - **BR** = 0: No tangential transition
  - **BR** > 0: Rounding radius
  - **BR** < 0: Width of chamfer
- **BE**: **Special feed rate factor** for **Chamf./round.** (default: 1)
  Special feed rate = active feed rate * **BE** (range: 0 < **BE** <= 1)

If you do not program the center, the control automatically calculates the possible solutions for the center and chooses that point as the center which results in the shortest arc.

> Programming:
> - **X**, **Y** and **Z** absolute, incremental, modal or **?**

Milling: Circular arc cw G12, G13 – center coordinates

G12 and **G13** move the tool on a circular arc to the **Final point** at the specified feed rate.

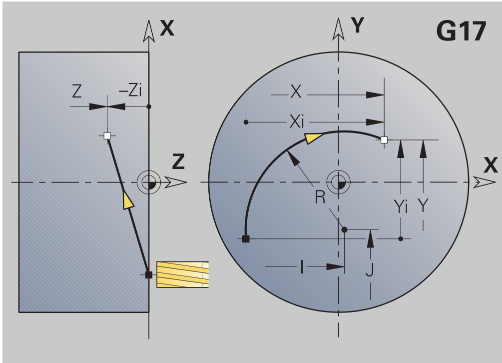
The execution of **G12** and **G13** varies depending on the **working plane**:

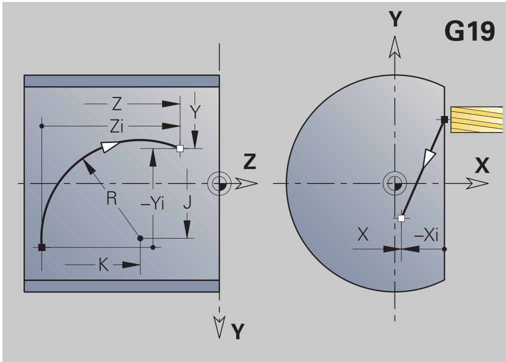
- **G17** Interpolation in the XY plane
  - Infeed in Z direction
  - Center definition: with **I**, **J**
- **G18** Interpolation in the XZ plane
  - Infeed in Y direction
  - Center definition: with **I**, **K**
- **G19** Interpolation in the YZ plane
  - Infeed in X direction
  - Center definition: with **J**, **K**

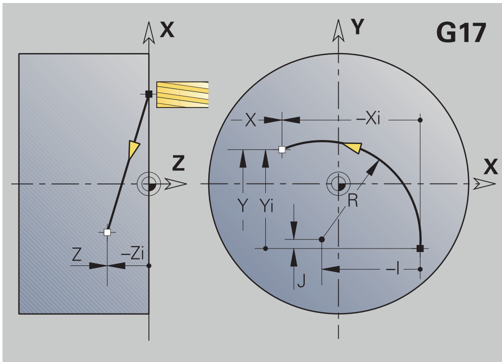
Parameters:

- **X**: **Diameter** – target point
- **Y**: **Length** – target point
- **Z**: **Length** – target point
- **I**: **Center** absolute (radius value)
- **J**: **Center**, absolute
- **K**: **Center** absolute
- **Q**: **Intersect. pt.** or **Final point** if the line segment intersects a circular arc (default: 0)
  - 0: Near point of intersection
  - 1: Far point of intersection
- **BR**: **Chamf./round.** – defines the transition to the next contour element
  When entering a **Chamf./round.**, program the theoretical end point.
  - No entry: Tangential transition
  - **BR** = 0: No tangential transition
  - **BR** > 0: Rounding radius
  - **BR** < 0: Width of chamfer
- **E**: **Special feed rate factor** for the chamfer/rounding (default: 1)
  Special feed rate = active feed rate * **E** (range 0 < **E** <= 1)

If you do not program the center, the control automatically calculates the possible solutions for the center and chooses that point as the center which results in the shortest arc.



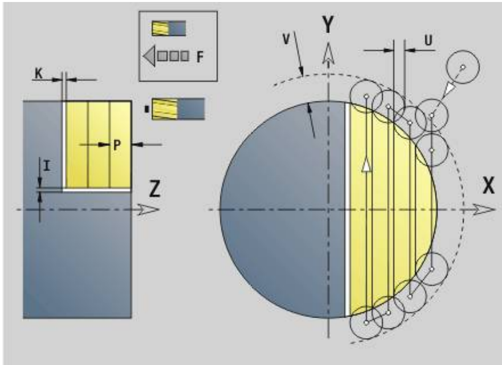


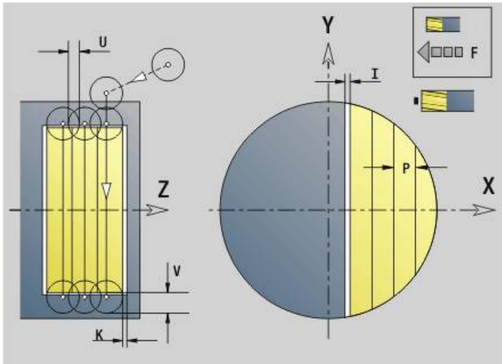


> Programming:
> - **X**, **Y** and **Z** absolute, incremental, modal or **?**

8.7 Milling cycles for the Y axis

Area milling - roughing G841





G841 roughs surfaces defined with **G376**-Geo (XY plane) or **G386**-Geo (YZ plane). The cycle mills from the outside toward the inside. The tool moves to the working plane outside of the workpiece material.

Parameters:

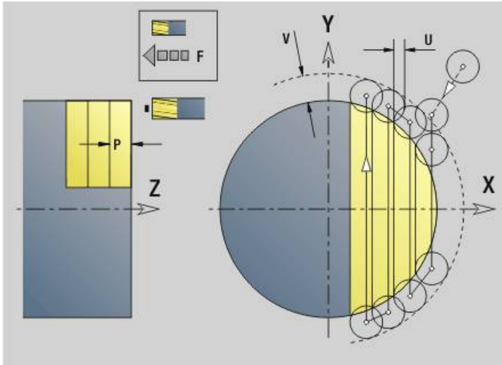
- **ID**: **Milling contour** – name of the milling contour
- **NS**: **Block number of contour** – reference to the contour description
- **P**: **Milling depth** – maximum infeed in the working plane
- **I**: **O-size X**
- **K**: **O-size Z**
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
- **V**: **Overrun factor** – defines the distance by which the tool should pass the outside radius of the workpiece (default: 0.5)
  Overrun = **V** * milling diameter
- **F**: **Approach feed** for plunging (default: active feed rate)
- **RB**: **Return plane** (default: back to start position)
  - XY plane: Retraction position in Z direction
  - YZ plane: Return position in X direction (diameter)

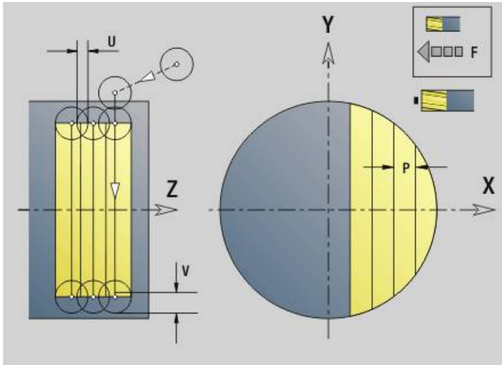
> Oversizes are taken into account:
> - **G57**: Oversize in X, Z direction
> - **G58**: Equidistant oversize in the milling plane

Cycle run

1. Start position (**X**, **Y**, **Z**, **C**) is the position before the cycle begins
2. Calculates the number of cutting passes (infeeds to the milling planes, infeeds in the milling depths)
3. Move to the safety clearance and plunge to the first milling depth.
4. Mill the first plane.
5. Retract by the safety clearance, return and cut to the next milling depth.
6. Repeat steps 4 and 5 until the complete area is milled.
7. Returns to **Return plane RB**

Area milling - finishing G842





G842 finishes surfaces defined with **G376**-Geo (XY plane) or **G386**-Geo (YZ plane). The cycle mills from the outside toward the inside. The tool moves to the working plane outside of the workpiece material.

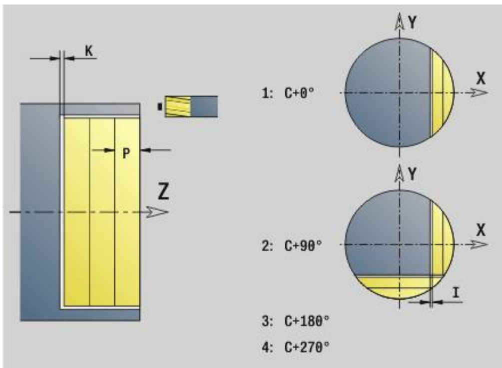
Parameters:

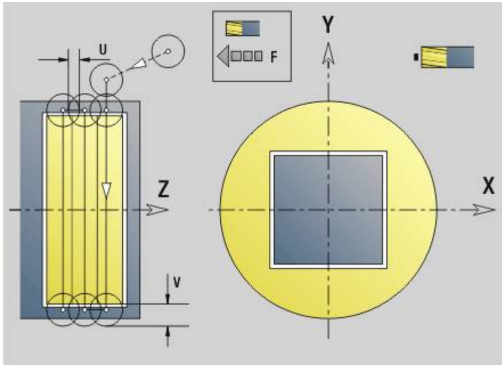
- **ID**: **Milling contour** – name of the milling contour
- **NS**: **Block number of contour** – reference to the contour description
- **P**: **Milling depth** – maximum infeed in the working plane
- **H**: **Mill cutting direction** for side finishing (default: 0)
  - **0: Up-cut**
  - **1: Climb**
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
- **V**: **Overrun factor** – defines the distance by which the tool should pass the outside radius of the workpiece (default: 0.5)
  Overrun = **V** * milling diameter
- **F**: **Approach feed** for plunging (default: active feed rate)
- **RB**: **Return plane** (default: back to start position)
  - XY plane: Retraction position in Z direction
  - YZ plane: Return position in X direction (diameter)

Cycle run

1. Start position (**X**, **Y**, **Z**, **C**) is the position before the cycle begins
2. Calculates the number of cutting passes (infeeds to the milling planes, infeeds in the milling depths)
3. Move to the safety clearance and plunge to the first milling depth.
4. Mill the first plane.
5. Retract by the safety clearance, return and cut to the next milling depth.
6. Repeat steps 4 and 5 until the complete area is milled.
7. Returns to **Return plane RB**

Centric polygon - roughing G843





G843 roughs centric polygons defined with **G477**-Geo (XY plane) or **G487**-Geo (YZ plane). The cycle mills from the outside toward the inside. The tool moves to the working plane outside of the workpiece material.

Parameters:

- **ID**: **Milling contour** – name of the milling contour
- **NS**: **Block number of contour** – reference to the contour description
- **P**: **Milling depth** – maximum infeed in the working plane
- **I**: **O-size X**
- **K**: **O-size Z**
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
- **V**: **Overrun factor** – defines the distance by which the tool should pass the outside radius of the workpiece (default: 0.5)
  Overrun = **V** * milling diameter
- **F**: **Approach feed** for plunging (default: active feed rate)
- **RB**: **Return plane** (default: back to start position)
  - XY plane: Retraction position in Z direction
  - YZ plane: Return position in X direction (diameter)

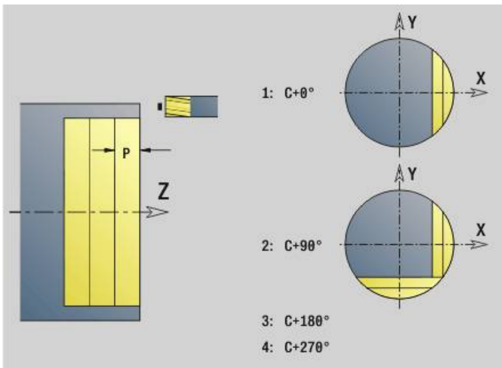
> Oversizes are taken into account:
> - **G57**: Oversize in X, Z direction
> - **G58**: Equidistant oversize in the milling plane

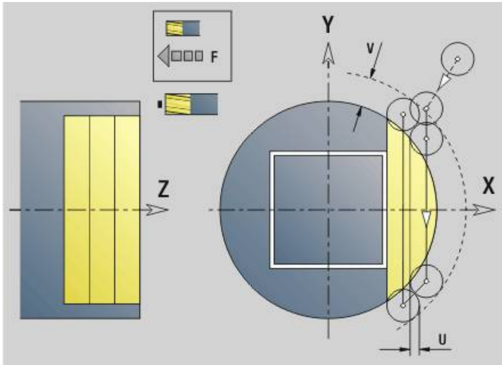
Cycle run

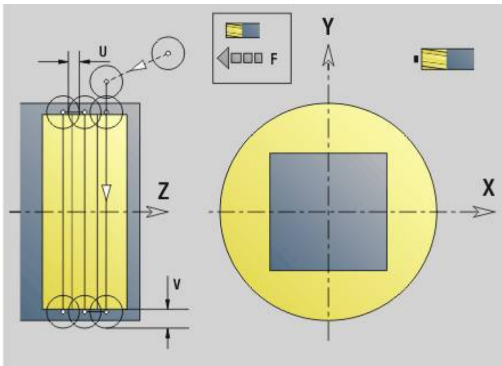
1. Start position (**X**, **Y**, **Z**, **C**) is the position before the cycle begins
2. Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling depths) and the spindle positions
3. Spindle turns to the first position. The tool moves to the safety clearance and plunges to the first milling depth.
4. Mill the first plane.
5. Retract by the safety clearance, return and cut to the next milling depth.
6. Repeat steps 4 and 5 until the complete area is milled.
7. The tool returns as defined in **Return plane J**; the spindle is rotated to the next position, the cutter moves to safety clearance and plunges to the first milling plane
8. Repeat steps 4 to 7 until all polygonal surfaces are milled.
9. Returns to **Return plane RB**

Centric polygon - finishing G844

G844 finishes centric polygons defined with **G477**-Geo (XY plane) or **G487**-Geo (YZ plane). The cycle mills from the outside toward the inside. The tool moves to the working plane outside of the workpiece material.







Parameters:

- **ID**: **Milling contour** – name of the milling contour
- **NS**: **Block number of contour** – reference to the contour description
- **P**: **Milling depth** – maximum infeed in the working plane
- **H**: **Mill cutting direction** for side finishing (default: 0)
  - **0: Up-cut**
  - **1: Climb**
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
- **V**: **Overrun factor** – defines the distance by which the tool should pass the outside radius of the workpiece (default: 0.5)
  Overrun = **V** * milling diameter
- **F**: **Approach feed** for plunging (default: active feed rate)
- **RB**: **Return plane** (default: back to start position)
  - XY plane: Retraction position in Z direction
  - YZ plane: Return position in X direction (diameter)

Cycle run

1. Start position (**X**, **Y**, **Z**, **C**) is the position before the cycle begins
2. Calculate the proportioning of cuts (infeeds to the milling planes, infeeds in the milling depths) and the spindle positions
3. Spindle turns to the first position. The tool moves to the safety clearance and plunges to the first milling depth.
4. Mill the first plane.
5. Retract by the safety clearance, return and cut to the next milling depth.
6. Repeat steps 4 and 5 until the complete area is milled.
7. The tool returns as defined in **Return plane J**; the spindle is rotated to the next position, the cutter moves to safety clearance and plunges to the first milling plane
8. Repeat steps 4 to 7 until all polygonal surfaces are milled.
9. Returns to **Return plane RB**

Pocket milling - roughing G845 (Y axis)

G845 roughs closed contours that are defined in the XY or YZ plane in the program sections:

- **FRONT Y**
- **REAR SIDE Y**
- **SURFACE Y**

Choose one of the following **Plunging behavior** settings, depending on the milling cutter you are using:

- Plunge vertically
- Plunge at a pre-drilled position
- Plunge in a reciprocating or helical motion

When **plunging at a pre-drilled position**, you have the following alternatives:

- Calculate positions, drill, mill. The machining process is performed in the following steps:
  - Insert drill.
  - Calculate hole positions with **G845 A1 ..:** or set the hole position at the center of the figure with **A2**
  - Predrill with **G71 NF ..:**
  - Call cycle **G845 A0 ..:** The cycle positions the tool above the hole; the tool plunges and mills the pocket

  > The parameters **O**=1 and **NF** must be defined.

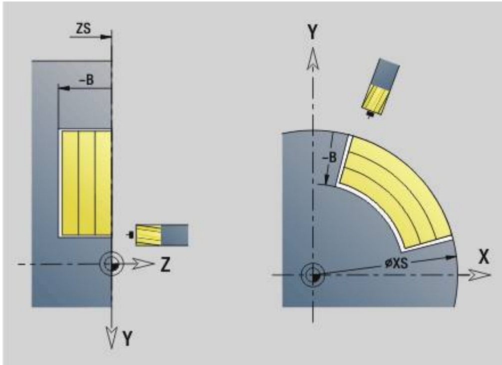
- Drill, mill. The machining process is performed in the following steps:
  - Drill a hole inside the pocket with **G71 ...**
  - Position the milling cutter above the hole and call **G845 A0 ...** The tool plunges and mills the section

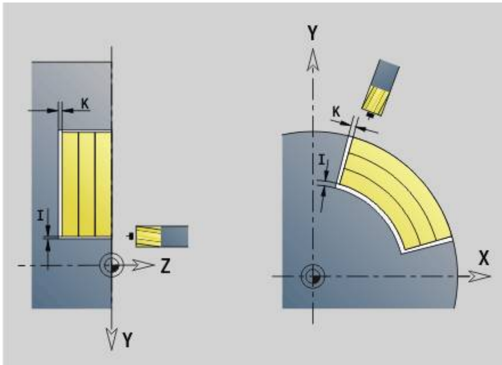
If the pocket consists of multiple sections, **G845** takes all the sections of the pocket into account for drilling and milling. Call **G845 A0 ...** separately for each section when calculating the hole positions without **G845 A1 ...**.

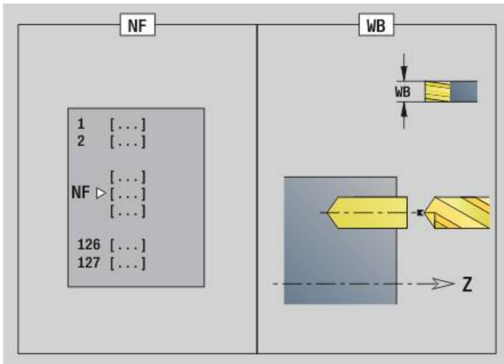
> **G845** takes the following oversizes into account:
> - **G57**: Oversize in X, Z direction
> - **G58**: Equidistant oversize in the milling plane
>
> Program oversizes for calculating the hole positions **and** for milling.

G845 (Y axis) – calculating hole positions

G845 A1 ... calculates the hole positions and stores them at the reference specified in **NF**. The cycle takes the diameter of the active tool into account when calculating the hole positions. Therefore, you need to insert the drill before calling **G845 A1 ...**. Program only the parameters given in the following table.







More information:

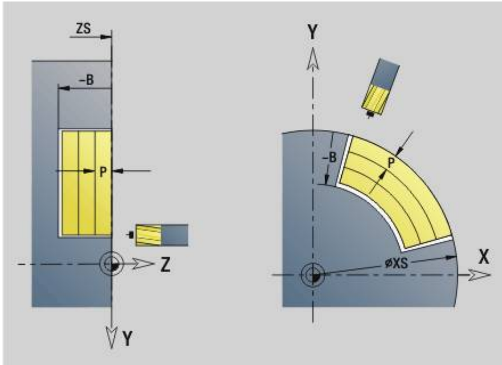
- **G845**—Fundamentals: **Further information:** "Pocket milling - roughing G845 (Y axis)", Page 659
- **G845**—Milling: **Further information:** "G845 (Y axis)—Milling", Page 661

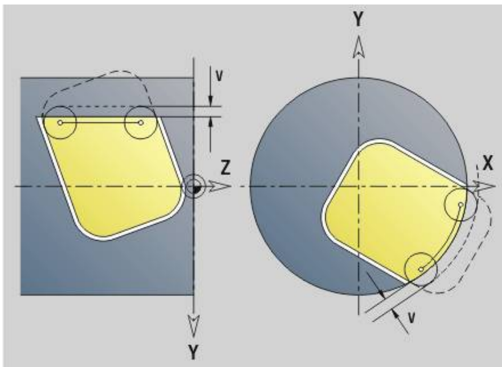
Parameters:

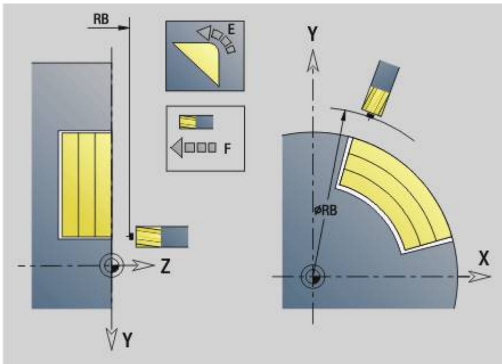
- **ID**: **Milling contour** – name of the milling contour
- **NS**: **Starting block no. of contour** – beginning of contour section
  - Figures: Block number of the figure
  - Free closed contour: First contour element (not starting point)
- **B**: **Milling depth** (default: depth from the contour description)
- **XS**: **Millg. top edge** lateral surface (replaces the reference plane from the contour definition)
- **ZS**: **Millg. top edge** face (replaces the reference plane from the contour definition)
- **I**: **O-size X**
- **K**: **O-size Z**
- **Q**: **Mach. direction** (default: 0)
  - **0: From the inside out**
  - **1: From the outside in**
- **A**: **(Mill=0/PredrillPos=1)**
- **NF**: **Position mark** – reference at which the cycle stores the predrill hole positions (range: 1 to 127)
- **WB**: **Diameter rework**

> - **G845** overwrites any hole positions that may still be stored at the reference **NF**
> - The parameter **WB** is used both for calculating the hole positions and for milling. When calculating the hole positions, **WB** describes the diameter of the milling cutter.

G845 (Y axis)–Milling







You can change the cutting direction with the parameters for **Direction H**, the **machining direction Q**, and the direction of tool rotation.

Program only the parameters given in the following table.

More information:

- G845–Fundamentals: **Further information:** "Pocket milling - roughing G845 (Y axis)", Page 659
- G845–Calculating hole positions: **Further information:** "G845 (Y axis) – calculating hole positions", Page 660

Parameters:

- **ID**: **Milling contour** – name of the milling contour
- **NS**: **Starting block no. of contour** – beginning of contour section
  - Figures: Block number of the figure
  - Free closed contour: First contour element (not starting point)
- **B**: **Milling depth** (default: depth from the contour description)
- **P**: **Max. approach** (default: milling in one infeed)
- **XS**: **Millg. top edge** lateral surface (replaces the reference plane from the contour definition)
- **ZS**: **Millg. top edge** face (replaces the reference plane from the contour definition)
- **I**: **O-size X**
- **K**: **O-size Z**
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
- **V**: **Overrun factor** – defines the distance by which the tool should pass the outside radius of the workpiece (default: 0.5)
  - 0: The defined contour is milled completely
  - 0 < **V** <= 1: Overrun = **V** * milling diameter
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **F**: **Approach feed** for plunging (default: active feed rate)
- **E**: **Reduced feed** for circular elements (default: active feed rate)
- **FP**: **Infeed rate in the plane** for the infeed to the next milling path
- **RB**: **Return plane** (default: back to start position)
  - XY plane: Retraction position in Z direction
  - YZ plane: Return position in X direction (diameter)
- **Q**: **Mach. direction** (default: 0)
  - **0: From the inside out**
  - **1: From the outside in**
- **A**: **(Mill=0/PredrillPos=1)** (default: 0)
- **NF**: **Position mark** – reference at which the cycle stores the predrill hole positions (range: 1 to 127)
- **O**: **Plunging behavior** (default: 0)

- **O**=0 (vertical plunging): The cycle moves the tool to the starting point; the tool plunges at the feed rate for infeed and mills the pocket
- **O**=1 (plunge at pre-drilled position):
  - If **NF** is programmed: The cycle positions the milling cutter above the first pre-drilled hole; the tool plunges and mills the first area. If applicable, the cycle positions the tool to the next pre-drilled hole and mills the next area, etc.
  - If **NF** is not programmed: The cycle plunges at the current position and mills the area. If applicable, position the tool to the next pre-drilled hole and mill the next area, etc.
- **O**=2 or 3 (plunge in a helical motion): The tool plunges at the angle **W** and mills full circles with the diameter **WB**. As soon as it reaches the milling depth **P**, the cycle switches to face milling
  - **O**=2—manually: The cycle plunges at the current position and machines the area that can be reached from this position
  - **O**=3—automatically: The cycle calculates the plunging position, plunges and machines this area. The plunging motion ends at the starting point of the first milling path, if possible. If the pocket consists of multiple areas, the cycle successively machines all the areas
- **O**=4 or 5 (reciprocating linear plunge): The tool plunges at the angle **W** and mills a linear path of length **WB**. You can define the orientation angle in **WE**. The cycle then mills along this path in the opposite direction. As soon as it reaches the milling depth **P**, the cycle switches to face milling
  - **O**=4—manually: The cycle plunges at the current position and machines the area that can be reached from this position
  - **O**=5—automatically: The cycle calculates the plunging position, plunges and machines this area. The plunging motion ends at the starting point of the first milling path, if possible. If the pocket consists of multiple areas, the cycle successively machines all the areas. The plunging position is determined from the type of figure and from **Q** as follows:
    - **Q0** (from the inside toward the outside):
      - Linear slot, rectangle, polygon: Reference point of the figure
      - Circle: Circle center
      - Circular slot, free contour: Starting point of the innermost milling path
    - **Q1** (from the outside toward the inside):
      - Linear slot: Starting point of the slot
      - Circular slot, circle: Not machined
      - Rectangle, polygon: Starting point of the first linear element
      - Free contour: Starting point of the first linear element (at least one linear element must exist)

- **O**=6 or 7 (reciprocating circular plunge): The tool plunges at the plunging angle **W** and mills a circular arc of 90°. The cycle then mills along this path in the opposite direction. As soon as it reaches the milling depth **P**, the cycle switches to face milling. **WE** defines the arc center, **WB** the arc radius
  - **O**=6—manually: The tool position corresponds to the center of the circular arc. The tool moves to the arc starting point and plunges
  - **O**=7—automatically (only permitted for circular slots and circles): The cycle calculates the plunging position on the basis of **Q**:
    - **Q0** (from the inside toward the outside):
      - Circular slot: The circular arc lies on the curvature radius of the slot
      - Circle: Not permitted
    - **Q1** (from the outside toward the inside): Circular slot, circle: The circular arc lies on the outermost milling path
- **W**: **Plunging angle** in infeed direction
- **WE**: **Position angle** of the milling path/circular arc

  Reference axis:
  - Front or rear face: Positive XK axis
  - Lateral surface: Positive Z axis

  Default orientation angle, depending on **O**:
  - **O**=4: **WE**=0°
  - **O**=5 and
    - Linear slot, rectangle, polygon: **WE** = orientation angle of the figure
    - Circular slot, circle: **WE**=0°
    - Free contour and **Q0** (from the inside toward the outside): **WE**=0°
    - Free contour and **Q1** (from the outside toward the inside): Orientation angle of the starting element
- **WB**: **Diameter rework** (default: 1.5 * milling diameter)

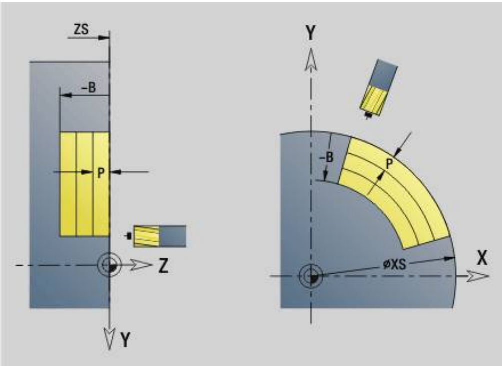
Cutting direction, machining direction and direction of tool rotation.

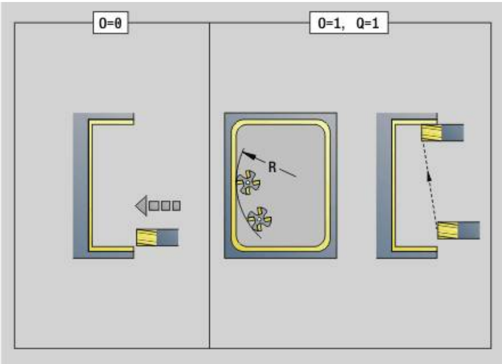
> For the machining direction **Q=1** (from the outside toward the inside), please note:
> - The contour must start with a linear element
> - If the starting element is < **WB**, **WB** is reduced to the length of the starting element
> - The length of the starting element must not be less than 1.5 times the diameter of the milling cutter

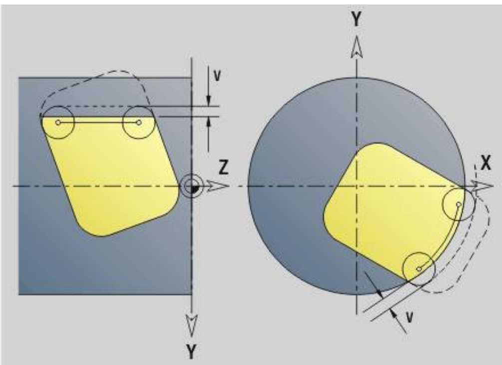
Cycle run:

1 Start position (**X**, **Y**, **Z**, **C**) is the position before the cycle begins
2 Calculates the number of cuts (infeeds to the milling planes, infeeds in the milling depths) and the plunging positions and paths for reciprocating or helical plunges.
3 Approaches to safety clearance and, depending on **O**, feeds to the first milling depth or approaches helically or on a reciprocating path
4 Mill the first plane.
5 Retract by the safety clearance, return and cut to the next milling depth.
6 Repeat steps 4 and 5 until the complete area is milled.
7 Returns to **Return plane RB**

Pocket milling - finishing G846 (Y axis)







G846 finishes closed contours defined in the XY or YZ plane in the program sections:

- **FRONT Y**
- **REAR SIDE Y**
- **SURFACE Y**

You can change the cutting direction with the parameters for **Mill cutting direction H**, the **Machining direction Q**, and the direction of tool rotation.

Parameters:

- **ID**: **Milling contour** – name of the milling contour
- **NS**: **Starting block no. of contour** – beginning of contour section
  - Figures: Block number of the figure
  - Free closed contour: First contour element (not starting point)
- **B**: **Milling depth** (default: depth from the contour description)
- **P**: **Max. approach** (default: milling in one infeed)
- **XS**: **Millg. top edge** lateral surface (replaces the reference plane from the contour definition)
- **ZS**: **Millg. top edge** face (replaces the reference plane from the contour definition)
- **R**: **Apprch angle** (default: 0)
  - **R**=0: Contour element is approached directly. Feed to the starting point above the milling plane, then vertical plunge
  - **R**>0: Tool moves on approaching/departing arc that connects tangentially to the contour element
- **U**: **Overlap factor** – defines the overlap of milling paths (default: 0.5) (range: 0 to 0.99)
  Overlap = **U** * milling diameter
- **V**: **Overrun factor** – defines the distance by which the tool should pass the outside radius of the workpiece (default: 0.5)
  Overrun = **V** * milling diameter
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **F**: **Approach feed** for plunging (default: active feed rate)

- **E**: **Reduced feed** for circular elements (default: active feed rate)
- **FP**: **Infeed rate in the plane** for the infeed to the next milling path
- **RB**: **Return plane** (default: back to start position)
  - XY plane: Retraction position in Z direction
  - YZ plane: Return position in X direction (diameter)
- **Q**: **Mach. direction** (default: 0)
  - **0: From the inside out**
  - **1: From the outside in**
- **O**: **Plunging behavior** (default: 0)
  - **O**=0 (plunge vertically): The cycle moves the tool to the starting point; the tool plunges and finishes the pocket
  - **O**=1 (approaching arc with depth feed): When machining the upper milling planes, the tool advances to the milling plane and then approaches on an arc. When machining the bottom milling plane, the tool plunges to the milling depth while moving on the approaching arc (three-dimensional approaching arc). You can use this approach behavior only in conjunction with an approaching arc **R**. The precondition is machining from the outside toward the inside (**O**=1)

Cutting direction, machining direction and direction of tool rotation.

Cycle run

1. Start position (**X**, **Y**, **Z**, **C**) is the position before the cycle begins
2. Calculates the number of cutting passes (infeeds to the milling planes, infeeds in the milling depths)
3. Move to the safety clearance and plunge to the first milling depth.
4. Mill the first plane.
5. Retract by the safety clearance, return and cut to the next milling depth.
6. Repeat steps 4 and 5 until the complete area is milled.
7. Returns to **Return plane RB**

Engraving in XY plane G803

G803 engraves character strings aligned linearly in the XY plane.

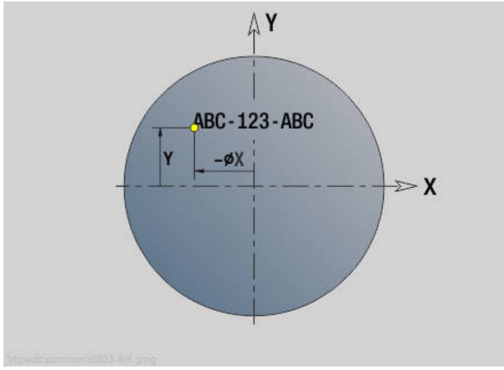
Further information: "Character sets", Page 484

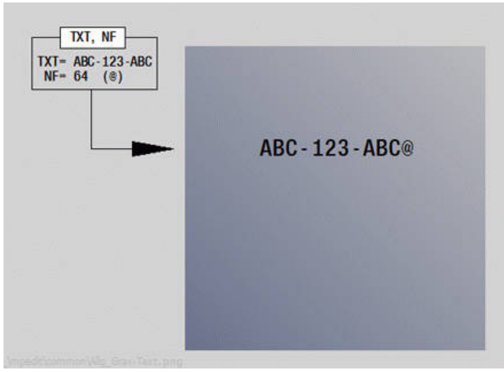
The cycles start engraving from the starting position or from the current position, if no starting position is defined.

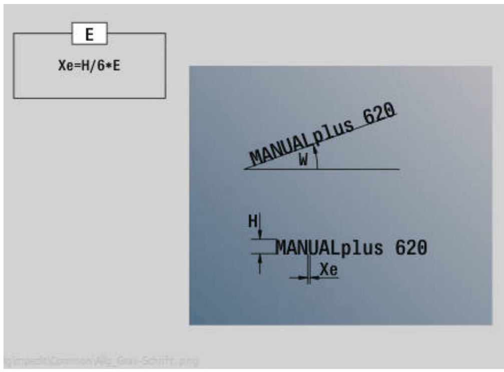
Example: If a character string is engraved with several calls, define the starting position in the first call. All other calls are programmed without a starting position.

Parameters:

- **X**, **Y**: **Start point**
- **Z**: **Final point** – Z position, infeed depth during milling
- **RB**: **Return plane** – Z position to which the tool retracts for positioning
- **ID**: **Text** to be engraved
- **NF**: **Char. no.** – ASCII code of the character to be engraved
- **NS**: **Block number of contour** – reference to the contour description
- **W**: **Inclinat. ang.** of the character string
  Example: 0° = vertical characters—the characters are arranged one after the other in positive X direction
- **H**: **Font height**
- **E**: **Distance factor**
  The spacing between the characters is calculated according to the following formula: **H** / 6 * **E**
- **F**: **Plunging feed rate factor** (plunging feed rate = current feed rate * **F**)
- **O**: **Mirror writing**
  - **0** (**No**): Engraving is not mirrored
  - **1** (**Yes**): Engraving is mirrored (mirror writing)







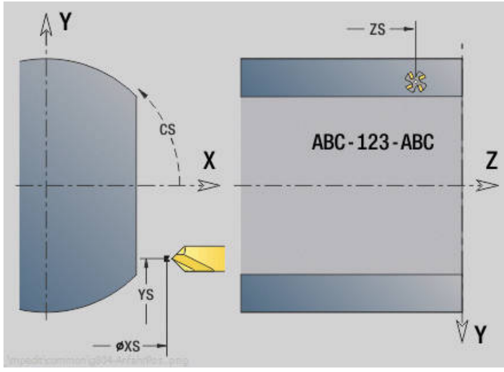
Engraving in YZ plane G804

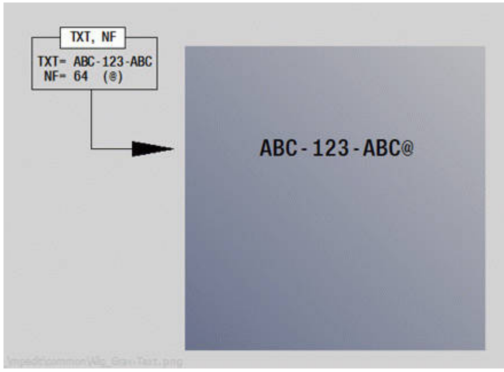
G804 engraves character strings aligned linearly in the YZ plane.

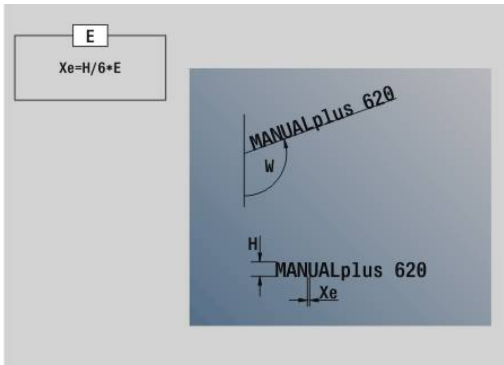
Further information: "Character sets", Page 484

The cycles start engraving from the starting position or from the current position, if no starting position is defined.

Example: If a character string is engraved with several calls, define the starting position in the first call. All other calls are programmed without a starting position.



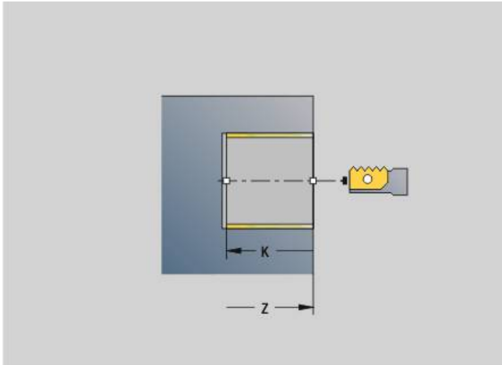


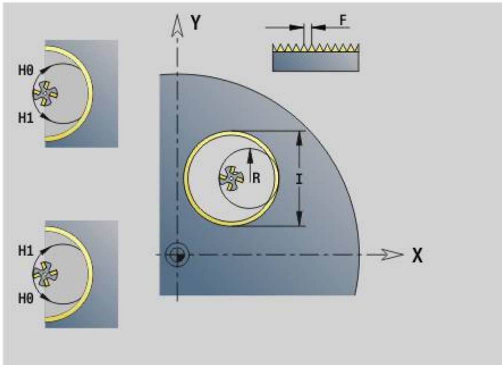


Parameters:

- **Y**, **Z**: **Start point**
- **X**: **Final point** – X position, infeed depth during milling (diameter value)
- **RB**: **Return plane** – X position to which the tool retracts for positioning
- **ID**: **Text** to be engraved
- **NF**: **Char. no.** – ASCII code of the character to be engraved
- **NS**: **Block number of contour** – reference to the contour description
- **W**: **Inclinat. ang.** of the character string
- **H**: **Font height**
- **E**: **Distance factor**
  The spacing between the characters is calculated according to the following formula: **H** / 6 * **E**
- **F**: **Plunging feed rate factor** (plunging feed rate = current feed rate * **F**)
- **O**: **Mirror writing**
  - **0** (**No**): Engraving is not mirrored
  - **1** (**Yes**): Engraving is mirrored (mirror writing)

Thread milling in XY plane G800





G800 mills a thread in existing holes.

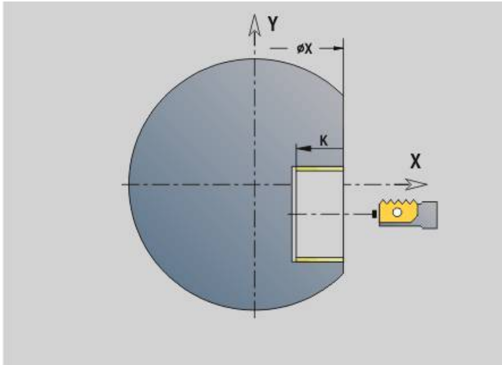
Place the tool at the center of the hole before calling **G799**. The cycle positions the tool to the **End point thread** inside the hole. Then, the tool approaches at **Apprch angle R** and mills the thread. With each rotation, the tool executes an infeed movement by the **Thread pitch F**. Then, the cycle retracts the tool and returns it to the **Start pt. Z**. The **V** parameter allows you to define whether the thread is milled in one rotation or, with single-point tools, in multiple rotations.

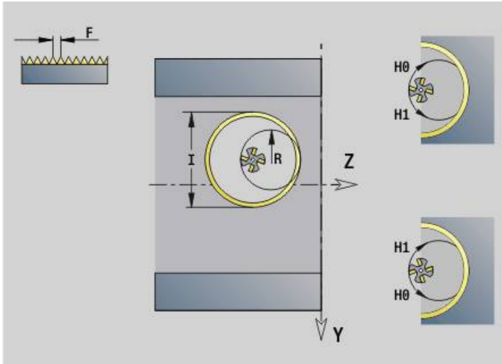
Parameters:

- **ID**: **Milling contour** – name of the milling contour
- **NS**: **Block number of contour** – reference to the contour description
- **I**: **Thread diameter**
- **Z**: **Start pt. Z**
- **K**: **Thread depth**
- **R**: **Approach radius**
- **F**: **Thread pitch**
- **J**: **Direction of thread:**
  - **0: Right-hand thread**
  - **1: Left-hand thread**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **V**: **Milling method**
  - **0: One revolution** – the thread is milled in a 360-degree helix
  - **1: Two or more revolutions** – the thread is milled in several helix paths (single-point tool)

> Use thread-milling tools for cycle **G800**.

Thread milling in YZ plane G806





G806 mills a thread in existing holes.

Place the tool at the center of the hole before calling **G799**. The cycle positions the tool to the **End point thread** inside the hole. Then, the tool approaches at **Apprch angle R** and mills the thread. With each rotation, the tool executes an infeed movement by the **Thread pitch F**. Then, the cycle retracts the tool and returns it to the **Start pt. Z**. The **V** parameter allows you to define whether the thread is milled in one rotation or, with single-point tools, in multiple rotations.

Parameters:

- **ID**: **Milling contour** – name of the milling contour
- **NS**: **Block number of contour** – reference to the contour description
- **I**: **Thread diameter**
- **X**: **Start pt. X**
- **K**: **Thread depth**
- **R**: **Approach radius**
- **F**: **Thread pitch**
- **J**: **Direction of thread:**
  - **0: Right-hand thread**
  - **1: Left-hand thread**
- **H**: **Mill cutting direction**
  - **0: Up-cut**
  - **1: Climb**
- **V**: **Milling method**
  - **0: One revolution** – the thread is milled in a 360-degree helix
  - **1: Two or more revolutions** – the thread is milled in several helix paths (single-point tool)

> Use thread-milling tools for cycle **G800**.

Skiving G807

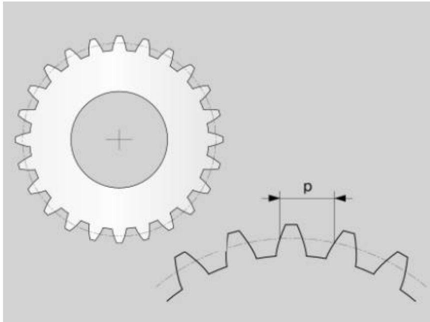
The **G807** function allows you to machine helical or straight-cut cylindrical gears.

After selecting the function, you can specify whether machining will take place before or behind the center of rotation or at the inside/outside of the workpiece. You can optionally define an inclination of the tool.

Material is removed by the axial feed movement of the tool in combination with the skiving movement.

Formulas

Use the following formulas to calculate missing data.



Formula symbols:

- n_T: Tool spindle speed
- n_W: Workpiece spindle speed
- z_T: Number of tool teeth
- z_W: Number of workpiece teeth
- m: Module
- p: Pitch
- h: Tooth height
- d: Pitch-circle diameter
- z: Number of teeth
- c: Trough-to-tip clearance
- d_a: Diameter of the addendum circle (outside diameter)
- d_f: Root circle diameter

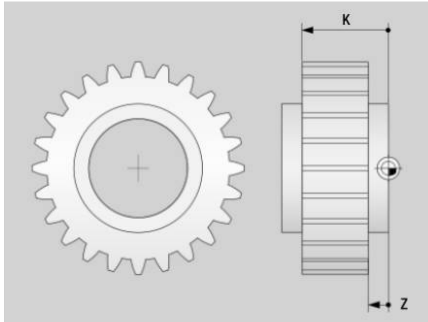
Formulas for speed calculation

Definition	Formula
Workpiece spindle	$n_W = n_T * \frac{z_T}{z_W}$
Tool spindle	$n_T = n_W * \frac{z_W}{z_T}$

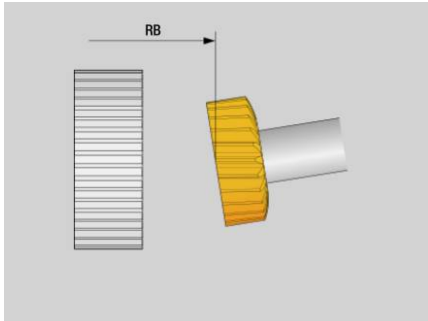
Formulas for straight-cut gears

Definition	Formula	
Module	$m = \frac{p}{\pi}$	$m = \frac{d}{z}$
Pitch	$p = \pi * m$	
Pitch-circle diameter	$d = m * z$	
Tooth height	$h = 2 * m + c$	
Diameter of the addendum circle (outside diameter)	$d_a = m * (z + 2)$	$d_a = d + 2 * m$
Root circle diameter	$d_f = d - 2 * (m + c)$	
Root circle diameter if tooth height > 0	$d_f = d_a - 2 * (h + c)$	
Number of teeth	$z = \frac{d}{m}$	$z = \frac{d_a - 2 * m}{m}$

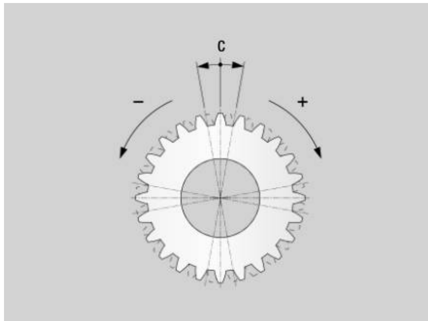
Parameters



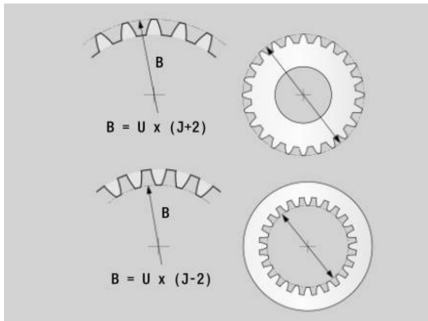
- **Z**: **Start point**
  Starting point of gearing system; enter an absolute value
- **K**: **Final point**
  End point of gearing system; enter an absolute value
- **WC**: **Tool lead angle**
  Thread angle of the milling cutter
- **RB**: **Return plane**
  Clearance height before and after machining; enter an absolute value



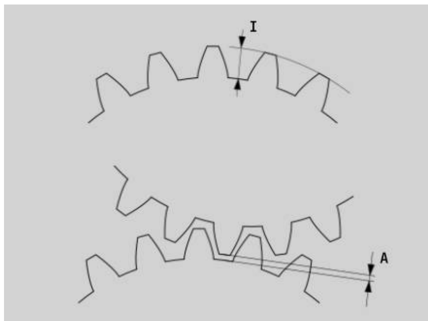
- **C**: **Offset C angle**
  Optional spindle position (e.g., for alignment with a hole)
- **AN**: **Pref. dir. incidence angle**
  Preferred direction of the B axis
  - **0: +B**
  - **1: -B**
- **J**: **Number of workpiece teeth**
  This parameter is mandatory for the calculation of further values.
- **U**: **Module**
  If you specify the **J Number of workpiece teeth** and **B Tip circle diameter** parameters, the control will calculate the **U Module** parameter automatically. If you additionally specify the **U Module** parameter, the control will ignore this input.
  To machine a helical gear, enter the transverse module in the **U Module** parameter.



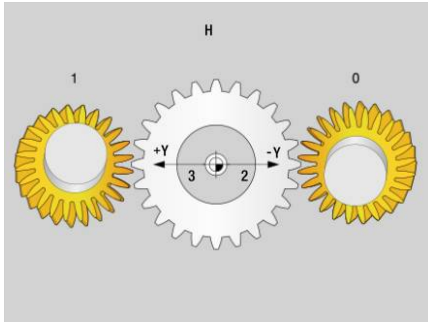
- **B**: **Tip circle diameter**
  If you specify the **J Number of workpiece teeth** and **U Module** parameters, the control will calculate the **B Tip circle diameter** parameter automatically. If you specify the **B Tip circle diameter** parameter, the control will ignore the entry in the **U Module** parameter.



- **I**: **Tooth height**
  If you do not specify the **I Tooth height** parameter, the control requires the **B Tip circle diameter** or **U Module** parameter for calculation. In addition to the tooth height, the control will also calculate the resulting root circle diameter.
- **A**: **Crest clearance**
  Distance between the addendum circle of the gear to be cut and the root circle of the mating gear.
  If you specify the **A Crest clearance** parameter, the control will consider this value for the root circle diameter calculation.



- **W**: **Angle of inclination**
  Angle of the helical gear
  For straight-cut gears, this angle is 0°.
- **E**: **Pre-positioning feed rate**
  Feed rate for all pre-positioning movements, including the inclination movement
- **S**: **Cutting speed** in m/min
- **H**: **Machining side**



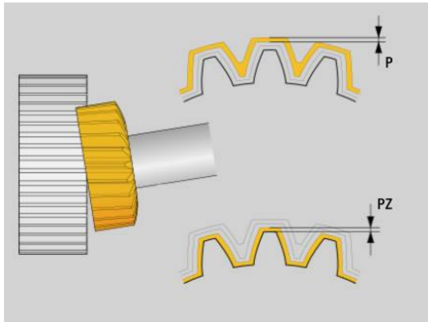
- **0: Behind workpiece, outside**
- **1: Before workpiece, outside**
- **2: Behind workpiece, inside**
- **3: Before workpiece, inside**

- **V**: **Direction of tool rotation**
  - **3: M3**
  - **4: M4**
- **O**: **Rotation direction reversal**

  Optional reversal of the workpiece spindle rotation direction
  - **0: No**
  - **1: Yes**
- **P**: **First infeed**

  Incremental value of the infeed depth for the first cut
- **PZ**: **Last infeed**

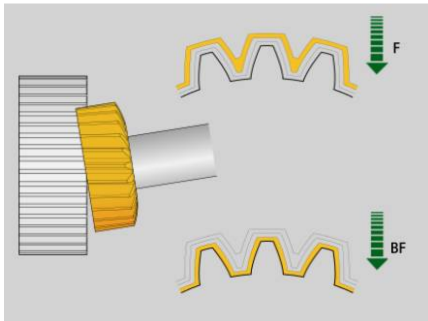
  Incremental value of the infeed depth for the last cut



- **F**: **First feed rate**

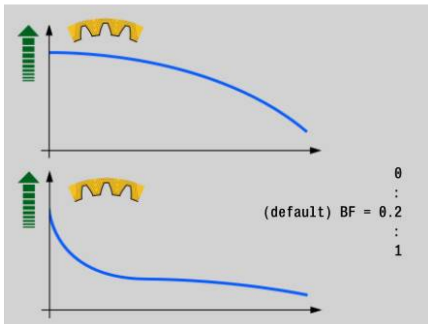
  Feed rate in millimeters per workpiece revolution for the first cut.
- **BF**: **Last feed rate**

  Feed rate in millimeters per workpiece revolution for the last cut.



- **FP**: **Feed-rate reduction factor**

  The reduction factor defines the reduction of the feed rate that should decrease with increasing cutting numbers. The higher the cutting number, the faster the feed rate will be adjusted until the value of the **BF Last feed rate** parameter is reached.

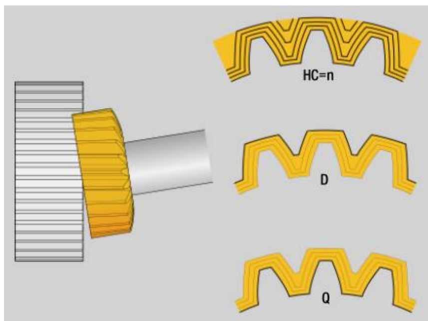


- **D**: **No. of first cut**

  Number of the cut with which the control starts machining
- **Q**: **No. of last cut**

  Number of the cut with which the control stops machining
- **HC**: **Number of cuts**

  If you omit the **HC Number of cuts** parameter, the control will calculate the required minimum number of cuts.



Notes

- The speed ratio between the workpiece and the tool is derived from the number of teeth of the gear and the number of cutting edges. Define the number of cutting edges in the **Tool editor**. Create the tool as a milling cutter.
- Before starting a machining operation, make sure that the direction of rotation has been set correctly for both spindles. If required, program a low spindle speed that allows you to see clearly whether the direction of rotation is correct. Please remember that the correct direction of rotation depends on the cutting direction of the tool and the machining side.
- If you set the **HC Number of cuts** parameter to **2**, the control will ignore the **PZ Last infeed** and **BF Last feed rate** parameters. If you set the **HC Number of cuts** parameter to **1**, the control will additionally ignore the **P First infeed** parameter.
- The control calculates the approach length and the idle travel distance automatically. Both are required for completely machining the gear between **Z Start point** and **K Final point**. When clamping the workpiece, make sure that it protrudes sufficiently from the fixture to avoid any collision of the tool with the fixture. Verify proper execution of machining in the simulation.
- Please note, however, that the simulation cannot display the gearing system. The simulation is suitable for checking the tool position and the infeed movements, for example.

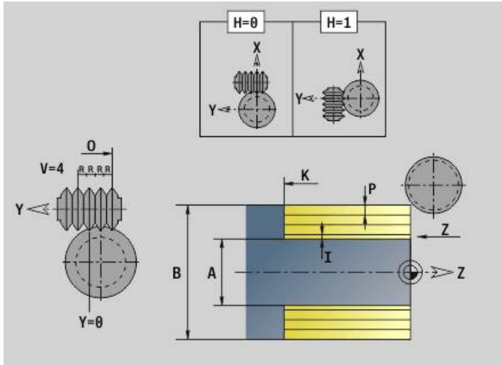
Hobbing G808

G808 mills a gear profile from the **Start point Z** to the **Final point K**. In **W** you enter the angular position of the tool.

If an oversize has been programmed, hobbing is split up in rough-machining and subsequent finishing.

In parameters **O**, **R** and **V** you define the tool shift. Shifting by **R** ensures a uniform wear of the hob cutter.

Use the parameter **U** to specify the gear ratio in the tool drive.



Parameters:

- **Z**: **Start point**
- **K**: **Final point**
- **C**: **Angle** – offset angle of the C axis
- **H**: **Infeed axis**
  - 0: Tool infeed is performed in the X axis
  - 1: Tool infeed is performed in the Y axis
- **Q**: **Spindle with workpiece**
  - 0: Spindle 1 (main spindle) holds the workpiece
  - 1: Spindle 2 holds the workpiece
  - 2: Spindle 3 holds the workpiece
  - 3: Spindle 4 holds the workpiece
- **AC**: **Tool spindle no. 0..3**
  - 0: Spindle 1 (main spindle) holds the tool
  - 1: Spindle 2 holds the tool
  - 2: Spindle 3 holds the tool
  - 3: Spindle 4 holds the tool
- **A**: **Root circle diameter**
- **B**: **Tip circle diameter**
- **J**: **Number of workpiece teeth**
- **W**: **Angle position**
- **WC**: **Angle of inclination** of the gear
- **S**: **Cutting speed** in m/min
- **F**: **Feed per revolution**
- **D**: **Turn. direct.** of the workpiece
  - 3: **M3**
  - 4: **M4**
- **P**: **Maximum infeed**
- **I**: **Allowance**
- **E**: **Finishing feed**
- **O**: **Shift starting position**
- **R**: **Shift value**
- **V**: **Shift quantity**
- **U**: **Transmission ratio**

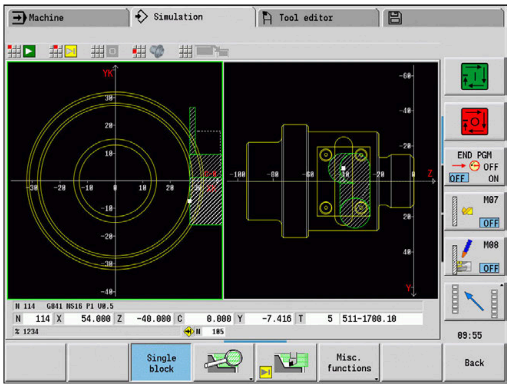
> Program **G728** to compensate for an offset occurring when you machine helical teeth.
>
> **Further information:** "Compens. for helical teeth G728", Page 510

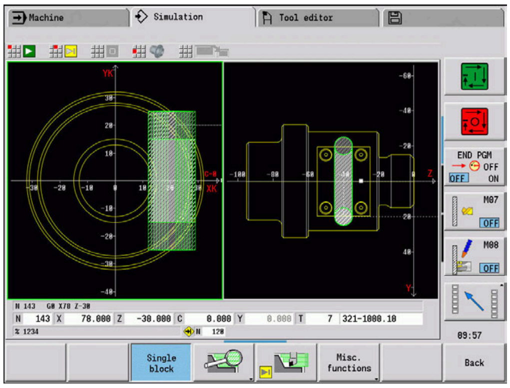
8.8 Example program

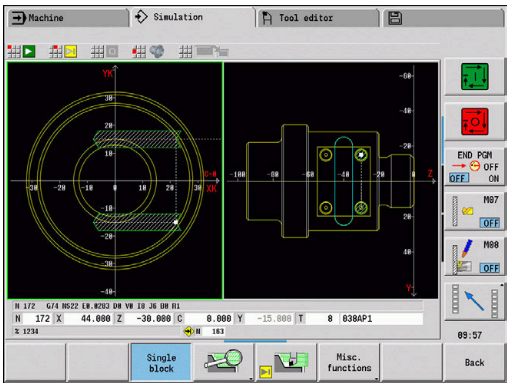
Machining with the Y axis

The milling and drilling contours are nested in the following NC program. A linear slot is machined on the single surface. On the same single surface, a hole pattern with two holes is machined both to the left and right of the slot.

At first, the turning operation is performed, and then the **single surface** is milled. Following that, the linear slot is machined using the **Pocket milling, lateral surface Y** unit. Then the slot is deburred. Further units are used to center the hole patterns, then drill them and finally tap the holes.







Example: Y axis [BSP_Y.NC]

PROGRAM HEAD		
#MATERIAL	ALUMINUM	
#PIECE	Y AXIS	
#UNIT	METRIC	
TURRET 1		
T1	ID"Roughing 80 G."	
T2	ID"NC center drill"	
T3	ID"Finishing 35 G."	
T4	ID"Drill 5.2mm"	
T5	ID"Thread outside"	

T6	ID"Tapping M6"	
T8	ID"Mill D16mm"	
T10	ID"Mill D6mm"	
T12	ID"Deburring_m"	
BLANK		
N 1 G20 X70 Z97 K1		
FINISHED PART		
N 2 G0 X0 Z0		
N 3 G1 X30 BR-2		
N 4 G1 Z-20		
N 5 G25 H7 I1.5 K7 R1 W30 FP2		Undercut DIN 76
N 6 G1 X56 BR-1		
N 7 G1 Z-60		
N 8 G1 X64 BR-1		
N 9 G1 Z-75 BR-1		
N 10 G1 X44 BR3		
N 11 G1 Z-95 BR-1		
N 12 G1 X0N 13 G1 Z0		
SURFACE Y X56 C0		Define YZ plane
N 14 G308 ID"Surface"		
N 15 G386 Z-55 Ki8 B30 X56 C0		Single surface
N 16 G308 ID"Slot 10mm" P-2		
N 17 G381 Z-40 Y0 A90 K50 B10		Linear slot on single surface
N 18 G309		
N 19 G308 ID"Hole_1 M6" P-15		
N 20 G481 Q2 Z-30 Y15 K-30 J-15		Linear pattern on single surface
N 21 G380 B5.2 P15 W118 I6 J10 F1 V0 o7		Drilling, tapping, centering
N 22 G309		
N 23 G308 ID"Hole_2 M6" P-15		
N 24 G481 Q2 Z-50 Y15 K-50 J-15		Linear pattern on single surface
N 25 G380 B5.2 P15 W118 I6 J10 F1 V0 O7		Drilling, tapping, centering
N 26 G309		
N 27 G309		
MACHINING		
N 28 UNIT ID"START"		[**Program beginning**]
N 30 G26 S3500		
N 31 G126 S2000		
N 32 G59 Z256		
N 33 G140 D1 X400 Y0 Z500		
N 34 G14 Q0 D1		

N 35 END_OF_UNIT	
N 36 UNIT ID"G820_ICP"	[G820 Roughing in ICP]
N 38 T1	
N 39 G96 S220 G95 F0.35 M3	
N 40 M8	
N 41 G0 X72 Z2	
N 42 G47 P2	
N 43 G820 NS3 NE3 P2 I0 K0 H0 Q0 V3 D0	
N 44 G47 M9	
N 45 END_OF_UNIT	
N 46 UNIT ID"G810_ICP"	[G810 Longitudinal roughing in ICP]
N 48 T1	
N 49 G96 S220 G95 F0.35 M3	
N 50 M8	
N 51 G0 X72 Z2	
N 52 G47 P2	
N 53 G810 NS4 NE9 P3 I0.5 K0.2 H0 Q0 V0 D0	
N 54 G14 Q0 D1	
N 55 G47 M9	
N 56 END_OF_UNIT	
N 57 UNIT ID"G890_ICP"	[G890 Contouring in ICP]
N 59 T3	
N 60 G96 S260 G95 F0.18 M4	
N 61 M8	
N 62 G0 X72 Z2	
N 63 G47 P2	
N 64 G890 NS4 NE9 V1 Q0 H3 O0 B0	
N 65 G14 Q0 D1	
N 66 G47 M9	
N 67 END_OF_UNIT	
N 68 UNIT ID"G32_MAN"	[G32 Cylindrical thread, direct]
N 70 T5	
N 71 G97 S800 M3	
N 72 M8	
N 73 G0 X30 Z5	
N 74 G47 P2	
N 75 G32 X30 Z-19 F1.5 BD0 IC8 H0 V0	
N 76 G14 Q0 D1	
N 77 G47 M9	
N 78 END_OF_UNIT	

N 79 UNIT ID"C_AXIS_ON"	[C axis ON]
N 81 M14	
N 82 G110 C0	
N 83 END_OF_UNIT	
N 84 UNIT ID"G841_Y_MANT"	[Single surface in Y axis, latrl.]
N 86 T8	
N 87 G197 S1200 G195 F0.25 M104	
N 88 M8	
N 89 G19	
N 90 G110 C0	
N 91 G0 Y0	
N 92 G0 X74 Z10	
N 93 G147 K2 I2	
N 94 G841 ID"Surface" P5	[Mill a single surface]
N 95 G47 M9	
N 96 G14 Q0 D1	
N 97 G18	
N 98 END_OF_UNIT	
N 99 UNIT ID"G845_TAS_Y_MANT"	[ICP pocket mill, lateral surf. Y]
N 101 T10	
N 102 G197 S1200 G195 F0.18 M104	
N 103 G19	
N 104 M8	
N 105 G110 C0	
N 106 G0 Y0	
N 107 G0 X74 Z-40	
N 108 G147 I2 K2	
N 109 G845 ID"Slot 10 mm" Q0 H0	Mill a slot on single surface
N 110 G47 M9	
N 111 G14 Q0 D1	
N 112 G18	
N 113 END_OF_UNIT	
N 114 UNIT ID"G840_ENT_Y_MANT"	[G840 Deburring]
N 116 T12	
N 117 G197 S800 G195 F0.12 M104	
N 118 G19	
N 119 M8	
N 120 G110 C0	
N 121 G0 Y0	
N 122 G0 X74 Z-40	
N 123 G147 I2 K2	

N 124 G840 ID"Slot 10mm" Q1 H0 P0.8 B0.15	Deburr a slot on a single surface
N 125 G47 M9	
N 126 G14 Q0 D1	
N 127 G18	
N 128 END_OF_UNIT	
N 129 UNIT ID"G72_ICP_Y"	[**G72 Boring, cntrsinkg ICP Y**]
N 131 T2	
N 132 G197 S1000 G195 F0.22 M104	
N 133 M8	
N 134 G147 K2	
N 135 G72 ID"Hole_1 M6" D0	Center the holes of the first pattern
N 136 G47 M9	
N 137 END_OF_UNIT	
N 138 UNIT ID"G72_ICP_Y"	[**G72 Boring, cntrsinkg ICP Y**]
N 140 T2	
N 141 G197 S1000 G195 F0.22 M104	
N 142 M8	
N 143 G147 K2	
N 144 G72 ID"Hole_2 M6" D0	Center the holes of the second pattern
N 145 G47 M9	
N 146 G14 Q0 D1	
N 147 END_OF_UNIT	
N 148 UNIT ID"G74_ICP_Y"	[**G74 Drilling in ICP Y**]
N 150 T4	
N 151 G197 S1200 G195 F0.24 M103	
N 152 M8	
N 153 G147 K2	
N 154 G74 ID"Hole_1 M6" D0 V2	Holes of the first pattern
N 155 G47 M9	
N 156 END_OF_UNIT	
N 157 UNIT ID"G74_ICP_Y"	[**G74 Drilling in ICP Y**]
N 159 T4	
N 160 G197 S1200 G195 F0.24 M103	
N 161 M8	
N 162 G147 K2	
N 163 G74 ID"Hole_2 M6" D0 V2	Holes of the second pattern
N 164 G47 M9	
N 165 G14 Q0 D1	
N 166 END_OF_UNIT	

N 167 UNIT ID"G73_ICP_Y"	[**G73 Tapping in ICP Y**]
N 169 T6	
N 170 G197 S800 M103	
N 171 M8	
N 172 G147 K2	
N 173 G73 ID"Hole_1 M6" F1	Tapping, first pattern
N 174 G47 M9	
N 175 END_OF_UNIT	
N 176 UNIT ID"G73_ICP_Y"	[**G73 Tapping in ICP Y**]
N 178 T6	
N 179 G197 S800 M103	
N 180 M8	
N 181 G147 K2	
N 182 G73 ID"Hole_2 M6" F1	Tapping, second pattern
N 183 G47 M9	
N 184 G14 Q0 D1	
N 185 END_OF_UNIT	
N 186 UNIT ID"C_AXIS_OFF"	[**C axis OFF**]
N 188 M15	
N 189 END_OF_UNIT	
N 190 UNIT ID"END"	[**Program end**]
N 192 M30	
N 193 END_OF_UNIT	
END	

9

TURN PLUS (Option 63)

9.1 The function TURN PLUS

To create programs with **TURN PLUS**, you program the blank and finished part with the aid of interactive graphics. The working plan is then generated automatically. As a result you get a commented and structured NC program.

With **TURN PLUS** you can create NC programs for the following applications:

- Turning
- Milling and drilling with the C axis
- Milling and drilling with the Y axis
- Full-surface machining

TURN PLUS concept

The workpiece description is the basis for working plan generation. The strategy for generating the working plan is specified in the **Machining sequence**.

TURN PLUS generates the working plan, which takes technology attributes such as oversizes, tolerances, etc. into account.

On the basis of the contour follow-up, **TURN PLUS** optimizes the paths for approach and avoids air cuts or collisions between workpiece and cutting edge.

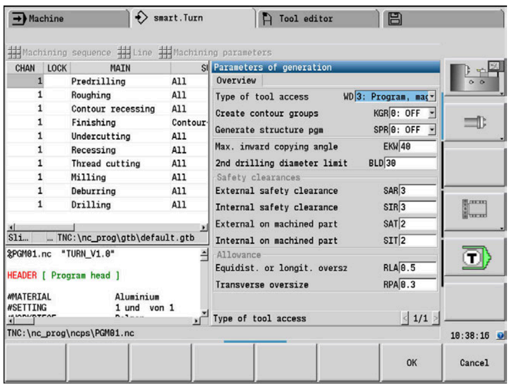
For tool selection, **TURN PLUS** uses the tools from the NC program or from the current turret assignment/magazine list, depending on the machine parameter settings. If there is no suitable tool in the turret assignment/magazine list, **TURN PLUS** looks for a suitable tool in the tool database and selects it. When programming the **Tool selection TS** parameter, you can select the tools manually.

The technology database provides the cutting data to **TURN PLUS**.

Machining parameters

The **Machining parameters** define details of machining. This allows you to adapt **TURN PLUS** to your individual needs.

When clamping the workpiece, **TURN PLUS** can determine the cutting limits and the datum shift for the NC program, depending on the machine parameter settings.



> **Before** generating the working plan, please note: You define the default values for machining parameters as well as general settings in the machine parameters.

Further information: User's Manual

With the **Machining parameters** menu item, it is possible to set the most important parameters during the programming process. The control will apply these settings to the machine parameters.

Here, you can program settings such as

- Type of tool access
- Contour groups
- Structure program
- Set-up clearance
- Oversize

9.2 Automatic working plan generation (AWG) submode

The **AWG** submode generates the work blocks of the working plan in the sequence defined in **Machining sequence**. You define the machining details in the **Machining parameters** input form. The **TURN PLUS** function automatically determines all required elements of a work block. Use the **machining sequence editor** to specify the machining sequence.

A work block has the following content:

- Tool call
- Cutting values (technology data)
- Approach (may be omitted)
- Machining cycle
- Tool retraction (may be omitted)
- Moving to tool change point (may be omitted)

> Refer to your machine manual.
>
> The machine tool builder can provide you with a machine-dependent start unit containing the definitions of various transfer parameters that automatically take into account a bar feeder, for example.

You can change or supplement the generated work blocks subsequently.

TURN PLUS simulates machining in the **AWG** control graphic. You can set the sequence and representation of the control graphic by soft key.

Further information: User's Manual

> **TURN PLUS** outputs warnings during the contour analysis if certain areas cannot be machined at all or not completely. Check the respective sections after program creation and adapt them to your needs.

> In the **convertICP** (no. 602023) machine parameter, you define whether the control loads the programmed or calculated values into the NC program.

Information for using AWG

Please note when using automatic working plan generation:

- The **AWG** submode divides each circle at the quadrant boundaries. As a result, the program generated by the **AWG** may contain more contour elements than the original program.
- **AWG** automatically closes open contours.
- **AWG** always creates the contours in counterclockwise (CCW) direction.
- **AWG** always shifts the starting point of the contour to the lower left corner.

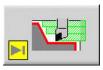
Generating a working plan

> **After** the working plan generation, consider the following: If no chucking equipment has been defined in the program as yet, **TURN PLUS** defines the chucking equipment for a specific type of clamping/clamping length and adjusts the cutting limitation accordingly. Adapt the values in the finished NC program.

Generating a working plan with **TURN PLUS**:

- Press the **TURN PLUS** soft key
- > **TURN PLUS** opens the most recently selected machining sequence..

- To activate **AWG** submode, press the **AWG** soft key
- > **TURN PLUS** displays the contours of the workpiece blank and the finished part in the graphics window.



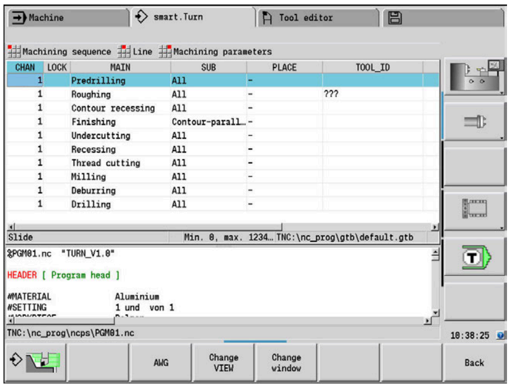
- Press the **Simulation** soft key
- > The **AWG** control graphic and the program generation are started.

- Press the **Back** soft key to return to the **TURN PLUS** menu

- Press the **Back** soft key to switch to **smart.Turn** mode of operation
- Apply the name of the current NC program without any changes
- Alternatively, enter the name under which the NC program should be saved

- Press the **Save** soft key to overwrite the current NC program

Machining sequence – fundamentals



In the **Machining sequence** you define the sequence in which the machining steps will be carried out.

TURN PLUS analyzes the contour in the sequence defined in **Machining sequence**. In this process the control determines the areas to be machined and the tool parameters. The **AWG** submode analyzes the contour using the **Machining parameters**.

TURN PLUS distinguishes between:

- **Main mach. operation** (e.g., undercutting)
- **Anc. mach. operation** (e.g., undercut type H, K, or U)
- **Machining location** (e.g., external or internal)
- **Tool selection** (automatic or manual)

Anc. mach. operation and **Machining location** allow you to refine the machining specification. If you do not define the **Anc. mach. operation** or the **Machining location**, the **AWG** generates machining blocks for all machining submodes and machining locations.

The following factors additionally influence the working plan generation:

- Geometry of the contour
- Attributes of the contour
- Tool availability
- Machining parameters

> If you only define the **Main mach. operation** in the **Machining sequence**, all **machining submodes** it comprises are executed in a specific sequence. However, you can also program the machining submodes and machining locations individually in any desired sequence in the **Machining sequence**. In this case, you should define the associated main machining operation again after having defined the machining submodes. This way, you can ensure that all machining submodes and locations will be taken into account.

The **AWG** submode does **not** generate the work blocks if any required preparatory step is missing, or if the appropriate tool is not available, etc. **TURN PLUS** skips machining operations and machining sequences that do not make sense in the machining process.

NOTICE

Danger of collision!

In **AWG** submode with drilling and milling (e.g. **Main mach. operation 11: Milling**) the control does not consider the current rotational position—instead, the **Finished part contour** is used as reference. Danger of collision during pre-positioning and machining!

- Program turning operations (e.g. **Main mach. operation 3: Roughing**) before the drilling and milling operations

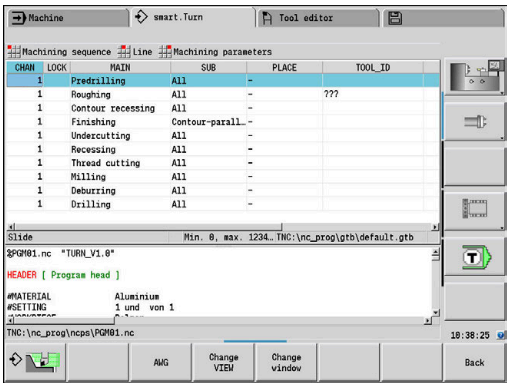
Organizing machining sequences

- **TURN PLUS** always uses the current machining sequence. The **current machining sequence** can be edited or overwritten by loading another **Machining sequence**.
- When you open **TURN PLUS**, the most recently used **Machining sequence** is automatically displayed.

Switching between views

The **Machining sequence** and the NC program can be displayed in a horizontal or vertical window layout. Press the **SWITCH VIEWS** soft key to switch between the two views.

Press the **CHANGE WINDOW** soft key to move the cursor between the Program and the Machining Sequence window.



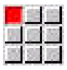
Editing and managing the Machining sequence

TURN PLUS uses the currently active machining sequence. You can change the **Machining sequence** and adapt it to your range of parts.

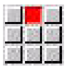
Opening the Machining sequence

To open any desired **Machining sequence**, proceed as follows:

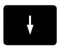
- ▶ Select **TURN PLUS**



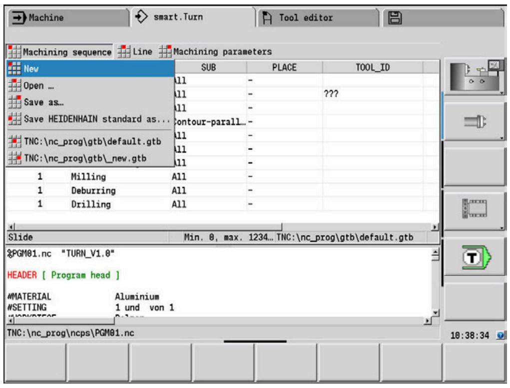
- ▶ Select the **Machining sequence**



- ▶ Select **Open ...**
- > **TURN PLUS** opens the selection list with the machining sequence files.



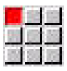
- ▶ Select the desired file



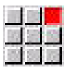
Saving the Machining sequence

To save any desired **Machining sequence**, proceed as follows:

- ▶ Select **TURN PLUS**



- ▶ Select the **Machining sequence**



- ▶ Select **Save as...**
- > **TURN PLUS** opens the selection list with the machining sequence files.
- ▶ Enter the file name
- ▶ Alternative: Overwrite an existing file

Creating a default machining sequence

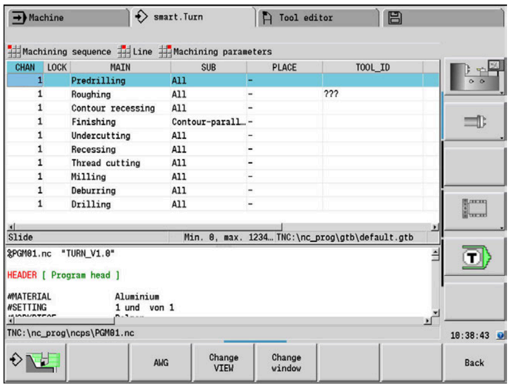
To create a default machining sequence, proceed as follows:

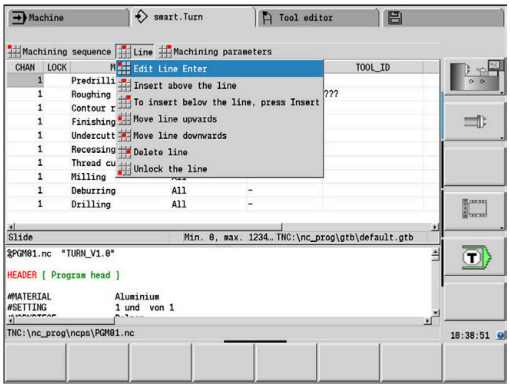
TURN PLUS

- Select **TURN PLUS**
- Select the **Machining sequence**
- Select **Save HEIDENHAIN standard as...**
- > **TURN PLUS** opens the selection list with the machining sequence files.
- Enter a file name under which you wish to store the HEIDENHAIN default machining sequence

Editing a Machining sequence

To edit a **Machining sequence**, proceed as follows:





- Position the cursor

TURN PLUS

- Select **TURN PLUS**
- Select **Line**
- Select a function
  - Inserting a new machining operation
  - Moving a machining operation
  - Editing a machining operation
  - Deleting a machining operation

Inserting a new machining operation:

- Select **Insert above the line** to insert a new machining operation before the cursor position
- Select **To insert below the line, press Insert** to insert a new machining operation after the cursor position

Moving a machining operation:

- Select **Move line upwards**
- Alternative: Select **Move line downwards**

Editing a machining operation:

- Select **Edit Line Enter**

OK

- Press the **OK** soft key

Deleting a machining operation:

- Select **Delete line**

Overview of machining sequences

The following table lists the possible combinations of **Main mach. operation** – **Anc. mach. operation** – **Machining location** and explains the working method of the **AWG**.

Machining sequence Predrilling

Main mach. operation	Anc. mach. operation	Machining location	Execution
Predrilling			**Contour analysis:** Determining the drilling steps **Machining parameters**: Centric predrilling (no. 602100)
	All	–	Predrilling

Machining sequence for Roughing

Main mach. operation	Anc. mach. operation	Machining location	Execution
Roughing			**Contour analysis**: Dividing the contour into areas for longitudinal/transverse external machining and longitudinal/transverse internal machining based on the transverse/longitudinal ratio **Sequence**: First external, then internal machining **Machining parameters**: Roughing (no 602200)
	All	–	Transverse machining, Longitudinal machining Extnl. and Internal
	Longitudinal machining	–	Longitudinal machining – Extnl. and Internal
	Longitudinal machining	**Extnl.**	Longitudinal machining – Extnl.
	Longitudinal machining	**Internal**	Longitudinal machining – Internal
	Transverse machining	–	Transverse machining – Extnl. and Internal
	Transverse machining	**Extnl.**	Transverse machining – Extnl.
	Transverse machining	**Internal**	Transverse machining – Internal
	Contour parallel	–	Contour-parallel machining – Extnl. and Internal
	Contour parallel	**Extnl.**	Contour-parallel machining – Extnl.
	Contour parallel	**Internal**	Contour-parallel machining – Internal

Machining sequence for Hollowing

Main mach. operation	Anc. mach. operation	Machining location	Execution
Hollowing			**Contour analysis**: Dividing the contour into areas for external and internal machining Recess areas (recesses) are determined and machined at a feasible plunge angle **Machining parameter**: Roughing or finishing
	Two tools	–	Extnl. and Internal with two tools or one tool with two B-axis angles
	Two tools	**Extnl.**	Extnl. With two tools or one tool with two B-axis angles
	Two tools	**Internal**	Internal with two tools or one tool with two B-axis angles
	Two tools	**External/front**	Axial machining – Extnl. With two tools or one tool with two B-axis angles
	Two tools	**External/rear**	Extnl./rear with two tools or one tool with two B-axis angles
	Two tools	**Internal/front**	Axial machining – Internal with two tools or one tool with two B-axis angles
	Neutral tool	-	Extnl. and Internal with a neutral tool or button tool
	Neutral tool	**Extnl.**	Extnl. with a neutral tool or button tool
	Neutral tool	**Internal**	Internal with a neutral tool or button tool
	Neutral tool	**External/front**	Axial machining – Extnl. with a neutral tool or button tool
	Neutral tool	**External/rear**	Extnl./rear with a neutral tool or button tool
	Neutral tool	**Internal/front**	Axial machining – Internal with a neutral tool or button tool

Machining sequence Finishing

Main mach. operation	Anc. mach. operation	Machining location	Execution
Finishing			**Contour analysis**: Dividing the contour into areas for external and internal machining **Sequence**: First external, then internal machining **Machining parameter**: finishing (no 602300)
	Contour parallel	–	External/internal machining
	Contour parallel	**Extnl.**	External machining
	Contour parallel	**Internal**	Internal machining
	Neutral tool	-	Extnl. and Internal with a neutral tool or button tool
	Neutral tool	**Extnl.**	Extnl. with a neutral tool or button tool
	Neutral tool	**Internal**	Internal with a neutral tool or button tool
	Neutral tool	**External/front**	Axial machining – Extnl. with a neutral tool or button tool
	Neutral tool	**External/rear**	Extnl./rear with a neutral tool or button tool
	Neutral tool	**Internal/front**	Axial machining – Internal with a neutral tool or button tool

Machining sequence Contour recessing

Main mach. operation	Anc. mach. operation	Machining location	Execution
Contour recessing			**Contour analysis**: Recess areas (recesses) are determined and machined based on the **inward copying angle EKW** **Sequence**: First outside, then inside machining **Machining parameter**: Global parameter for finished parts (no. 601900)
	All	–	Radial/axial machining – external and internal Shaft machining: external axial machining is executed on the front and rear
	Longitudinal machining	**Extnl.**	Radial machining – Extnl.
	Longitudinal machining	**Internal**	Radial machining – Internal
	Transverse machining	**Outside/front**	Axial machining – Extnl.
	Transverse machining	**Inside/front**	Axial machining – Internal

> **Recessing** and **Contour recessing** are used alternatively.

Machining sequence Recessing

Main mach. operation	Anc. mach. operation	Machining location	Execution
Recessing			**Contour analysis**: ■ **Without previous roughing** – the complete contour, including recess areas (undefined recesses), is machined ■ **With previous roughing** – Recess areas (undefined recesses) are determined and machined based on the **inward copying angle EKW** **Sequence**: First external, then internal machining **Machining parameter**: Global parameter for finished parts (no. 601900)
	All	–	Radial/axial machining – Extnl. and Internal
	Longitudinal machining	**Extnl.**	Radial machining – Extnl.
	Longitudinal machining	**Internal**	Radial machining – Internal
	Transverse machining	**Outside/front**	Axial machining – Extnl.
	Transverse machining	**Inside/front**	Axial machining – Internal

Recessing and **Contour recessing** are used alternatively.

Machining sequence Undercutting

Main mach. operation	Anc. mach. operation	Machining location	Execution
Undercutting			**Contour analysis** – determining the **Undercut** form elements: ▪ **Type H** – machining using single paths of traverse; copying tool (type 22x) ▪ **Type K** – machining using single paths of traverse; copying tool (type 22x) ▪ **Form U (G25 H4)** – machining using single paths of traverse; recessing tool (type 15x) **Sequence**: First external, then internal machining; first radial, then axial machining
	All	–	All recess types – Extnl. and Internal
	All	**Extnl.**	All recess types – Extnl.
	All	**Internal**	All recess types – Internal
	Type H, **Type K**, **Form U (G25 H4)**	–	Radial/axial machining – Extnl. and Internal
	Type H, **Type K**, **Form U (G25 H4)**	**Extnl.**	Machining – Extnl.
	Type H, **Type K**, **Form U (G25 H4)**	**Internal**	Machining – Internal

Machining sequence Recessing

Main mach. operation	Anc. mach. operation	Machining location	Execution
Recessing			**Contour analysis:** Determining the **Recess** form elements: ■ **Type S** (circlip – recess type S) ■ **Type D** (sealing ring – recess type D) ■ **Type A** (general recess) ■ **Type FK** (relief turn F) – FK is machined using only **Recessing** with **inward copying angle EKW** **Sequence**: First outside, then inside machining **Machining parameter** (for type FK): Global parameter for finished parts (no. 601900)
	All	–	All recess types; radial/axial machining; Extnl. and Internal
	Type S, **Type D**, **Type A**, **Type FK**	–	Radial/axial machining – Extnl. and Internal
	Type S, **Type D**, **Type A**, **Type FK**	**Extnl.**	Radial machining – Extnl.
	Type S, **Type D**, **Type A**, **Type FK**	**Internal**	Radial machining – Internal
	Type S, **Type D**, **Type A**, **Type FK**	**Outside/front**	Axial machining – Extnl.
	Type S, **Type D**, **Type A**, **Type FK**	**Inside/front**	Axial machining – Internal

Machining sequence Drilling

Main mach. operation	Anc. mach. operation	Machining location	Execution
Drilling			**Contour analysis**: Determining the **Hole** form elements **Sequence** – drilling operations/drilling combinations: ■ Centering / countersinking ■ Drilling ■ Countersinking / drilling and countersinking ■ Reaming / drilling with reaming ■ Tapping / drilling with thread **Sequence** – machining location: ■ Centric ■ Front face (also machines Y front if needed) ■ Lateral surface (also machines Y surface if needed) – then the elements are machined according to the **sequence** of geometrical definition
	All	–	All drilling/boring operations at all machining locations
	All	**Centric**	Centric machining of all drilling/boring operations
	All	**Front**	All drilling/boring operations on the front face
	All	**Lateral**	All drilling/boring operations on the lateral surface
	All	**Front only C**	Drilling/boring operations of the C axis on the front face
	All	**Cylinder only C**	Drilling/boring operations of the C axis on the lateral surface
	All	**Backward only C**	Drilling/boring operations of the C axis on the rear face
	All	**Front only Y**	Drilling/boring operations of the Y axis on the front face
	All	**Cylinder only Y**	Drilling/boring operations of the Y axis on the lateral surface
	All	**Backward only Y**	Drilling/boring operations of the Y axis on the rear face
	Centering, **Drilling**, **Countersinking**, **Reaming**, **Thread**	–	Machining at all machining locations
	Centering, **Drilling**, **Countersinking**, **Reaming**, **Thread**	**Centric**	Centric machining on the front face

Main mach. operation	Anc. mach. operation	Machining location	Execution
	Centering, **Drilling**, **Countersinking**, **Reaming**, **Thread**	**Front**	Machining on the face
	Centering, **Drilling**, **Countersinking**, **Reaming**, **Thread**	**Lateral**	Machining on the lateral surface
	Centering, **Drilling**, **Countersinking**, **Reaming**, **Thread**	**Front only C**	Machining operations of the C axis on the lateral surface
	Centering, **Drilling**, **Countersinking**, **Reaming**, **Thread**	**Cylinder only C**	Machining operations of the C axis on the lateral surface
	Centering, **Drilling**, **Countersinking**, **Reaming**, **Thread**	**Backward only C**	Machining operations of the C axis on the rear face
	Centering, **Drilling**, **Countersinking**, **Reaming**, **Thread**	**Front only Y**	Machining operations of the Y axis on the lateral surface
	Centering, **Drilling**, **Countersinking**, **Reaming**, **Thread**	**Cylinder only Y**	Machining operations of the Y axis on the lateral surface
	Centering, **Drilling**, **Countersinking**, **Reaming**, **Thread**	**Backward only Y**	Machining operations of the Y axis on the rear face

Machining sequence Thread cutting

Main mach. operation	Anc. mach. operation	Machining location	Execution
Thread cutting			**Contour analysis**: Determining the **Thread** form elements **Sequence**: First external, then internal machining; the elements are then machined according to the sequence of geometrical definition
	All	–	External and internal machining of cylindric (longitudinal), tapered, and transverse threads
	All	**Extnl.**	External machining of cylindric (longitudinal), tapered, and transverse threads
	All	**Internal**	Internal machining of cylindric (longitudinal), tapered, and transverse threads
	Cylinder	–	Machining of cylindric external and internal threads
	Cylinder	**Extnl.**	Machining of cylindric external threads
	Cylinder	**Internal**	Machining of cylindric internal threads
	Transverse	–	External and internal machining of transverse threads
	Transverse	**Extnl.**	External machining of transverse threads
	Transverse	**Internal**	Internal machining of transverse threads
	Taper	–	External and internal machining of taper threads
	Taper	**Extnl.**	External machining of taper threads
	Taper	**Internal**	Internal machining of taper threads

Machining sequence for milling

Main mach. operation	Anc. mach. operation	Machining location	Execution
milling			**Contour analysis**: Determining the milling contours **Sequence** – milling operation: ■ Linear and circular slots ■ Open contours ■ Closed contours (pockets), single surfaces and centric polygons **Sequence** – machining location: ■ Front face (also machines Y front if needed) ■ Lateral surface (also machines Y surface if needed) – then the elements are machined according to the **sequence** of geometrical definition
	All	–	All milling operations at all machining locations
	All	**Front**	All milling operations on the front face
	All	**Lateral**	All milling operations on the lateral surface
	All	**Front only C**	All milling operations of the C axis on the front face
	All	**Cylinder only C**	All milling operations of the C axis on the lateral surface
	All	**Backward only C**	All milling operations of the C axis on the rear face
	All	**Front only Y**	All milling operations of the Y axis on the front face
	All	**Cylinder only Y**	All milling operations of the Y axis on the lateral surface
	All	**Backward only Y**	All milling operations of the Y axis on the rear face
	Area, **Contour**, **Slot milling**, **Pocket**	–	Milling operations at all machining locations
	Area, **Contour**, **Slot milling**, **Pocket**	**Front**	Milling operations on the front face
	Area, **Contour**, **Slot milling**, **Pocket**	**Lateral**	Milling operations on the lateral surface
	Area, **Contour**, **Slot milling**, **Pocket**	**Front only C**	Milling operations of the C axis on the front face
	Area, **Contour**, **Slot milling**, **Pocket**	**Cylinder only C**	Milling operations of the C axis on the lateral surface
	Area, **Contour**, **Slot milling**, **Pocket**	**Backward only C**	Milling operations of the C axis on the rear face
	Area, **Contour**, **Slot milling**, **Pocket**	**Front only Y**	Milling operations of the Y axis on the front face

Main mach. operation	Anc. mach. operation	Machining location	Execution
	Area, **Contour**, **Slot milling**, **Pocket**	**Cylinder only Y**	Milling operations of the Y axis on the lateral surface
	Area, **Contour**, **Slot milling**, **Pocket**	**Backward only Y**	Milling operations of the Y axis on the rear face

Machining sequence Cut-off

Main mach. operation	Anc. mach. operation	Machining location	Execution
Cut-off	**All**	–	The workpiece is cut off
	Full-surface machining	–	The workpiece is cut off and rechucked

Machining sequence for Rechucking

Main mach. operation	Anc. mach. operation	Machining location	Execution
Rechucking	**Full-surface machining**	–	The workpiece is rechucked.

Machining sequence for Special machining tasks

Main mach. operation	Anc. mach. operation	Machining location	Execution
Special machining tasks	**All**	–	The defined subprogram is run.

Machining sequence for Deburring

Main mach. operation	Anc. mach. operation	Machining location	Execution
Deburring			**Contour analysis**: Determine milling contours with the **Deburring** attribute **Sequence** – **Machining location**: ■ Front face (also machines Y front if needed) ■ Lateral surface (also machines Y surface if needed) – then the elements are machined according to the **sequence** of geometrical definition
	All	–	All milling operations at all machining locations
	All	**Front**	Deburring of all milling operations on the front face
	All	**Lateral**	Deburring of all milling operations on the lateral surface
	All	**Front only C**	Deburring of all milling operations of the C axis on the front face

Main mach. operation	Anc. mach. operation	Machining location	Execution
	All	**Cylinder only C**	Deburring of all milling operations of the C axis on the lateral surface
	All	**Backward only C**	Deburring of all milling operations of the C axis on the rear face
	All	**Front only Y**	Deburring of all milling operations of the Y axis on the front face
	All	**Cylinder only Y**	Deburring of all milling operations of the Y axis on the lateral surface
	All	**Backward only Y**	Deburring of all milling operations of the Y axis on the rear face
	Contour, **Slot milling**, **Pocket** (*)	–	Deburring of the selected element at all machining locations
	Contour, **Slot milling**, **Pocket** (*)	**Front**	Deburring of the selected element on the front face
	Contour, **Slot milling**, **Pocket** (*)	**Lateral**	Deburring of the selected element on the lateral surface
	Contour, **Slot milling**, **Pocket** (*)	**Front only C**	Deburring of the selected element with the C axis on the front face
	Contour, **Slot milling**, **Pocket** (*)	**Cylinder only C**	Deburring of the selected element with the C axis on the lateral surface
	Contour, **Slot milling**, **Pocket** (*)	**Backward only C**	Deburring of the selected element with the C axis on the rear face
	Contour, **Slot milling**, **Pocket** (*)	**Front only Y**	Deburring of the selected element with the Y axis on the front face
	Contour, **Slot milling**, **Pocket** (*)	**Cylinder only Y**	Deburring of the selected element with the Y axis on the lateral surface
	Contour, **Slot milling**, **Pocket** (*)	**Backward only Y**	Deburring of the selected element with the Y axis on the rear face

***: Define the type of contour**

Machining sequence Engraving

Main mach. operation	Anc. mach. operation	Machining location	Execution
Engraving			**Sequence** – machining location: ■ Front face (also machines Y front if needed) ■ Lateral surface (also machines Y surface if needed) – then the elements are machined according to the **sequence** of geometrical definition
	All	–	Engraving of all elements at all machining locations
	All	**Front**	Engraving of all elements on the front face
	All	**Lateral**	Engraving of all elements on the lateral surface
	All	**Front only C**	Engraving of all elements with the C axis on the front face
	All	**Cylinder only C**	Engraving of all elements with the C axis on the lateral surface
	All	**Backward only C**	Engraving of all elements with the C axis on the rear face
	All	**Front only Y**	Engraving of all elements with the Y axis on the front face
	All	**Cylinder only Y**	Engraving of all elements with the Y axis on the lateral surface
	All	**Backward only Y**	Engraving of all elements with the Y axis on the rear face

Machining sequence Milling, finishing

Main mach. operation	Anc. mach. operation	Machining location	Execution
Milling, finishing			**Contour analysis**: Determining the milling contours **Sequence** – milling operation: ■ Linear and circular slots ■ Open contours ■ Closed contours (pockets), single surfaces and centric polygons **Sequence** – machining location: ■ Front face (also machines Y front if needed) ■ Lateral surface (also machines Y surface if needed) – then the elements are machined according to the **sequence** of geometrical definition
	All	–	Finishing of all elements at all machining locations
	All	**Front**	Finishing of all elements on the front face
	All	**Lateral**	Finishing of all elements on the lateral surface
	All	**Front only C**	Finishing of all elements with the C axis on the front face
	All	**Cylinder only C**	Finishing of all elements with the C axis on the lateral surface
	All	**Backward only C**	Finishing of all elements with the C axis on the rear face
	All	**Front only Y**	Finishing of all elements with the Y axis on the front face
	All	**Cylinder only Y**	Finishing of all elements with the Y axis on the lateral surface
	All	**Backward only Y**	Finishing of all elements with the Y axis on the rear face
	Contour, **Slot milling**, **Pocket** (*)	–	Finishing of the selected element at all machining locations
	Contour, **Slot milling**, **Pocket** (*)	**Front**	Finishing of the selected element on the front face
	Contour, **Slot milling**, **Pocket** (*)	**Lateral**	Finishing of the selected element on the lateral surface
	Contour, **Slot milling**, **Pocket** (*)	**Front only C**	Finishing of the selected element with the C axis on the front face
	Contour, **Slot milling**, **Pocket** (*)	**Cylinder only C**	Finishing of the selected element with the C axis on the lateral surface
	Contour, **Slot milling**, **Pocket** (*)	**Backward only C**	Finishing of the selected element with the C axis on the rear face
	Contour, **Slot milling**, **Pocket** (*)	**Front only Y**	Finishing of the selected element with the Y axis on the front face

Main mach. operation	Anc. mach. operation	Machining location	Execution
	Contour, **Slot milling**, **Pocket** (*)	**Cylinder only Y**	Finishing of the selected element with the Y axis on the lateral surface
	Contour, **Slot milling**, **Pocket** (*)	**Backward only Y**	Finishing of the selected element with the Y axis on the rear face

***: Define the milling operation**

9.3 AWG control graphic

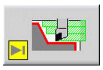
When you create a program in **AWG** submode, the programmed workpiece blank and finished part are displayed in the simulation window. In addition, all machining steps are simulated successively. The workpiece blank contour will be updated during machining.

Setting the AWG control graphic

To use the **AWG** control graphic, proceed as follows:

AWG

- ▶ Press the **AWG** soft key
- \> The control starts the **AWG** control graphic.



- ▶ Start the simulation of the machining steps
- \> The control displays any dialogs that provide information about machining and tools.

The control displays the soft-key icon with a contour outlined in red to identify the **AWG** control graphic.

To set the display of the tool paths and the simulation mode, you proceed in the same way as in **Simulation** submode.

Further information: User's Manual

Closing the AWG control graphic

The control generates the NC program during simulation. After you have simulated the machining process, you can close the graphics window.

Back

- ▶ Press the **Back** soft key
- \> The control returns to the **TURN PLUS** function.

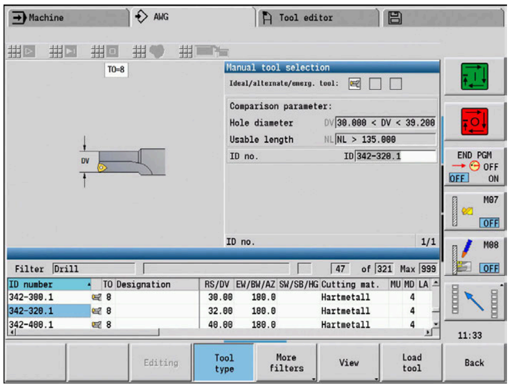
Back

- ▶ Press the **Back** soft key
- \> The control opens the **Save as** dialog box.

In the **File name** dialog box, the control displays the name of the open NC program. If you do not enter another file name, the open NC program will be overwritten. Alternatively, you can save the machining operation to another program.

9.4 Machining information

Tool selection and turret assignment list



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

The tool selection depends on:

- Machining direction
- Contour to be machined
- Machining sequence
- Setting of the Type of tool access machining parameter
- Machine parameter settings

> You can influence the parameter **Type of tool access WD** both with the machining parameters and with the machine parameter (no. 602001).

If the ideal tool is not available, **TURN PLUS**:

- First looks for a replacement tool,
- Then for an emergency tool.

If necessary, TURN PLUS adapts the machining cycle to the requirements of the replacement or emergency tool. If more than one tool is suitable for a machining operation, **TURN PLUS** uses the optimal tool. If **TURN PLUS** cannot find a suitable tool, select the tool manually.

The mount type distinguishes between different tool holders.

Further information: User's Manual

TURN PLUS checks whether the mount type in the tool holder description and the mount type in the turret pocket description are the same.

> Depending on machine parameter **defaultG59** (no. 602022), **TURN PLUS** automatically calculates the required datum shift for the workpiece and activates it with **G59**.
>
> **Further information:** User's Manual
>
> To calculate the datum shift, **TURN PLUS** takes the following values into account:
>
> - **Length Z** (workpiece blank description)
> - **Ov.size K** (description of workpiece blank)
> - **Edge of chuck Z** (description of chucking equipment and machining parameters)
> - **Chuck jaw reference B** (description of chucking equipment and machining parameters)

> In **AWG** submode, multipoint tools and tool holders for manual tool change are used as programmed under the **MANUAL TOOL** program section code.

Manual tool selection

TURN PLUS selects the tools depending on the machining parameters **Type of tool access WD** and **Tool selection TS**. If **TURN PLUS** cannot find a suitable tool in the specified lists, you need to select the tools manually.

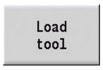
For machines with a Multifix tool holder, the control uses the tools selected under **MANUAL TOOL** as a tool pool.

TURN PLUS automatically predefines comparison parameters. With the soft keys, you can select the list in which you want to look for the tools.

Manual tool selection:

- Press the **Tool list** soft key

- Alternative: Press the **Turret list** soft key
- Choose a tool from the list



- Press the **Load tool** soft key to enter the tool in the tool selection

- Press the **Apply** soft key to conclude the tool selection

> If you press the **Tool graphics** soft key, the control will show the tool control graphics instead of the help graphic. The tool control graphics show you the actual data of the selected tool, for example the tool orientation.

Hollowing (internal profiling)

During **Hollowing** the control also machines recess areas whose thread angle is steeper than the **inward copying angle EKW**.

For this turning operation, the control uses either a neutral tool or a suitable pair of tools.

Hollowing with two tools

If this setting is used, **AWG** looks for two tools with different geometries in order to machine the contour from two directions.

If this setting is used and the machine additionally provides a B axis, then the control uses **one** tool with two different B-axis angles.

Hollowing with a neutral tool

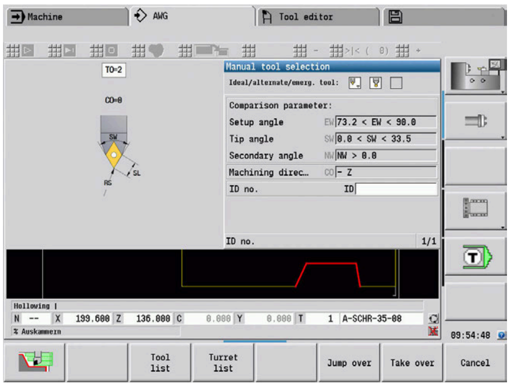
AWG looks for a neutral tool. If a neutral tool is not available, then the control uses a button tool.

Operating information

- Verify that **Hollowing** comes before recessing in the machining sequence in order to ensure that the corresponding areas have not been machined already.
- If the areas to be hollowed are too narrow, the control issues the message **Recess for automatic internal profiling too narrow** and opens the dialog for manual tool selection. The control displays the machining direction beneath the emergency tool.

  You have the following options:
  - If you skip the dialog, then the control cancels the **Hollowing** operation and later machines the contour with a contour-recessing operation.
  - If you confirm the dialogs, the contour will not be machined completely.

  If needed, you can change the tool angle to ensure that the required plunging angle is achieved and the contour is machined completely.



Contour recessing, Recessing

The **Cutting radius** must be smaller than the smallest inside radius of the recess contour, but >= 0.2 mm.

TURN PLUS determines the **Cutting width** from the recess contour:

- Recess contour includes paraxial floor elements with radii on both sides: **SB** <= **b** + 2 * **r** (if radii differ: smallest radius).
- Recess contour includes paraxial floor elements without radii or with a radius on one side: **SB** <= **b**
- The recess contour does not include paraxial floor elements: The **Cutting width** is determined from the recessing width divisor SBD (machining parameter no. 60240).

Abbreviations:

- **SB**: **Cutting width**
- **b**: Width of floor element
- **r**: Radius

Drillg

Depending on the geometry of the bore hole, the **AWG** submode determines the appropriate tool. For centric bore holes, **TURN PLUS** uses stationary tools.

Cutting data and coolant

To determine the cutting parameters, **TURN PLUS** uses the

- **Materials** (program head)
- **Cutting materials** (tool parameters)
- **Machining operation** (main operation in the machining sequence)

The values determined are multiplied by the tool-dependent compensation factors.

Further information: User's Manual

Note for roughing and finishing operations:

- Main feed rate for use of the primary cutting edge
- Auxiliary feed rate for use of the secondary cutting edge

Note for milling operations:

- Main feed rate for machining in the milling plane
- Auxiliary feed rate for infeed movements

For threading, drilling and milling operations, the cutting speed is converted into rotational speed.

Coolant: Depending on the workpiece material, cutting material and machining operation specified in the technology database, you can define whether coolant is used or not. The **AWG** submode activates the appropriate coolant circuits for the respective tool.

If you have specified that coolant is to be used, the **AWG** submode activates the coolant circuits for the respective machining block.

Speed limitation: **TURN PLUS** limits the rotational speed to the maximum speed specified in the TSF menu.

Inside contours

TURN PLUS machines continuous inside contours up to the transition from the deepest point to a greater diameter.

The end position for drilling, roughing and finishing operations depends on:

- Cutting limit, inside
- **Overhang length, inside ULI** (machining parameter no. 602227)

Prerequisite: The usable tool length must be sufficient for the machining operation. If it is not, then this parameter defines the inside machining operation. The following examples explain the principle.

Limits for internal machining operations:

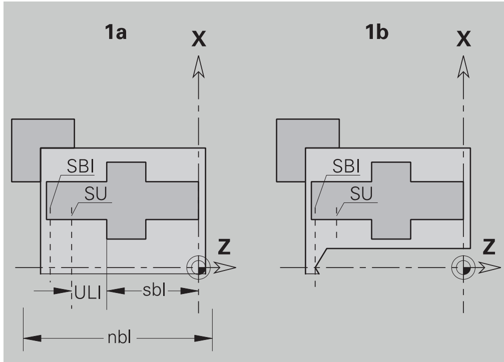
- **Predrilling**: **SBI** limits the drilling operation
- **Roughing**: **SBI** or **SU** limits the roughing operation
  - **SU** = basic length of roughing cut (**sbl**) + overhang length, inside (**ULI**)
  - To avoid **residual rings** during the machining process, **TURN PLUS** leaves an area of 5° in front of the roughing limit
- **Finishing**: **sbl** limits the finishing operation

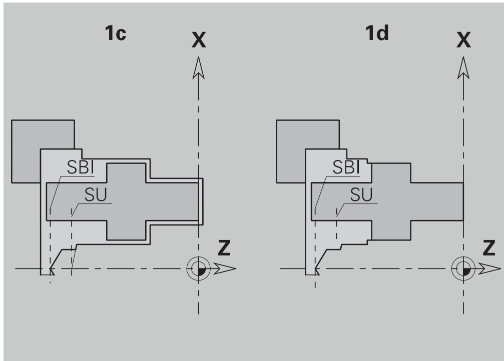
Roughing limit in front of cutting limit

Example 1: The roughing limit (**SU**) is located **in front of** the cutting limit, inside (**SBI**).

Abbreviations:

- **SBI**: Cutting limit, inside
- **SU**: Roughing limitation (**SU** = **sbl** + **ULI**)
- **sbl**: Basic length of roughing cut (deepest point of inside contour)
- **ULI**: Overhang length, inside (machining parameter no. 602227)
- **nbl**: Usable tool length (tool parameter)



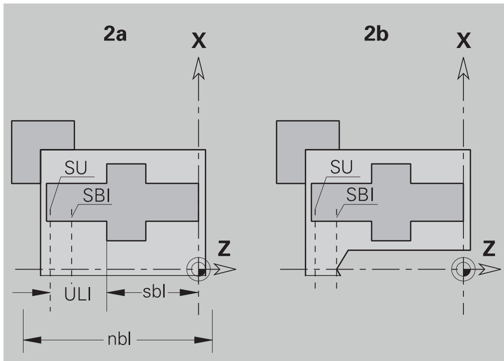


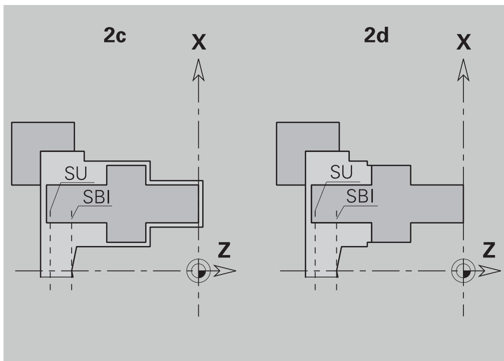
Roughing limit behind cutting limit

Example 2: The roughing limit (**SU**) is located **behind** the cutting limit, inside (**SBI**).

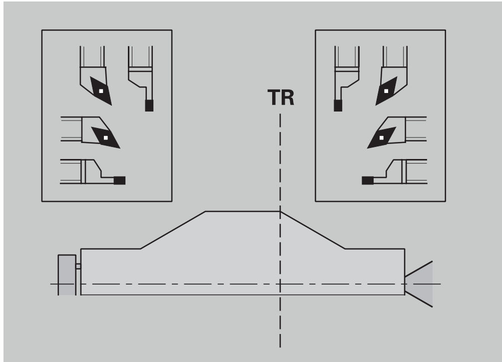
Abbreviations:

- **SBI**: Cutting limit, inside
- **SU**: Roughing limitation (**SU** = **sbl** + **ULI**)
- **sbl**: Basic length of roughing cut (deepest point of inside contour)
- **ULI**: Overhang length, inside (machining parameter no. 602227)
- **nbl**: Usable tool length (tool parameter)





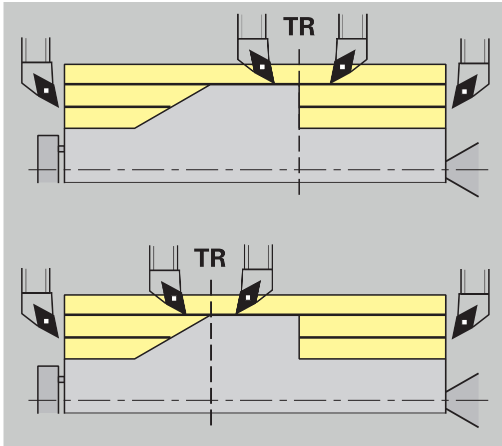
Shaft machining



For shafts, **TURN PLUS** supports rear-face machining of outside contours in addition to standard machining operations. This enables shafts to be completely machined using one setup. You can select the clamping type for the **Shaft machining AWG** (**1: Shaft/chuck** or **2: Shaft/face driver**) in the **V** input parameter in the chucking equipment dialog.

Precondition for **shaft machining**: The workpiece is clamped at spindle and tailstock.

NOTICE
Danger of collision! In **AWG** submode and during machining on the front or rear face, the control does not automatically monitor for collisions, and automatic retraction of the tailstock is not supported. Danger of collision during machining! ▶ Test the NC program in **Simulation** submode, using the graphic displayed there. ▶ Edit the NC program if required



Separation point TR

The **Separation point TR** subdivides the workpiece into a front and rear area. If no **Separation point** has been specified, **TURN PLUS** sets a separation point at the transition from the largest to a smaller diameter. Position the **separation points** at outside corners.

Tools for machining the

- Area on front side: Main machining direction – Z; and primarily left recessing or tapping tools, etc.
- Area on rear side: Main machining direction + Z; and primarily right recessing or tapping tools, etc.

Setting/changing the **Separation point**:

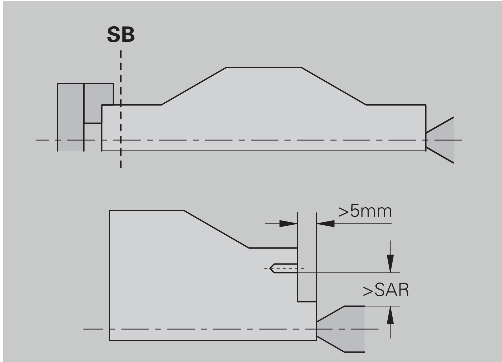
Further information: "Separation point G44", Page 306

Protection zones for drilling and milling operations

TURN PLUS machines drilling and milling contours on transverse surfaces (front or rear face) if:

- (Horizontal) distance to transverse surface > 5 mm, or
- Distance between chucking equipment and drilling/milling contour is > **SAR** (SAR: See user parameter).

If jaws are used for clamping the shaft at the spindle side, **TURN PLUS** accounts for the **Cutting limit, outside O**.



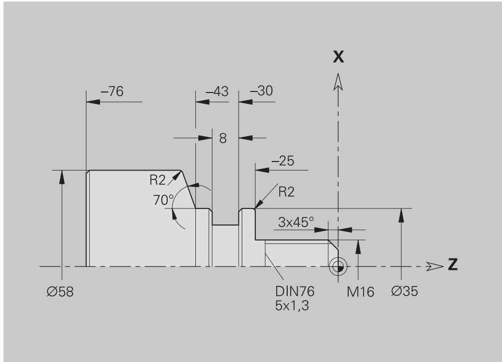
Machining information:

- **Chucking the workpiece at the spindle**: Ensure that the area, where the blank part is chucked, is premachined. Otherwise, the cutting limitation might adversely affect the machining strategies
- **Machining of bars**: **TURN PLUS** does not control the bar loader and does not move the tailstock and steady rest components. Workpiece adjustment between collet and dead center during machining operations is not supported.
- **Transverse machining**:
  - Please note that the entries made for **Machining sequence** apply to the complete workpiece and thus also to the transverse machining of shaft ends
  - In **AWG** submode, inside areas on the rear face will not be machined. If jaws are used for clamping the shaft at the spindle side, the rear face will not be machined
- **Longitudinal machining**: First the front area is machined, then the rear area
- **Collision prevention** – If machining operations are not performed without collisions, you can do the following:
  - Add a retraction of the tailstock, a positioning of the steady rest, etc. to the program
  - Add cutting limits to the program to avoid collisions
  - Suppress automatic machining in **AWG** submode by assigning the **do not machine** attribute or by entering the machining location in the **Machining sequence**
  - Define an oversize=0 for the workpiece blank. As a consequence, the front area is not machined (e.g. shafts cut to length and centered shafts).

9.5 Example

On the basis of the production drawing, the working steps for defining the contour of the workpiece blank and the finished part, the setup procedures and automatic working plan generation are explained.

- Workpiece blank: Ø60 x 80
- Material: Ck 45



Creating a program

▶ Select the **Prog** menu item

▶ Select the **New** menu item

▶ Select the **New DIN PLUS program Ctrl+N** menu item

> The control opens the **Save as** dialog box.

▶ Enter a program name

▶ Press the **Save** soft key

> The control opens the **Program head (short)** dialog box.

▶ Select a material from the fixed-word list

▶ Press the **OK** soft key

Defining the workpiece blank

▶ Select the **ICP** menu item

▶ Select the **Workpiece blank** menu item

> The control opens the **ICP editor** submode.

▶ Select the **Bar** menu item

> The **ICP editor** opens the **Bar** dialog box.

▶ Define the blank:
- **X**: **Diameter** = 60 mm
- **Z**: **Length** of workpiece blank = 80 mm
- **K**: **O-size Z** = 2 mm

▶ Press the **Save** soft key

> The **ICP editor** displays the workpiece blank.

▶ Press the **Back** soft key

Defining the basic contour

- Select the **ICP** menu item
- Select the **Finished part** menu item
- Select the **Contour** menu item
- Select the **Line** menu item
- Enter the coordinates:
  - **XS**: **Start point** of contour = 0 mm
  - **ZS**: **Starting point** of contour = 0 mm
  - **X**: **Target point** = 16 mm
- Press the **Save** soft key

- Select the **Line** menu item
- **Z**: **Target point** = –25 mm
- Press the **Save** soft key

- Select the **Line** menu item
- **X**: **Target point** = 35 mm
- Press the **Save** soft key

- Select the **Line** menu item
- **Z**: **Target point** = –43 mm
- Press the **Save** soft key

- Select the **Line** menu item
- Enter the coordinates:
  - **X**: **Target point** = 58 mm
  - **AN**: **Angle to Z axis** = 70°
- Press the **Save** soft key

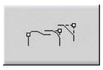
- Select the **Line** menu item
- **Z**: **Target point** = –76 mm
- Press the **Save** soft key

- Select the **Line** menu item
- **X**: **Target point** = 0 mm
- Press the **Save** soft key

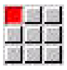
- Press the **Back** soft key

Defining form elements

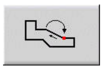
Defining chamfer at **Corner**:



- Press the **Form elements** soft key



- Select the **Cham** menu item



- Select the desired corner

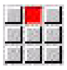
Select

- Press the **Select** soft key
- In the **Cham** dialog box: Enter **Chamfer width** = 3 mm

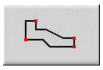
Save

- Press the **Save** soft key

Defining rounding arcs:



- Select the **Rounding** menu item



- Select the desired corner

Mark

- Select further corners if required

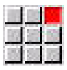
Select

- Press the **Select** soft key
- In the **Rounding** dialog box: Enter **Rounding radius** = 2 mm

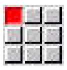
Save

- Press the **Save** soft key

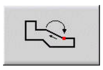
Defining an undercut:



- Select the **Undercut** menu item



- Select the **Undercut DIN 76** menu item



- Select the desired corner

Select

- Press the **Select** soft key
- > The **ICP editor** opens the **Undercut DIN 76** dialog box.
- The undercuts are already predefined in the control.

Save

- Press the **Save** soft key

Defining a recess:

- Select the **Recess** menu item
- Select the **Recess standard / G22** menu item
- Select the desired surface
- Press the **Select** soft key
- In the **Recess standard / G22** dialog box: Enter the required values
  - **Destinat. point Z** = –38 mm
  - **Inner corner I** = 27 mm
  - **Inner corner Ki** = 8 mm – activate the **Incremental** soft key
  - **Outs.rad./cham. B** = –1 mm
- Press the **Save** soft key

Defining a thread:

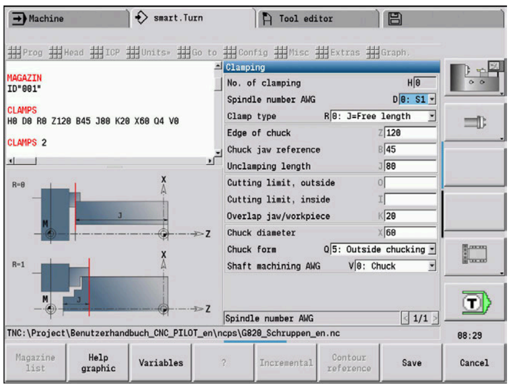
- Select the **Thread** menu item
- Select the desired surface
- Press the **Select** soft key
- > The **ICP editor** opens the **Thread** dialog box.
- The threads are already predefined in the control
- Press the **Save** soft key
- Press the **Back** soft key

Preparing the machining process, chucking

Depending on machine parameter **defaultG59** (no. 602022), **TURN PLUS** automatically calculates the required datum shift for the workpiece and activates it with **G59**.

To calculate the datum shift, **TURN PLUS** takes the following values into account:

- **Length Z** (description of workpiece blank)
- **Allowance K** (description of workpiece blank)
- **Edge of chuck Z** (description of chucking equipment or machining parameters)
- **Chuck jaw reference B** (description of chucking equipment or machining parameters)



Adding chucking equipment:

- Select the **Head** menu item
- Select the **Insert chucking equipment** menu item
- Describe the chucking equipment:
  - Enter the **No. of clamping**
  - Select the **Spindle number AWG**
  - Select the **Clamp type**
  - Enter the **Chuck jaw reference**
  - Enter the **Unclamping length**
  - Enter the **Cutting limit, outside**
  - Enter the **Cutting limit, inside**
  - Enter the **Overlap jaw/workpiece**
  - Enter the **Chuck diameter**
  - Enter the **Chuck form**
  - Select **Shaft machining AWG**
- > **TURN PLUS** takes the chucking equipment and cutting limits into account for the program creation.
- Press the **Save** soft key

Generating and saving a working plan

Generating a working plan:

- Press the **TURN PLUS** soft key
- If necessary, select the **Machining sequence**
- Press the **AWG** soft key
- Display the **AWG** control graphic

Saving the program:

- Press the **Back** soft key
- Press the **Back** soft key
- Check/edit the file name
- Press the **Save** soft key
- > **TURN PLUS** saves the NC program.

> The **AWG** submode generates the work blocks according to the **Machining sequence** and the settings of the **Machining parameters**.

9.6 Full-surface machining with TURN PLUS

Rechucking the workpiece

Refer to your machine manual.

Rechucking workpieces depends on the configuration of your machine.

The control uses subprograms for rechucking which are adapted by the machine tool builder.

TURN PLUS provides three full-surface machining variants:

- Rechucking the workpiece in the main spindle. Both setups are contained in one NC program
- Transferring the workpiece from the main spindle to the counter spindle (chuck part)
- Parting and picking-off the workpiece with the counter spindle.

TURN PLUS selects the required rechucking variant on the basis of the fixture and the machining sequence.

In the **CfgExpertProgam** machine parameters (no. 606800), a specific subprogram is defined for each rechucking variant controlling the sequence of rechucking.

Defining the chucking equipment for full-surface machining

The full-surface machining sequence is defined in the chucking-equipment dialog. You also define the datums, pick-up position and cutting limits in this dialog.

Example of first setup for full-surface machining

Parameter:

- **No. of clamping H**: CHUCKING EQUIPMENT 1
- **Spindle number AWG D**:
  - 0: Main spindle
- **Clamp type R**:
  - **0: J=Free length**
  - **1: J=Fixing length**
- **Edge of chuck Z**: No input (the **AWG** submode uses the value from the user parameters)
- **Chuck jaw reference B**: No input (the **AWG** submode uses the value from the user parameters)
- **Clamping or Unclamping length J**: Specify the clamping length / free length
- **Cutting limit, outside O**: is calculated by the **AWG** submode (if outside chucking is used)
- **Cutting limit, inside I**: Is calculated by the **AWG** submode (if inside chucking is used)
- **Overlap jaw/workpiece K**: Overlap of jaw/workpiece
- **Chuck diameter X**: Clamping diameter of workpiece blank
- **Chuck form Q**:
  - **5: Outside chucking**
  - **5: Inside chucking**
- **Shaft machining AWG V**: Select the desired **AWG** strategy

Example: Defining the first chucking equipment

...	
CHUCKING EQUIPMENT 1	
H0 D0 R0 J100 K15 X120 Q4 V0	
...	

Example of second setup for full-surface machining

Parameters:

- **No. of clamping H**: CHUCKING EQUIPMENT 2
- **Spindle number AWG D**:
  - 0: Main spindle
  - 3: Opposing spindle (depending on type of rechucking)
- **Clamp type R**:
  - **0: J=Free length**
  - **1: J=Fixing length**
- **Edge of chuck Z**: No input (the **AWG** submode uses the value from the user parameters)
- **Chuck jaw reference B**: No input (the **AWG** submode uses the value from the user parameters)
- **Clamping or Unclamping length J**: Specify the clamping length / free length
- **Cutting limit, outside O**: Is calculated by the **AWG** submode (if outside chucking is used)
- **Cutting limit, inside I**: Is calculated by the **AWG** submode (if inside chucking is used)
- **Overlap jaw/workpiece K**: Overlap of jaw/workpiece
- **Chuck diameter X**: Clamping diameter of workpiece blank
- **Chuck form Q**:
  - **5: Outside chucking**
  - **5: Inside chucking**
- **Shaft machining AWG V**: Select the desired **AWG** strategy

Example: Defining the second chucking equipment

...	
CHUCKING EQUIPMENT 2	
H0 D3 R1 J15 K-15 X68 Q4 V0	
...	

Automatic working plan generation for full-surface machining

During automatic program creation (**AWG** submode), the machining steps for the first setup are created first. Then **AWG** submode opens a dialog window that prompts for the rechucking parameters.

Default values that were calculated by **AWG** from the defined workpiece contour are already entered for the parameters in the dialog window. You can use these values or change them. After you have confirmed the values, the **AWG** submode generates the machining sequence for the second setup.

> In the machine parameters the machine tool builder defines the input parameters to be displayed in the dialog windows during rechucking.
>
> You can also include further input parameters in the dialog windows. In the **CfgExpertProgPara** machine parameters (no. 606900), select the required parameter list. In the desired parameter, enter a default value that is assigned to this parameter in the dialog window. Enter 9999999 to display the parameter without a default value.

Rechucking the workpiece in the main spindle

The subprogram for rechucking in the main spindle is defined in the user parameter **Parameter list - manual rechucking** (standard program: Rechuck_manual.ncs).

At the end of the **Machining sequence**, define a machining step with the **Rechucking Main mach. operation** and the **Full-surface machining Anc. mach. operation**.

In parameter **D** of the chucking equipment description, select the main spindle for both pieces of chucking equipment.

Example: Defining the chucking equipment

```
...
CHUCKING EQUIPMENT 1
H0 D0 R0 J80 K15 X120 Q4 V0
CHUCKING EQUIPMENT 2
H0 D0 R0 J100 K15 X120 Q4 V0
...
```

Transferring the workpiece from the main spindle to the counter spindle

The subprogram for transferring the workpiece from the main spindle to the counter spindle is defined in the user parameter **Parameter list - complete rechucking** (standard program: Rechuck_complete.ncs).

At the end of the machining sequence, define a machining step with the **Rechucking Main mach. operation** and the **Full-surface machining Anc. mach. operation**.

In parameter **D** of the chucking equipment description, select the main spindle for the first chucking equipment and the counter spindle for the second chucking equipment.

Example: Defining the chucking equipment

...	
CHUCKING EQUIPMENT 1	
H0 D0 R0 J80 K15 X120 Q4 V0	
CHUCKING EQUIPMENT 2	
H0 D0 R0 J100 K15 X120 Q4 V0	
...	

Parting and picking-off the workpiece with the counter spindle

The subprogram for parting and picking-off with the counter spindle is defined in the user parameter **Parameter list - rechucking, parting** (standard program: Rechuck_complete.ncs).

At the end of the machining sequence, define a machining step with the **Cut-off Main mach. operation** and the **Full-surface machining Anc. mach. operation**.

In parameter **D** of the chucking equipment description, select the main spindle for the first chucking equipment and the opposing spindle for the second chucking equipment.

Example: Defining the chucking equipment

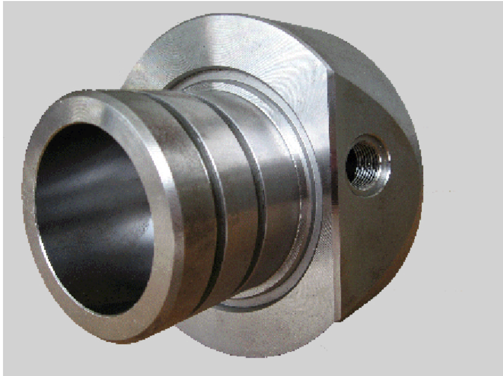
...	
CHUCKING EQUIPMENT 1	
H0 D0 R0 J80 K15 X120 Q4 V0	
CHUCKING EQUIPMENT 2	
H0 D0 R0 J100 K15 X120 Q4 V0	
...	

10

B Axis (Option 54)

10.1 Fundamentals

Tilted working plane



Refer to your machine manual.

The machine tool builder determines the scope of function and behavior of this function.

The B axis makes it possible to drill, bore and mill in oblique planes. To make programming easy, the coordinate system is tilted in such a way that you can define the drilling patterns and milling contours in the YZ plane. The actual drilling or milling operation is then performed in the tilted plane.

Further information: "Tilt working plane G16", Page 648

The separation of contour description and machining also applies to machining operations in tilted planes. Contour regeneration is not available.

Contours in tilted planes are identified with the **SURFACE Y** section code.

Further information: "SURFACE Y section", Page 87

The control supports NC program creation with the B axis in **ISO Mode** and in the **smart.Turn** operating mode.

The graphical simulation shows the machining operation in a tilted working plane in the familiar lathe and front windows, as well as in the **side view (YZ)**.

If you are using a tool with an angled tool holder, you can also use the tilted working plane without the B axis. Define the angle for the tool holder as **Angular offset RW** in the tool description.

TCPM

With the **TCPM** (Tool Center Point Management) function, you can change the behavior of the rotary axes for tilting.

With **TCPM** disabled, the axis will rotate about its mechanical center of rotation; with **TCPM** enabled, the tool tip will remain at the center of rotation and the linear axes perform a compensation movement.

The **TCPM** function allows you to machine a contour with simultaneous inclination of the B-axis.

Use the **TCPM G928** function to activate or deactivate this function.

Further information: "TCPM G928", Page 494

Tools for the B axis

An advantage of the B axis is that it allows flexible use of the tools during turning operations. 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 spindle and the counter spindle.

In this way, you need fewer tools and fewer tool changes.

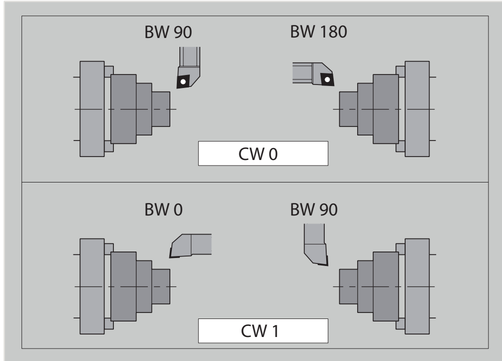
Tool data: All tools are described in the tool database by specifying the X, Z and Y dimensions as well as the compensation values. These dimensions are referenced to the **tilt angle B=0°** (reference position).

In addition, you specify the **Reverse the tool CW** parameter. It defines the working positions of tools that are not driven (turning tools).

The tilt angle of the B axis is not maintained with the tool data. This angle needs to be defined in the tool call or when inserting the tool.

Tool orientation and position display: For turning tools, the position the tool tip is calculated based on the orientation of the cutting edge.

The control calculates the tool orientation of lathe tools by means of the tool angle and point angle.

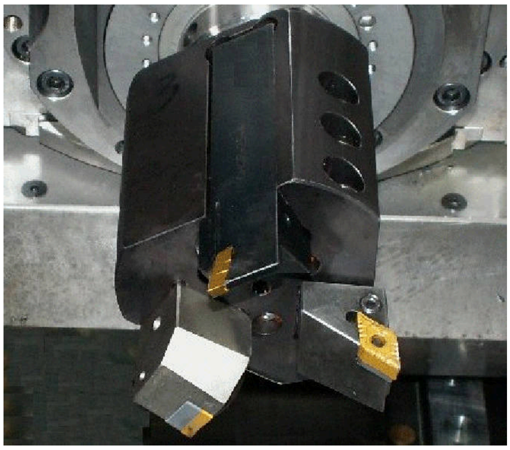


Multipoint tools for the B axis

If several tools are mounted on a tool holder, this is referred to as a **multipoint tool**. Each cutting edge (tool) of a multipoint tool is assigned a separate **ID no.** and description.

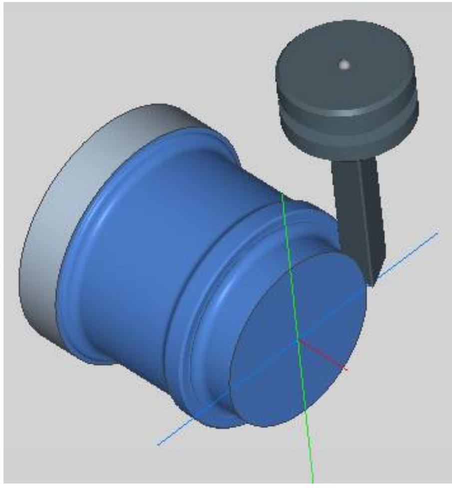
The position angle, which is identified by **CW** in the figure, is included in the tool data. When a cutting edge (tool) of a multipoint tool is activated, the control will rotate the multipoint tool into the correct position. The position is determined from the position angle, to which the offset position angle from the tool change routine is added. This allows inserting the tool either in the **normal** attitude or **upside down**.

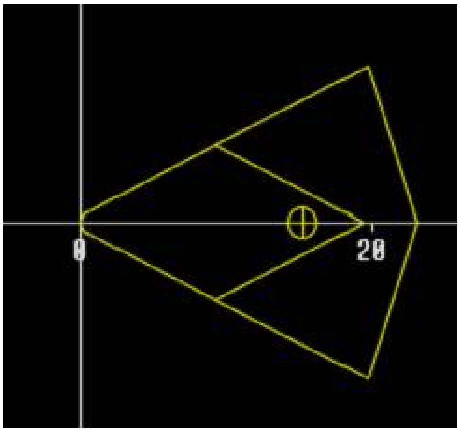
The photo shows a multipoint tool with three cutting edges.



High Dynamic Turning

High Dynamic Turning





The control supports **High Dynamic Turning** (**HDT**) developed by the company CERATIZIT. This method allows you to perform the most common turning operations with only one tool.

HDT tools are held by a milling spindle that is operated as a B axis (**B2**).

During **HDT** machining, the coordinate system is tilted such that the mechanical Y axis performs the diameter infeed.

- You define the special indexable inserts and the required tool holders in **Tool editor** mode.
  see User's Manual
- Function **G931** allows you to activate and deactivate **HDT mode**.
  see "HDT mode G931", Page 504
- You can check the machining process in **Simulation** submode. The 2-D simulation shows the tool paths and the cutting edge of the tool. The 3-D simulation shows the complete machining operation, including the tool holders.
- The machine data display notifies you about the current activation status of **HDT mode**. The control displays the same icon during simulation.
  see User's Manual

Prerequisites:

- Machine with Y axis and B axis
- Machine with milling spindle (**B2** axis)
- B-axis machining (option 54)
- The machine and control are prepared for **HDT** machining
  The machine tool builder must, for example, prepare the required kinematics descriptions and adapt the configuration settings.

Notes:

- You can use **HDT** mode both on the main spindle and on a counter spindle.
- Due to the geometry of the **HDT** tools, among other things, undercuts and inside machining are not possible in **HDT** mode.

10.2 B axis compensation

Compensation during program run

Tool compensation: Enter the determined compensation values in the tool compensation form.

Also define further functions that were active while machining the measured surface:

- **Angle in the B axis BW**
- **Reverse the tool CW**
- **Machining operation KM**
- **Angle G16**

The control converts the measured data into dimensions referencing position **B**=0 and saves it in the tool database.

Compensating the tool during program run:

Tool correct.

- Press the **Tool correct.** soft key during program run
- > The control opens the **Set tool correction** dialog box
- Enter the desired values

Save

- Press the **Save** soft key

In the **T** box (machine display), the control indicates the compensation values referencing the current B axis angle and the tool position angle.

> - The control saves the tool compensation data in the tool database, together with the other tool data
> - If the B axis is tilted, the control takes the tool compensation data into account when calculating the tool tip position

Additive correction is independent of the tool data. The compensation values are effective in the X, Y, and Z directions. Tilting the B axis has no influence on additive compensation values.

10.3 Simulation

Simulation of the tilted plane

3-D view: The simulation correctly displays tilted Y planes and referenced elements (pockets, holes, patterns etc.).

Contour graphics: The simulation displays the YZ view of the workpiece and the contours of the tilted planes in the side view. To represent the drilling patterns and milling contours perpendicularly to the tilted plane, i.e. without distortion, the simulation ignores the rotation of the coordinate system and a shift within the rotated coordinate system.

With contour graphics for tilted planes, please note the following:

- The **K** parameter in **G16** or **SURFACE Y** defines the **start** of the drilling pattern or milling contour in the Z direction.
- The drilling patterns and milling contours are drawn perpendicularly to the tilted plane. This results in a **shift** relative to the turning contour

Milling, drilling and boring operations: When you use the **side view** to display the tool paths in the tilted plane, the same rules apply as for the contour graphics.

During work on the tilted plane, the **front window** shows the outline of the tool. The tool width is simulated true to scale. In this way, you can check the overlap of milling paths. The tool paths are also represented true to scale (in perspective view) as line graphics.

In all **additional windows**, the simulation shows the tool and the cutting path when the tool is perpendicular to the relevant plane. A tolerance of +/– 5° is taken into account. When the tool is not perpendicular to the plane, it is represented as a **light dot** and the tool path is depicted as a line.

Refer to your machine manual.

The depiction of the tool carrier depends on the machine.

The graphic displays a tool carrier if the following requirements are fulfilled:

- The machine tool builder has saved a description of the tool carrier, e.g. B axis head
- You have assigned a tool holder to a tool

Example: Contour in tilted plane

...	
FINISHED PART	
N2 G0 X0 Z0	
N3 G1 X50	
N4 G1 Z-50	
N5 G1 X0	
N6 G1 Z0	
SURFACE Y X50 C0 B80 I25 K-10 H0	
N7 G386 Z0 Ki10 B-30 X50 C0	Single surface
SURFACE Y X50 C0 B20 I25 K-20 H1	
N8 G384 Z-10 Y10 X50 R10 P5	Full circle
...	

Displaying the coordinate system

The simulation can show the shifted/rotated coordinate system in the **lathe window** if required.

To use this feature, you need to stop the simulation.

Displaying the coordinate system:

-/+

- Press the **-/+** key
- > The simulation displays the current coordinate system

The coordinate system disappears when the next command is simulated or when you press the **-/+** key once again.

Position display with the B and Y axes

The following boxes of the display **cannot be edited**:

- **N**: Block number of the NC source block
- **X**, **Z**, **C**: Position values (actual values)

The other boxes can be set with the **Split-Screen Layout** key (three arrows arranged in a circle):

- Default settings (values of the selected slide)
  - **Y**: Position value (actual value)
  - **T**: Tool data with turret pocket, (in "(..)") and **ID no.**
- B axis settings
  - **B**: Tilt angle of the B axis
  - **G16**/**B**: Angle of the tilted plane

11

Overview of Units (Option 9)

11.1 Units – "Turning" group

"Roughing" group

Unit	Description	Page
G810_ICP	**G810 Longitudinal in ICP** Roughing an ICP contour longitudinally	Page 106
G820_ICP	**G820 Transverse in ICP** Roughing an ICP contour transversely	Page 108
G830_ICP	**G830 Contr. parallel in ICP** Roughing parallel to the contour in ICP	Page 110
G835_ICP	**G835 Bidirectional in ICP** Roughing an ICP contour in two directions	Page 111
G810_G80	**G810 Longitudinal, direct** Longitudinal roughing with direct contour input	Page 112
G820_G80	**G820 Transverse, direct** Transverse roughing with direct contour input	Page 175
G895_ICP	**G895 Simultaneous roughing**	Page 115

"Finishing" group

Unit	Description	Page
G890_ICP	**G890 Contouring in ICP** Finishing an ICP contour	Page 171
G890_G80_L	**G890 Contouring, direct longitdnl.** Longitudinal finishing with direct contour input	Page 173
G890_G80_P	**G890 Contouring, direct transverse** Transverse finishing with direct contour input	Page 175
G85x_DIN_E_F_G	**G890 Relief, type E,F,DIN76** Finishing the undercuts according to DIN509 type E and F and the thread undercut DIN76	Page 177
MEASURE_G809	**G809 Measuring cut**	Page 179
G891_ICP	**G891 Simultaneous finishing**	Page 180

"Recessing" group

Unit	Description	Page
G860_ICP	**G860 Contour recess in ICP** Recessing an ICP contour	Page 119
G869_ICP	**G869 Recess turning in ICP** Recess turning an ICP contour	Page 120
G860_G80	**G860 Contour recess, direct** Contour recessing with direct contour input	Page 122
G869_G80	**G869 Recess turning, direct** Recess turning with direct contour input	Page 123
G859_Cut_off	**G859 Parting** Parting a bar with direct position input	Page 124
G85x_Cut_H_K_U	**G85X Undercutting (H,K,U)** Make undercuts of type H, K and U	Page 125
G870_ICP	**G870 ICP Recessing** Machining a recess	Page 125

"Thread" group

Unit	Description	Page
G32_MAN	**G32 Thread, direct** Thread with direct contour definition	Page 186
G31_ICP	**G31 Thread, ICP** Thread on any desired ICP contour	Page 187
G352_API	**G352 API thread** API thread with direct contour definition	Page 189
G32_KEG	**G32 Tapered thread** Tapered thread with direct contour definition	Page 190

11.2 Units – "Drilling" group

"Centric drilling" group

Unit	Description	Page
G74_Zentr	**G74 Centric drilling** Drilling and pecking with X=0	Page 127
G73_Zentr	**G73 Centric tapping** Tapping with X=0	Page 128

"ICP drilling, C axis" group

Unit	Description	Page
G74_ICP_C	**G74 Drilling in ICP C** Drilling and pecking with ICP pattern	Page 151
G73_ICP_C	**G73 Tapping in ICP C** Tapping with ICP pattern	Page 152
G72_ICP_C	**G72 Boring, cntrsinkg ICP C** Countersinking with ICP pattern	Page 154
G75_BF_ICP_C	**G75 Bore milling ICP C face** Bore milling with ICP pattern on the front face	Page 154
G75_EN_ICP_C	**G75 Deburring ICP C face** Deburring with ICP pattern on the front face	Page 155
G75_BF_ICP_C_MANT	**G75 Bore milling ICP C lateral** Bore milling with ICP pattern on the lateral surface	Page 156
G75_EN_ICP_C_MANT	**G75 Deburring ICP C lateral** Deburring with ICP pattern on the lateral surface	Page 157

"C-axis face drilling" group

Unit	Description	Page
G74_Bohr_Stirn_C	**G74 Single hole** Drilling and pecking a single hole	Page 131
G74_Lin_Stirn_C	**G74 Linear pattern drilling** Drilling and pecking a linear hole pattern	Page 133
G74_Cir_Stirn_C	**G74 Circ. pattern drilling** Drilling and pecking a circular hole pattern	Page 135
G73_Gew_Stirn_C	**G73 Tapping** Tapping a single hole	Page 137
G73_Lin_Stirn_C	**G73 Thread, linear pattern** Tapping a linear hole pattern	Page 138
G73_Cir_Stirn_C	**G73 Thread, circ. pattern** Tapping a circular hole pattern	Page 139

"C-axis lateral surface drilling" group

Unit	Description	Page
G74_Bohr_Mant_C	**G74 Single hole** Drilling and pecking a single hole	Page 141
G74_Lin_Mant_C	**G74 Linear pattern drilling** Drilling and pecking a linear hole pattern	Page 143
G74_Cir_Mant_C	**G74 Circ. pattern drilling** Drilling and pecking a circular hole pattern	Page 145
G73_Gew_Mant_C	**G73 Tapping** Tapping a single hole	Page 147
G73_Lin_Mant_C	**G73 Thread, linear pattern** Tapping a linear hole pattern	Page 148
G73_Cir_Mant_C	**G73 Thread, circ. pattern** Tapping a circular hole pattern	Page 149

11.3 Units – "Predrilling in C axis" group

"Predrilling in C-axis, face" group

Unit	Description	Page
DRILL_STI_KON_C	**Predrill face, G840 contour mill C** Determine the predrilling position and machine a hole	Page 158
DRILL_STI_840_C	**Predrill on face, G840 ICP in C** Determine the predrilling position and machine a hole	Page 162
DRILL_STI_TASC	**Predrill face G845, pocket mill in C** Determine the predrilling position and machine a hole	Page 160
DRILL_STI_845_C	**Predrill on face, G845 ICP in C** Determine the predrilling position and machine a hole	Page 163

"Predrilling in C axis, lateral surface" group

Unit	Description	Page
DRILL_MAN_KON_C	**Predrill latrl., G840 contour mill C** Determine the predrilling position and machine a hole	Page 164
DRILL_MAN_840_C	**Predrill lateral surf., G840 ICP in C** Determine the predrilling position and machine a hole	Page 168
DRILL_MAN_TAS_C	**Predrill face G845, pocket mill in C** Determine the predrilling position and machine a hole	Page 166
DRILL_MAN_845_C	**Predrill lateral surf., G845 ICP in C** Determine the predrilling position and machine a hole	Page 169

11.4 Units – "Milling in C axis" group

"Milling in C-axis, face" group

Unit	Description	Page
G791_Nut_Stirn_C	**G791 Linear slot** Milling a linear slot	Page 192
G791_Lin_Stirn_C	**G791 Linear slot pattern** Milling of linear slots in a linear pattern	Page 193
G791_Cir_Stirn_C	**G791 Circular slot pattern** Milling of linear slots in a circular pattern	Page 194
G797_STIRNFR_C	**G797 Face milling** Milling various figures as islands	Page 195
G797_ICP	**G797 face milling ICP** Milling closed contours as islands	Page 207
G799_GewindeFR_C	**G799 Thread milling** Inside thread milling in a single hole	Page 196
G840_FIG_STIRN_C	**G840 Contour mllg., figures** Milling figures inside, outside and on the contour	Page 197
G84X_FIG_STIRN_C	**G84X Pocket millg., figures** Roughing out closed figures, inside	Page 199
G801_GRA_STIRN_C	**G801 Engraving** Engraving characters strings on the face	Page 201

"C-axis ICP face milling" group

Unit	Description	Page
G840_Kon_C_STIRN	**G840 Contour milling, ICP** Machining ICP contours on the front face inside, outside, and on the contour	Page 202
G845_TAS_C_STIRN	**G845 Pocket milling, ICP** Inside rough-out of closed ICP contours on the front face	Page 204
G840_ENT_C_STIRN	**G840 Deburring** Deburring ICP contours on the front face	Page 223
G797_ICP	**G797 face milling ICP** Milling ICP contours on the front face	Page 223
G847_KON_C_STIRN	**G847 Contour milling** Rough-out of ICP contours on the front face using trochoidal milling	Page 207
G848_TAS_C_STIRN	**G848 Pocket milling** Rough-out of figures on the front face using trochoidal milling	Page 210

"C-axis lateral surface milling" group

Unit	Description	Page
G792_NUT_MANT_C	**G792 Linear slot** Milling a linear slot	Page 212
G792_LIN_MANT_C	**G792 Linear slot pattern** Milling of linear slots in a linear pattern	Page 213
G792_CIR_MANT_C	**G792 Circular slot pattern** Milling of linear slots in a circular pattern	Page 214
G798_WENDEL-NUT_C	**G798 Helical slot milling** Milling a thread-shaped helical slot	Page 215
G840_FIG_MANT_C	**G840 Contour mllg., figures** Milling figures inside, outside and on the contour	Page 216
G84x_FIG_MANT_C	**G84X Pocket millg., figures** Roughing out closed figures, inside	Page 222
G802_GRA_MANT_C	**G802 Engraving** Engraving characters strings on the lateral surface	Page 223

"C-axis ICP lateral surface milling" group

Unit	Description	Page
G840_Kon_C_Mant	**G840 Contour milling, ICP** Machining ICP contours on the lateral surface inside, outside, and on the contour	Page 221
G845_TAS_C_MANT	**G845 Pocket milling, ICP** Inside rough-out of closed ICP contours on the lateral surface	Page 222
G840_ENT_C_MANT	**G840 Deburring** Deburring ICP contours on the lateral surface	Page 223
G847_KON_C_MANT	**G847 Contour milling** Rough-out of ICP contours on the lateral face using trochoidal milling	Page 224
G848_TAS_C_MANT	**G848 Pocket milling** Rough-out of figures on the lateral surface using trochoidal milling	Page 226

11.5 Units – "Drilling, predrilling in Y axis" group

"ICP drilling, Y axis" group

Unit	Description	Page
G74_ICP_Y	**G74 Drilling in ICP Y** Drilling and pecking with ICP pattern	Page 236
G73_ICP_Y	**G73 Tapping in ICP Y** Tapping with ICP pattern	Page 237
G72_ICP_Y	**G72 Boring, cntrsinkg ICP Y** Countersinking with ICP pattern	Page 238
G75_BF_ICP_Y	**G75 Bore milling ICP Y face** Bore milling with ICP pattern on the front face	Page 239
G75_EN_ICP_Y	**G75 Deburring ICP Y face** Deburring with ICP pattern on the front face	Page 240
G75_BF_ICP_Y_MANT	**G75 Bore milling ICP Y lateral** Bore milling with ICP pattern on the lateral surface	Page 241
G75_EN_ICP_Y_MANT	**G75 Deburring ICP Y lateral** Deburring with ICP pattern on the lateral surface	Page 242

"Predrilling in Y axis" group

Unit	Description	Page
DRILL_STI_840_Y	**G840 ICP predrilling, contour milling in XY plane** Determine the predrilling position and machine a hole	Page 243
DRILL_STI_845_Y	**G845 ICP predrilling, pocket milling in XY plane** Determine the predrilling position and machine a hole	Page 244
DRILL_MAN_840_Y	**G840 ICP predrilling, contour milling in YZ plane** Determine the predrilling position and machine a hole	Page 246
DRILL_MAN_845_Y	**G845 ICP predrilling, pocket milling in YZ plane** Determine the predrilling position and machine a hole	Page 247

11.6 Units – "Milling in Y axis" group

"Milling in front face" group (XY plane)

Unit	Description	Page
G840_Kon_Y_Stirn	**G840 Contour milling** Machining contours in the XY plane inside, outside and on the contour	Page 249
G845_Tas_Y_Stirn	**G845 Pocket milling** Inside rough-out of closed contours in the XY plane	Page 250
G840_ENT_Y_STIRN	**G840 Deburring** Deburring contours in the XY plane	Page 252
G801_GRA_STIRN_C	**G841 Single surf.** Milling a single surface (flat) in the XY plane	Page 253
G840_Kon_C_STIRN	**G843 Centric polygon** Milling a centric polygon in the XY plane	Page 254
G803_GRA_Y_STIRN	**G803 Engraving** Engraving character strings in the XY plane	Page 255
G800_GEW_Y_STIRN	**G800 Thread milling** Milling a thread in an existing hole in the XY plane	Page 256
G847_KON_Y_STIRN	**G847 Contour milling** Rough-out of ICP contours on the XY plane using trochoidal milling	Page 257
G848_TAS_Y_STIRN	**G848 Pocket milling** Rough-out of figures on the XY plane using trochoidal milling	Page 259

"Milling in lateral surface" group (YZ plane)

Unit	Description	Page
G840_Kon_Y_Mant	**G840 Contour milling** Machining contours in the YZ plane inside, outside and on the contour	Page 261
G845_Tas_Y_Mant	**G845 Pocket milling** Inside rough-out of closed contours in the YZ plane	Page 262
G840_ENT_Y_MANT	**G840 Deburring** Deburring contours in the YZ plane	Page 263
G801_GRA_STIRN_C	**G841 Single surf.** Milling a single surface (flat) in the YZ plane	Page 264
G840_Kon_C_STIRN	**G843 Centric polygon** Milling a centric polygon in the YZ plane	Page 265
G804_GRA_Y_MANT	**G803 Engraving** Engraving character strings in the YZ plane	Page 266
G806_GEW_Y_MANT	**G800 Thread milling** Milling a thread in an existing hole in the YZ plane	Page 267
G847_KON_Y_MANT	**G847 Contour milling** Rough-out of ICP contours on the YZ plane using trochoidal milling	Page 268
G848_TAS_Y_MANT	**G848 Pocket milling** Rough-out of figures on the YZ plane using trochoidal milling	Page 270

11.7 Units – "Special units" group

Unit	Description	Page
START	**Program beginning (START)** For functions required at the beginning of the program	Page 228
C_AXIS_ON	**C axis ON** Activate C-axis interpolation	Page 229
C_AXIS_OFF	**C axis OFF** Deactivate C-axis interpolation	Page 230
SUBPROG	**Subprogram call** Calling any desired subprogram	Page 230
REPEAT	**Process logic - repetition** Describing a WHILE loop to repeat parts of the program	Page 231
END	**Program end (END)** For functions required at the end of the program	Page 232
ROTWORKPLANE	**Tilt plane** Tilting the working plane	Page 233

12

Overview of G Codes

12.1 Section codes

Program section codes

Program head	Page
Program head / PROGRAM HEAD	Page 82
Turret / TURRET	Page 85
Clamping / CHUCKING EQUIPMENT	Page 84
Magazine / MAGAZINE	Page 85
Contour group / Contour group	Page 85
Manual tool / MANUAL TOOL	Page 85

Contour definition	page
Workpiece blank / BLANK	Page 86
Aux. workpiece blank / AUX. BLANK	Page 86
Finished part / FINISHED PART	Page 86
Auxiliary contour / AUXILIARY CONTOUR	Page 86

C-axis contours	Page
Front / FRONT	Page 86
REAR SIDE / REAR SIDE	Page 86
Lateral / LATERAL	Page 86

Y-axis contours	Page
Face Y / FRONT Y	Page 86
REAR SIDE Y / REAR SIDE Y	Page 86
Lateral surface in Y / SURFACE Y	Page 87

Workpiece machining	Page
MACHINING / MACHINING	Page 88
End / END	Page 88

Subprograms	Page
Subprogram / SUBROUTINE	Page 88
Return / RETURN	Page 88

Others	Page
CONST	Page 89
VAR	Page 89
ZUORDNUNG [ASSIGNMENT]	Page 90

12.2 Overview of G commands in the CONTOUR section

G commands for turning contours

Workpiece-blank definition		Page
G20-Geo	**Chuck part cyl./tube**	Page 287
G21-Geo	**Cast part**	Page 287

Basic contour elements		Page
G0-Geo	**Starting point**	Page 288
G1-Geo	**Distance**	Page 289
G2-Geo	**Circular arc cw**	Page 290
G3-Geo	**Circular arc ccw**	Page 290
G12-Geo	**Circ. arc. abs. cw**	Page 291
G13-Geo	**Circ. arc. abs. ccw**	Page 291

Contour form elements		Page
G22-Geo	**Recess (standard)**	Page 293
G23-Geo	**Recess (general)**	Page 295
G24-Geo	**Thread with undercut**	Page 296
G25-Geo	**Undercut**	Page 297
G34-Geo	**Thread (standard)**	Page 300
G37-Geo	**Thread (general)**	Page 301
G49-Geo	**Bore hole (centric)**	Page 303

Help commands for contour definition		Page
	Overview: Attributes for contour description	Page 304
G10-Geo	**Peak-to-valley h.**	Page 304
G38-Geo	**Feed rate reduction**	Page 305
G44	**Separation point**	Page 306
G52-Geo	**Contour-parallel oversize**	Page 306
G95-Geo	**Feed per rotation**	Page 307
G149-Geo	**Additive correction**	Page 308

G commands for C-axis contours

Overlapping contours		Page
G308-Geo	Start pocket/isl.	Page 309
G309-Geo	End of pocket/island	Page 309

Front and rear face contours		Page
G100-Geo	Start point	Page 315
G101-Geo	Linear face	Page 315
G102-Geo	Circ. arc face cw	Page 316
G103-Geo	Circ. arc face cw	Page 316
G300-Geo	Hole, face	Page 317
G301-Geo	Linear slot, face	Page 391
G302-Geo	Slot cw, face	Page 391
G303-Geo	Slot ccw, face	Page 391
G304-Geo	Full circle, face	Page 392
G305-Geo	Rectangle, face	Page 392
G306-Geo	Text for front face C	Page 319
G307-Geo	Polygon on front face	Page 393
G401-Geo	Pattern linear face	Page 320
G402-Geo	Pattern circ. face	Page 321
G405-Geo	DataMatrix face C	Page 322

Lateral surface contours		Page
G110-Geo	Start point	Page 323
G111-Geo	Surface linear	Page 323
G112-Geo	Circ.arc surface ccw	Page 324
G113-Geo	Circ.arc surface ccw	Page 324
G310-Geo	Hole, surface	Page 325
G311-Geo	Linear slot, surf.	Page 325
G312-Geo	Slot cw, surface	Page 326
G313-Geo	Slot ccw, surface	Page 326
G314-Geo	Full circle, surf.	Page 326
G315-Geo	Rectangle, surface	Page 327
G316-Geo	Text for lateral face C	Page 327
G317-Geo	Polygon on lateral surface	Page 328
G411-Geo	Pattern linear surf	Page 328
G412-Geo	Pattern circ. surf.	Page 329
G415-Geo	DataMatrix lateral C	Page 330

G commands for Y-axis contours

XY plane		**Page**
G170-Geo	**Start point**	Page 629
G171-Geo	**Linear face**	Page 629
G172-Geo	**Circ. arc face ccw**	Page 630
G173-Geo	**Circ. arc face ccw**	Page 630
G370-Geo	**Hole in XY plane**	Page 631
G371-Geo	**Linear slot in XY plane**	Page 632
G372-Geo	**Slot cw in XY plane**	Page 632
G373-Geo	**Slot ccw in XY plane**	Page 632
G374-Geo	**Full circle in XY plane**	Page 633
G375-Geo	**Rectangle in XY plane**	Page 633
G377-Geo	**Polygon in XY plane**	Page 634
G378-Geo	**Text for front face Y**	Page 635
G471-Geo	**Linear pattern, face**	Page 635
G472-Geo	**Circular patt., face**	Page 636
G475-Geo	**DataMatrix XY plane**	Page 637
G376-Geo	**Single surface in XY plane**	Page 634
G477-Geo	**Centr. polygon, face**	Page 637

YZ plane		**Page**
G180-Geo	**Start point**	Page 638
G181-Geo	**Surface linear**	Page 638
G182-Geo	**Circ.arc surface ccw**	Page 639
G183-Geo	**Circ.arc surface ccw**	Page 639
G380-Geo	**Hole in YZ plane**	Page 640
G381-Geo	**Linear slot in YZ plane**	Page 640
G382-Geo	**Slot cw in YZ plane**	Page 641
G383-Geo	**Slot ccw in YZ plane**	Page 641
G384-Geo	**Full circle in YZ plane**	Page 641
G385-Geo	**Rectangle in YZ plane**	Page 642
G387-Geo	**Polygon in YZ plane**	Page 642
G388-Geo	**Text for lateral face Y**	Page 643
G481-Geo	**Lin. patt., side v.**	Page 643
G482-Geo	**Circ. patt, side v.**	Page 644
G485-Geo	**DataMatrix YZ plane**	Page 645
G386-Geo	**Single surface in XY plane**	Page 645
G487-Geo	**Centr. polyg., surf.**	Page 646

12.3 Overview of G commands in the MACHINING section

G commands for turning contours

Tool positioning without machining		Page
G0	Rapid traverse	Page 331
G14	Tool change point	Page 332
G140	Tool change point	Page 332
G701	Rap.trav.mach.coord.	Page 331
G977	LIFTOFF	Page 499

Simple linear and circular movements		Page
G1	Linear movement	Page 333
G2	Circular arc ccw	Page 334
G3	Circular arc ccw	Page 334
G12	Circular arc ccw	Page 335
G13	Circular arc ccw	Page 335

Feed rate and spindle speed		Page
Gx26	Speed limitation	Page 336
G64	Interrupt Feed	Page 337
G48	Reduce rapid traverse	Page 336
Gx93	Feed per tooth	Page 338
G94	Constant feed	Page 338
Gx95	Feed per rotation	Page 339
Gx96	Cutting speed	Page 339
Gx97	Spindle speed	Page 340

Oversizes		Page
G50	Switch off oversize	Page 347
G52	Switch off oversize	Page 347
G57	Oversize axis parall	Page 347
G58	Oversize cont.parallel	Page 348

Datum shifts		Page
	Overview of datum shifts	Page 343
G51	**Datum shift**	Page 344
G53/G54/G55	**Datum shift**	Page 345
G56	**Datum shift**	Page 345
G59	**Datum shift**	Page 346
G152	**Datum shift in C**	Page 440
G920	**Shift OFF**	Page 493
G921	**Shift and tool dimensions OFF**	Page 493
G980	**Shift ON**	Page 500
G981	**Shift and tool dimensions ON**	Page 500

Safety clearances		Page
G47	**Safety clearance**	Page 349
G147	**Safety clearance**	Page 349

Tool-tip radius compensation (TRC/MCRC)		Page
G40	**TRC/MCRC OFF**	Page 341
G41	**Switch on the TRC (left)**	Page 342
G42	**Switch on the TRC (right)**	Page 341

Tools, types of compensation		Page
T	**Tool**	Page 350
G148	**Correction of cut**	Page 351
G149	**Additive correction**	Page 352
G150	**Right tool tip**	Page 353
G151	**Left tool tip**	Page 353

Cycles for turning

Simple turning cycles		Page
G80	Cycle end	Page 390
G81	Simpl.longit.drill.	Page 557
G82	Simpl. face drill.	Page 558
G83	Repeat contour cycle	Page 559
G86	Simple recessing cycle	Page 559
G87	Cycle radius	Page 562
G88	Cycle chamfer	Page 562

Drilling cycles		Page
G36	Thread boring	Page 351
G71	Simple drilling	Page 423
G72	Boring/cnt-sink	Page 425
G73	Tapping	Page 426
G74	Deep boring	Page 428

Undercuts		Page
G25	Undercut	Page 297
G85	Cycle undercut	Page 414
G851	Undercut DIN 509 E	Page 415
G852	Undercut DIN 509 F	Page 416
G853	Undercut DIN 76	Page 417
G856	FORM U undercut	Page 419
G857	FORM H undercut	Page 420
G858	FORM K undercut cy.	Page 420

Contour-based turning cycles		Page
G740	Repeat contour cycle	Page 368
G741	Repeat contour cycle	Page 368
G810	Longitud. roughing	Page 356
G820	Face roughing	Page 359
G830	Contour parallel	Page 362
G835	Contour cycle, bidirectional	Page 364
G860	Simple recessing cycle	Page 366
G869	Recessing	Page 370
G870	Recess cycle	Page 373
G890	Finish contour	Page 374
G891	Cycle for simultaneous finishing	Page 384
G895	Simultaneous roughing cycle	Page 378

Thread cycles		Page
G31	**Universal tapping cycle**	Page 398
G32	**Single thread cycle**	Page 403
G33	**Thread single path**	Page 405
G35	**ISO thread (metr.)**	Page 407
G350	**Single thread**	
G351	**Extended thread**	
G352	**Tapered API thread G352**	Page 408
G36	**Thread boring**	Page 422
G38	**Contour thread**	Page 410

Parting		Page
G859	**Cut-off cycle**	Page 413

C-axis machining

C axis		Page
G120	Reference diameter	Page 440
G152	Datum shift in C	Page 440
G153	Standardize C axis	Page 441
G154	Short path in C	Page 441

Single path—Front and rear face machining		Page
G100	Rap. trav. face	Page 442
G101	Linear face	Page 443
G102	Circ. arc face ccw	Page 445
G103	Circ. arc face ccw	Page 445

Single path—Lateral-surface machining		Page
G110	Start point	Page 447
G111	Surface linear	Page 447
G112	Circ.arc surface ccw	Page 449
G113	Circ.arc surface ccw	Page 449

Figures—Front and rear face machining		Page
G301	Linear slot, face	Page 391
G302	Slot cw, face	Page 391
G303	Slot ccw, face	Page 391
G304	Full circle, face	Page 392
G305	Rectangle, face	Page 392
G307	Polygon on front face	Page 393

Figures—Lateral-surface machining		Page
G311	Linear slot, surf.	Page 393
G312	Slot cw, surface	Page 394
G313	Slot ccw, surface	Page 394
G314	Full circle, surf.	Page 394
G315	Rectangle, surface	Page 395
G317	Polygon on lateral surface	Page 395

Milling cycles, face		Page
G791	Linear slot, face	Page 453
G793	Contour milling cycle, face	Page 455
G797	Area milling	Page 460
G799	Thread milling	

Milling cycles, lateral surface		Page
G792	Linear slot, surf.	Page 454

Milling cycles, lateral surface		Page
G794	Contour milling cycle, lateral	Page 457
G798	Hel. slot milling	Page 462

Predrilling cycles		Page
G840	Contour milling	Page 464
G845	Pocket milling - roughing	Page 473

Contour and pocket milling cycles		Page
G840	Contour milling	Page 466
G840	Deburring	Page 470
G845	Pocket milling - roughing	Page 474
G846	Pocket milling - finishing	Page 478

Engraving cycles		Page
G801	Engraving in XC	Page 487
G802	Engraving in ZC	Page 488

Y-axis machining

Working planes		Page
G16	Tilt working plane	Page 648
G17	XY plane	Page 647
G18	XZ plane	Page 647
G19	YZ plane	Page 647
G160	Tilting the working plane with spatial angle	Page 649

Tool positioning without machining		Page
G0	Start point	Page 650
G14	Tool change point	Page 650
G701	Pattern linear face	Page 651

Simple linear and circular movements		Page
G1	Linear movement	Page 652
G2	Circular arc ccw	Page 653
G3	Circular arc ccw	Page 653
G12	Circular arc ccw	Page 654
G13	Circular arc ccw	Page 654

Milling cycles		Page
G841	Area milling - roughing	Page 655
G842	Area milling - finishing	Page 656
G843	Centric polygon - roughing	Page 657
G844	Centric polygon - finishing	Page 658
G845	Predrilling, pocket milling	Page 660
G845	Pocket milling - roughing	Page 661
G846	Pocket milling - finishing	Page 664
G847	Trochoidal contour milling	Page 480
G848	Trochoidal pocket milling	Page 482
G800	Thread milling in XY	Page 668
G806	Thread milling in YZ	Page 669
G807	Skiving	Page 669
G808	Hobbing	Page 674

Engraving cycles		Page
G803	Engraving in XY	Page 666
G804	Engraving in YZ	Page 667
	Character set for engraving	Page 484

Variable programming, program branches

Programming with variables		Page
# variables	Variable types	Page 515
PARA	**Read configuration data**	Page 528
CONST	**Constant definition...**	Page 529
VAR	**Assignment of variables...**	Page 529

Subprograms		Page
	Subprogram call	Page 542

Data input and data output		Page
INPUT	Input (# variables)	Page 534
WINDOW	Open output window (# variables)	Page 533
PRINT	Output (# variables)	Page 534

Program Branches, Program Repeats		Page
IF..THEN..	Program branching	Page 535
WHILE..	Program repeat	Page 538
SWITCH..	Program branching	Page 540

Other G codes

Other G codes		Page
G4	Period of dwell	Page 490
G7	Prec. stop on	Page 490
G8	Prec. st. off	Page 491
G9	Block precision	Page 491
G30	Converting and mirroring	Page 547
G44	Separation point	Page 306
G60	Switch off prot.zone	Page 491
G62	One-sided synchron. (option 153)	Page 549
G63	Synchronous start of slides (option 153)	Page 550
G65	Chucking equipment	Page 490
G67	Wrkpc. blank contour	Page 490
G99	Select/position the contour	Page 548
G162	Synchr. marking (option 153)	Page 548
G238	Fingerprint (option 155)	Page 511
G702	Contour follow-up	Page 489
G703	Contour follow-up	Page 489
G720	Spindle synchronization	Page 551
G725	Eccentric turning	Page 506
G726	Transition to eccentric	Page 507
G727	Eccentric X	Page 509
G728	Compens. for helical teeth	Page 510
G901	Act.value in var.	Page 491
G902	Zero point in variables	Page 491
G903	Lag error in variables	Page 492
G904	Fill variable memory	Page 492
G905	Offset C angle	Page 552
G908	Blockwise 100%	Page 492
G909	Interpret. stop	Page 492
G910	Switch on measure.	Page 623
G911	Activate measuring path monitoring	Page 624
G912	Actual value detect.	Page 624
G913	End measurement	Page 624
G914	Deactivate measuring path monitoring	Page 624
G916	Traversing to a fixed stop	Page 553
G919	Spind. override 100%	Page 493
G920	Shift OFF	Page 493
G921	Shift and tool dimensions OFF	Page 493
G923	Handwheel offset in thread	Page 184

Other G codes		Page
G924	**Fluctuating speed**	Page 493
G925	**Force reduction**	Page 502
G927	**Convert lengths**	Page 494
G928	**TCPM**	Page 494
G930	**Sleeve monitoring**	Page 503
G931	**HDT mode**	Page 504
G932	**Look-ahead parameter**	Page 495
G939	**Component monitoring** (option 155)	Page 513
G940	**Convert variables automat.**	Page 496
G941	**DNC message**	Page 498
G976	**Misalignment compensation**	Page 498
G977	**LIFTOFF**	Page 499
G980	**Shift ON**	Page 500
G981	**Shift and tool dimensions ON**	Page 500
G995	**Monitoring zone**	Page 500
G996	**Load monitoring**	Page 501

Index

A

About this manual........................... 32
Actual values in variable G901... 491
Additive correction G149............ 352
Additive correction G149-Geo.... 308
Address parameters.................... 280
Angular measurement G787...... 620
API thread G352.......................... 408
Approach, smart.Turn.................. 104
attributes for contour description..... 304
Automatic job.................................. 95
Automatic working plan generation.. 683
AWG... 683
full-surface machining............ 719
list of machining..................... 688
machining sequence............... 685
machining sequence, editing. 686

B

B axis
compensation during program run... 725
flexible use of tools................. 723
fundamentals........................... 722
simulation................................ 726
TCPM.. 494
B axis
HDT.. 724
Blank contour G67 (for graphics).. 490
Bore hole
centric G49–Geo..................... 303
front face G300-Geo.............. 317
Boring G72.................................... 425

C

Calculating hole positions
G840... 464
G845... 473
G845 (Y axis)........................... 660
Calculation of pitch circle G786. 618
Calibrating
touch probe, standard G747.. 597
touch probe via two points G748... 599
Calibrating touch probes............ 597
Cast part G21-Geo...................... 287
C axis
G code...................................... 440
offset C angle G905............... 552
C-axis contours
Fundamentals......................... 309
Centric polygon
finishing G844......................... 658
roughing G843......................... 657
XY plane G477-Geo................ 637
YZ plane G487-Geo................ 646
chamfer G88................................. 562
Character set............................... 484
Chucking equipment in simulation G65.. 84
Chuck part bar/tube G20-Geo.... 287
Circular arc.......................... 334, 335
front face contour G102-/G103-Geo.. 316
front face G102/G103........... 445
lateral surface contour G112-/G113-Geo................................ 324
lateral surface G112/G113.... 449
milling G12, G13..................... 654
milling G2, G3.......................... 653
turning contour G12-/G13-Geo.. 291
turning contour G2-/G3-Geo.. 290
XY plane G172-/G173-Geo..... 630
YZ plane G182/G183-Geo...... 639
Circular measurement G785...... 616
Circular movement...................... 333
Circular pattern with circular slots.. 312
Circular slot
front face G302-/G303-Geo... 318
lateral surface G312-/G313-Geo.. 326
XY plane G372/G373-Geo...... 632
YZ plane G382/G383-Geo...... 641
Clamping in simulations G65..... 490
Code
ALLOCATION............................. 90
CONST.. 89
END... 88
RETURN...................................... 88
VAR... 89
Compensation.............................. 350
B axis.. 725
Compensation of right-hand/left-hand tool tip G150/G151........... 353
Conditional block run.................. 535
Configuration data, reading........ 528
Connection between geometry and machining commands............... 567
C axis – front face.................. 568
C axis – lateral surface........... 568
turning...................................... 567
Constant cutting speed Gx96..... 339
Contour
in XY plane.............................. 629
in YZ plane.............................. 638
simple G80.............................. 390
Contour and figure milling cycle on lateral surface G794.................... 457
Contour-based turning cycle...... 354
Contour finishing G890............... 374
Contour follow-up................. **62**, 489
on/off G703............................. 489
saving/loading G702.............. 489
Contour form................................ 100
Contour milling G840.................. 463
Contour programming................ 276
Contour repeat cycle G83........... 559
Contour thread G38.................... 410
Control graphic TURN PLUS....... 703
Controlled parting
with lag error monitoring G917.. 554
Converting and mirroring G30.... 547
Convert lengths G927................. 494
Coolant
TURN PLUS machining information.............................. 707
Correction
additive G149.......................... 352
additive G149-Geo.................. 308
Correction of cut (changing the cutter compensation) G148....................................... 351, 351
Countersinking G72.................... 425
Creating a job................................ 95
Cut-off cycle G859...................... 413
Cutter radius compensation....... 341
Cutting limit.................................. 628
Cutting speed, constant Gx96.... 339
Cycle end / simple contour G80 390

D

Data input..................................... 533
Data output................................... 533
Date... 517
Datum, C-axis object center G773.. 587
Datum, single point w/ C axis G772.. 585
Datum shift
absolute G59........................... 346
activating G980....................... 500
activating tool length G981.... 500
additive G56............................ 345
C axis G152............................. 440
deactivating G920................... 493
deactivating tool length G921.. 493
in variable G902...................... 491
overview................................... 343
relative G51.............................. 344
Datum shifts G53/G54/G55....... 345
Deburring G840........................... 470
Deep drilling G74......................... 428
Departure, smart.Turn................. 104
Determining cutting data, TURN PLUS... 707

Dialog in subprograms................ 543
DIN PLUS
    converting and mirroring G30.. 547
    example of full-surface machining with counter spindle...................................... 571
    example of full-surface machining with single spindle..... 573
DIN PLUS workpiece blank definition... 287
Direct program-run continuation G999.. 501
DNC information G941................ 498
Drilling
    bore milling G75..................... 431
    Deep drilling G74.................... 428
    TURN PLUS............................. 706
Drilling and boring
    overview.................................. 421
Drilling pattern
    circular on front face G745.... 434
    circular on lateral surface G746.. 437
    linear on front face G743....... 433
    linear on lateral surface G744.. 436
Drilling unit
    bore milling in ICP, C axis....... 154
    bore milling in ICP, C axis, on front face.................................. 154
    bore milling in ICP, C axis on lateral surface......................... 156
    bore milling in ICP, Y axis....... 239
    bore milling in ICP, Y axis on front face.. 239
    bore milling in ICP, Y axis on lateral surface......................... 241
    Boring, countersinking............ 130
    Boring, countersinking in ICP, Y axis.. 238
    Boring in ICP, countersinking, C axis.. 153
    Centric..................................... 127
    centric tapping........................ 128
    Circular drilling pattern on front face.. 135
    Circular drilling pattern on lateral surface...................................... 145
    Circular tapping pattern on front face.. 139
    Circular tapping pattern on lateral surface...................................... 149
    deburring in ICP, C axis, on front face.. 155
    deburring in ICP, C axis on lateral surface...................................... 157
    deburring in ICP, Y axis on front face.. 240
    deburring in ICP, Y axis on lateral surface...................................... 242
    Drilling in ICP, C axis.............. 151
    Drilling in ICP, Y axis.............. 236
    Linear drilling pattern on front face.. 133
    Linear drilling pattern on lateral surface...................................... 143
    Linear tapping pattern on front face.. 138
    Linear tapping pattern on lateral surface...................................... 148
    predrilling, contour milling in ICP, XY plane.................................. 243
    predrilling, contour milling in ICP, YZ plane.................................. 246
    predrilling, contour milling in ICP on front face........................... 162
    predrilling, contour milling in ICP on lateral surface.................... 168
    predrilling, contour milling of figures...................................... 158
    predrilling, contour milling of figures on lateral surface....... 164
    predrilling, pocket milling in ICP, XY plane.................................. 244
    predrilling, pocket milling in ICP, YZ plane.................................. 247
    predrilling, pocket milling in ICP on front face........................... 163
    predrilling, pocket milling in ICP on lateral surface.................... 169
    predrilling, pocket milling of figures...................................... 160
    predrilling, pocket milling of figures on lateral surface....... 166
    Single hole on front face........ 131
    Single hole on lateral surface 141
    Single tap hole on front face. 137
    Single tap hole on lateral surface...................................... 147
    Tapping in ICP, C axis............ 152
    Tapping in ICP, Y axis............ 237
Dwell time G4.............................. 490

E

Eccentric transition G726........... 507
Eccentric turning G725................ 506
Elements of a DIN program.......... 66
Engraving
    character set........................... 484
    XY plane G803........................ 666
    YZ plane G804........................ 667
Engraving on front face G801.... 487
Engraving on lateral surface G802... 488
Example
    full-surface machining with counter spindle........................ 571
    full-surface machining with single spindle.......................... 573
    machining with the Y axis...... 675
    measuring and compensating workpieces.............................. 625
    programming a machining cycle.. 282
    subprogram with contour repetitions................................ 565
    TURN PLUS............................ 711
Expert program............................ 283

F

Face roughing, simple G82......... 558
Face roughing G820.................. 359
Feed
    constant G94.......................... 338
    interrupted G64....................... 337
    per revolution G95-Geo.......... 307
    per revolution Gx95................ 339
    per tooth Gx93........................ 338
Feed per minute G94.................. 338
Feed per revolution G95............. 339
Feed rate...................................... 336
Feed rate override 100 % G908.. 492
Feed rate reduction G38-Geo..... 305
Figure milling cycle
    front face G793....................... 455
    lateral surface G794............... 457
File organization
    smart.Turn operating mode..... 78
Fill variable memory G904.......... 492
Find hole
    in C front face G780................ 608
    in C lateral surface G781........ 610
Find stud
    in C face G782........................ 612
    in C lateral surface G783........ 614
Fingerprint.................................... 511
Finishing unit
    ICP.. 171
    longitudinal, with direct contour input.. 173
    measuring cut......................... 179
    transverse, with direct contour input.. 175
    undercut, types E, F, DIN76.... 177
Fixed stop G916.......................... 553
Fluctuating spindle speed G924 493
Force reduction G925.................. 502
Form
    AppDep.................................... 104
    contour..................................... 100
    global....................................... 103

overview.. 99
tool.. 99
tool Ext.. 105
Front face contour......................... 315
Front face machining..................... 442
Full circle
front face G304-Geo.............. 318
lateral surface G314-Geo....... 326
XY plane G374-Geo................ 633
YZ plane G384-Geo................ 641
Full-surface machining
in DIN PLUS............................. 569
with TURN PLUS..................... 716
Function TURN PLUS................. 682

G

G17 XY plane................................ 647
G18 XZ plane, turning.................. 647
G19 YZ plane................................ 647
G code for contour description.. 743
additive correction G149........ 308
blank contour G67.................. 490
blockwise oversize G52.......... 306
bore hole (centric) G49........... 303
bore hole on front/rear face G300.. 317
cast part G21.......................... 287
centric polygon in XY plane G477.. 637
centric polygon in YZ plane G487.. 646
circular arc in front/rear face contour G102.......................... 316
circular arc in front/rear face contour G103.......................... 316
circular arc in lateral surface contour G112.......................... 324
circular arc in lateral surface contour G113.......................... 324
circular arc in XY plane G172 630
circular arc in XY plane G173 630
circular arc in YZ plane G182 639
circular arc in YZ plane G183 639
circular arc of turning contour G12.. 291
circular arc of turning contour G13.. 291
circular arc of turning contour G2.. 290
circular arc of turning contour G3.. 290
circular pattern in XY plane G472.. 636
circular pattern in YZ plane G482.. 644
circular pattern on lateral surface G412.. 329
circular slot in XY plane G372.. 632
circular slot in XY plane G373.. 632
circular slot in YZ plane G382.. 641
circular slot in YZ plane G383.. 641
circular slot on front/rear face G302.. 318
circular slot on front/rear face G303.. 318
circular slot on lateral surface G312.. 326
circular slot on lateral surface G313.. 326
DataMatrix for lateral surface G415.. 330
DataMatrix on front face G405.. 322
DataMatrix XY plane G475..... 637
DataMatrix YZ plane G485..... 645
end of pocket/island G309.... 309
feed per revolution G95.......... 307
feed rate reduction G38.......... 305
full circle in XY plane G374.... 633
full circle in YZ plane G384.... 641
full circle on front/rear face G304.. 318
full circle on lateral surface G314.. 326
G20 chuck part bar/tube........ 287
G402 circular pattern on front/ rear face.................................. 321
hole in XY plane G370........... 631
hole in YZ plane G380............ 640
hole on lateral surface G310. 325
linear patter in XY plane G471.. 635
linear pattern in YZ plane G481.. 643
linear pattern on front/rear face G401.. 320
linear pattern on lateral surface G411.. 328
linear slot in XY plane G371... 632
linear slot in YZ plane G381... 640
linear slot on front/rear face G301.. 317
linear slot on lateral surface G311.. 325
line segment in a lateral surface contour G111.......................... 323
line segment in a turning contour G1.. 289
line segment in front/rear face contour G101.......................... 315
line segment in XY plane G171.. 629
line segment in YZ plane G181.. 638
polygon in XY plane G377...... 634
polygon in YZ plane G387...... 642
polygon on front/rear face G307.. 320
polygon on lateral surface G317.. 328
recess (general) G23.............. 295
recess (standard) G22........... 293
rectangle in XY plane G375.... 633
rectangle in YZ plane G385.... 642
rectangle on front/rear face G305.. 319
rectangle on lateral surface G315.. 327
single surface in XY plane G376.. 634
single surface in YZ plane G386.. 645
starting point of contour in XY plane G170.............................. 629
starting point of contour in YZ plane G180.............................. 638
starting point of front/rear face contour G100.......................... 315
starting point of lateral surface contour G110.......................... 323
start of pocket/island G308... 309
superimposed elements G39 305
surface roughness G10.......... 304
text for front face, C axis, G306.. 319
text for front face Y G378...... 635
text for lateral surface, C axis, G316.. 327
text for lateral surface Y G388.. 643
thread (general) G37.............. 301
thread (standard) G34........... 300
thread with undercut G24...... 296
Undercut contour G25... 297, 555
G code for machining................. 746
absolute datum shift G59...... 346
activating datum shift, tool length G981.. 500
activating datum shift G980.. 500
actual values in variable G901.. 491
additive correction G149....... 352
additive datum shift G56........ 345
approach tool change point in Y axis G14................................... 650
area milling, finishing in Y axis G842.. 656
area milling, roughing in Y axis G841.. 655
area milling on front face

G797.. 460
axis-parallel oversize G57...... 347
bore milling G75..................... 431
boring, countersinking G72.... 425
centric polygon – finishing in Y axis G844................................. 658
centric polygon – roughing in Y axis G843................................. 657
chucking equipment G65......... 84
circular arc G12...................... 335
circular arc G13...................... 335
circular arc G2......................... 334
circular arc G3......................... 334
circular arc in Y axis G12........ 654
circular arc in Y axis G13........ 654
circular arc in Y axis G2.......... 653
circular arc in Y axis G3.......... 653
circular arc on front/rear face G102.. 445
circular arc on front/rear face G103.. 445
circular arc on lateral surface G112.. 449
circular arc on lateral surface G113.. 449
circular pattern on front face G745.. 434
circular pattern on lateral surface G746.. 437
circular slot on front face G302.. 391
circular slot on front face G303.. 391
circular slot on lateral surface G312.. 394
circular slot on lateral surface G313.. 394
clamping G65.......................... 490
compensation for helical teeth G728.. 510
compensation of left-hand tool tip G151.................................... 353
compensation of right-hand tool tip G150.................................... 353
component monitoring G939 513
constant cutting speed G96.. 339
constant feed G94.................. 338
contour and figure milling cycle on front face G793................. 455
contour and figure milling cycle on lateral surface G794.......... 457
contour cycle bidirectional G835 (contour-parallel with neutral tool)... 364
contour finishing G890........... 374
contour follow-up G703.......... 489
contour milling G840............. 463
contour-parallel oversize G58..... 348
contour-parallel roughing G830.... 362
contour thread G38................ 410
converting and mirroring G30... 547
correction of cut G148 (changing the cutter compensation)...... 351
cut-off cycle G859.................. 413
cycle end / simple contour G80.. 390
datum shift G51...................... 344
datum shift in C axis G152..... 440
datum shift in variable G902. 491
datum shifts G53/G54/G55... 345
deactivating datum shift G920.. 493
deactivating datum shifts, tool length G921............................. 493
Deep drilling G74.................... 428
define monitoring zone G995 500
definition of tool change point G140.. 332
direct program-run continuation G999.. 501
drilling cycle G71.................... 423
dwell time G4.......................... 490
eccentric polygon on lateral surface G317........................... 395
eccentric transition G726....... 507
eccentric turning G725........... 506
engraving in XY plane G803... 666
engraving in YZ plane, G804.. 667
engraving on front face G801.. 487
engraving on lateral surface G802.. 488
face roughing G820............... 359
feed per revolution G95.......... 339
feed per tooth G93................. 338
feed rate override 100 % G908.. 492
fill variable memory G904...... 492
fluctuating spindle speed G924.. 493
force reduction G925............. 502
full circle on front face G304. 392
full circle on lateral surface G314.. 394
G73 tapping............................. 426
G799 thread milling axial....... 439
HDT mode G931...................... 504
helical slot milling G798......... 462
hobbing G808.......................... 674
information to DNC G941...... 498
interpreter stop G909............. 492
interrupted feed G64.............. 337
lag error in variable G903....... 492
lateral surface, linear path G111.. 447
lift off G977.............................. 499
linear movement G1............... 333
linear movement in Y axis G1 652
linear pattern on front face G743.. 433
linear pattern on lateral surface G744.. 436
linear segment on front/rear face G101.. 443
linear slot on front face G301 391
linear slot on front face G791 453
linear slot on lateral surface G311.. 393
linear slot on lateral surface G792.. 454
line with chamfer G88............ 562
line with radius G87............... 562
longitudinal roughing G810... 356
measuring cut G809.............. 389
metric ISO thread G35........... 407
misalignment compensation G976.. 498
non-circular X G727............... 509
offset C angle G905............... 552
one-sided synchronization G62.... 549
pocket milling, finishing G846.. 478
Pocket milling, roughing G845.. 472
pocket milling – finishing in Y axis G846................................. 664
pocket milling – roughing in Y axis G845................................. 659
polygon on front/rear face G307.. 393
precision stop OFF G8............ 491
precision stop ON G7............. 490
rapid traverse, referencing the machine datum G701.... 331, 651
rapid traverse, referencing the workpiece datum G0.............. 331
rapid traverse in Y axis G0..... 650
rapid traverse on front/rear face G100.. 442
rapid traverse on lateral surface G110.. 447
recessing cycle G870............. 373
recessing with respect to the contour G860.......................... 366
recess turning cycle G869..... 370
rectangle on front face G305 392
rectangle on lateral surface G315.. 395
reduce rapid traverse G48...... 336
reference diameter G120....... 440

repeating recessing cycle G740 ... 368
repeating recessing cycle G741 ... 368
safety clearance for milling G147 ... 349
safety clearance G47 ... 349
saving/loading contour follow-up G702 ... 489
setting synchronizing mark G162 ... 548
short path in C G154 ... 441
simple contour repeat cycle G83 ... 559
simple face roughing G82 ... 558
simple longitudinal multi-start thread G351 ... 564
simple longitudinal roughing G81 ... 557
simple longitudinal single-start thread G350 ... 563
simple recessing cycle G86 ... 561
simultaneous finishing G891 384
simultaneous roughing G895 378
single thread cycle G32 ... 403
sleeve monitoring G930 ... 503
speed limitation G26 ... 336
spindle override 100 % G919. 493
spindle speed G97 ... 340
spindle synchronization G720 ... 551
standardize C axis G153 ... 441
switch off oversize G50 ... 347
switch off protection zone G60 ... 491
switch off TRC/MCRC G40 ... 341
switch on TRC/MCRC G41 ... 342
switch on TRC/MCRC G42 ... 342
synchronous start of slides G63 ... 550
tapered API thread G352 ... 408
thread, single path G33 ... 405
thread milling in XY plane G800 ... 668
thread milling in YZ plane G806 ... 669
tilt working plane G16 ... 648, 649
tool change point G14 ... 332
traverse to a fixed stop G916 553
trochoidal contour milling G847 ... 480
trochoidal pocket milling G848 ... 482
type of load monitoring G996 ... 501
undercut cycle G85 ... 414
undercut DIN 509 E with cylinder machining G851 ... 415
undercut DIN 509 F with cylinder machining G852 ... 416
Undercut DIN 76 with cylinder machining G853 ... 417
undercut type H G857 ... 420
undercut type K G858 ... 420
undercut type U G856 ... 419
universal tapping cycle G31 ... 398
workpiece group G99 ... 548

G codes for machining
- Controlled parting G917 ... 554
- G36 Tapping ... 422

Geometry command ... 274

G function for machining
- Measure machine condition G238 ... 511
- Skiving G807 ... 669

Global form ... 103

H

Handwheel superimposition ... 184, 396
HDT mode G931 ... 504
Helical slot milling G798 ... 462
Helical teeth G728 ... 510
Help commands for contour description ... 304
Help graphics for subprogram call ... 544
High Dynamic Turning ... 724
Hobbing G808 ... 674
Hole
- lateral surface G310-Geo ... 325
- XY plane G370-Geo ... 631
- YZ plane G380-Geo ... 640

I

IF.. Program branching ... 535
Inches
- conversion ... 496
- programming ... 65

Index of a parameter element, determining ... 529
Information to DNC G941 ... 498
In-process measurement ... 623
INPUT ... 534
Input of # variables ... 534
Inside contour
- TURN PLUS machining information ... 707

Interpreter stop G909 ... 492
Interrupted feed G64 ... 337
Island (DIN PLUS) ... 309
ISO program conversion ... 284

L

Lag error in variable G903 ... 492
Lateral surface
- contour ... 323
- machining ... 447
- SURFACE Y section ... 87

L call ... 542
Lift off after NC stop G977 ... 499
Linear and circular movements in Y axis ... 652
Linear axis ... 65
Linear movement ... 333, 652
Linear movement G1 ... 333
Linear path
- lateral surface G111 ... 447

Linear segment
- front face G101 ... 443

Linear slot
- front face G301-Geo ... 317
- front face G791 ... 453
- lateral surface G311-Geo ... 325
- lateral surface G792 ... 454
- XY plane G371-Geo ... 632
- YZ plane G381-Geo ... 640

Line segment
- front face contour G101-Geo 315
- lateral surface contour G111-Geo ... 323
- XY plane G171-Geo ... 629
- YZ plane G181-Geo ... 638

Line segment in a turning contour G1–Geo ... 289
Load monitoring G996 ... 501
Longitudinal roughing, simple G81 ... 557
Longitudinal roughing G810 ... 356

M

Machine command ... 546
Machining attributes for form elements ... 288
Machining command ... 274
Machining cycle ... 282
Machining information TURN PLUS ... 704
Machining on tapered contours. 498
Machining sequence AWG
- editing ... 686
- list of machining ... 688
- managing ... 686

Machining sequence in AWG
- general ... 685

M command ... 545
- machine command ... 546
- program-run control ... 545
- synchronous function M97 ... 550

Measure machine condition
- component monitoring G939 513
- fingerprint G238 ... 511

Measuring
- actual value determination

G912.......... 624
angular.......... 620
circular.......... 616
ending G913.......... 624
switch on G910.......... 623
with touch probe cycle.......... 601
Measuring cut G809.......... 389
Measuring path monitoring
activating G911.......... 624
deactivating G914.......... 624
Measuring the machine condition
overview.......... 511
Menu item
Configuration.......... 74
Extras.......... 76
geometry.......... 286
Go to.......... 73
Graphics.......... 77
ICP.......... 73
machining.......... 286
Misc.......... 75
program head.......... 72
program management.......... 72
units.......... 98
Menu structure
smart.Turn operating mode.......... 68
Metric ISO thread G35.......... 407
Milline pattern
circular on front face G745.......... 434
Milling
area milling on front face G797.......... 460
contour and figure milling cycle on front face G793.......... 455
contour and figure milling cycle on lateral surface G794.......... 457
Contour milling G840.......... 466
fundamentals G840.......... 463
helical slot G798.......... 462
linear slot on front face G791 453
linear slot on lateral surface G792.......... 454
pocket milling, finishing G846.......... 478
Pocket milling, roughing G845.......... 472
Pocket milling G845.......... 474
trochoidal contour milling G848.......... 480
trochoidal pocket milling G848.......... 482
Milling cycle
Y axis.......... 655
Milling cycles
overview.......... 451
Milling pattern
circular on lateral surface G746.......... 437
linear on front face G743.......... 433
linear on lateral surface G744.......... 436
Milling unit
centric polygon, XY plane.......... 254
centric polygon, YZ plane.......... 265
circular slot pattern on front face.......... 194
circular slot pattern on lateral surface.......... 214
contour milling, figures on lateral surface.......... 216
contour milling in ICP, XY plane.......... 249
contour milling in ICP, YZ plane.......... 261
contour milling in ICP on front face.......... 202
contour milling in ICP on lateral surface.......... 221
contour milling of figures on front face.......... 197
deburring, XY plane.......... 252
deburring, YZ plane.......... 263
deburring on front face.......... 206
deburring on lateral surface.. 223
engraving, XY plane.......... 255
engraving, YZ plane.......... 266
engraving on front face.......... 201
engraving on lateral surface.. 220
face milling.......... 195
face milling in ICP.......... 207
helical slot.......... 215
linear slot on front face.......... 192
linear slot on lateral surface.. 212
linear slot pattern on front face.......... 193
linear slot pattern on lateral surface.......... 213
pocket milling in ICP, XY plane.......... 250
pocket milling in ICP, YZ plane.......... 262
pocket milling in ICP on front face.......... 204
pocket milling in ICP on lateral surface.......... 222
pocket milling of figures on front face.......... 199
pocket milling of figures on lateral surface.......... 218
single surface, XY plane.......... 253
single surface, YZ plane.......... 264
thread milling.......... 196
thread milling, XY plane.......... 256
thread milling, YZ plane.......... 267
trochoidal pocket milling.......... 259
Milling unit, front face
trochoidal contour milling, C axis.......... 208
trochoidal contour milling, Y axis.......... 257
trochoidal pocket milling, C axis.......... 210
Milling unit, lateral surface
trochoidal contour milling, C axis.......... 224
trochoidal contour milling, Y axis.......... 268
trochoidal pocket milling, C axis.......... 226
trochoidal pocket milling, Y axis.......... 270
Misalignment compensation G788... 622
Misalignment compensation G976... 498
Monitoring zone, defining G995. 500
Multipoint tool.......... 93
for the B axis.......... 723

N

NC information, reading.......... 524, 525
NC program interpretation.......... 283
New NC program, create.......... 67
Non-circular X G727.......... 509

O

One-side synchronization G62... 549
Output of # variables.......... 534
Output window for variables.......... 533
Oversize.......... 347
axis-parallel G57.......... 347
contour-parallel (equidistant) G58.......... 348
G52-Geo.......... 306
switching off G50.......... 347
Overview form.......... 99

P

PARA
configuration data, reading.......... 528
determining the index of a parameter element.......... 529
Parallel editing.......... 69
Pattern
circular in XY plane G472-Geo.......... 636
circular in YZ plane G482-Geo.......... 644
circular on front face G402-Geo.......... 321
circular on front face G745.......... 434
circular on lateral surface G412-Geo.......... 329
circular on lateral surface

G746 ... 437
DataMatrix for lateral surface G415-Geo ... 330
DataMatrix on front face G405-Geo ... 322
DataMatrix XY plane G475-Geo ... 637
DataMatrix YZ plane G485-Geo ... 645
linear in XY plane G471-Geo .. 635
linear in YZ plane G481-Geo .. 643
linear on front face G401-Geo ... 320
linear on front face G743 ... 433
linear on lateral surface G411-Geo ... 328
linear on lateral surface G744 ... 436

Pocket milling
- finishing G846 ... 478
- Roughing G845 ... 472

Polygon
- lateral surface G317-Geo ... 328
- on front/rear face G307-Geo. 320
- XY plane G377-Geo ... 634
- YZ plane G387-Geo ... 642

Position of milling contour ... 309

Position of milling contour in Y axis ... 628

Precision stop
- blockwise G9 ... 491
- OFF G8 ... 491
- ON G7 ... 490

PRINT ... 534

Probing
- C axis G765 ... 603
- in two axes G766 ... 604
- in two axes G767 ... 605
- in two axes G768 ... 606
- in two axes G769 ... 607
- paraxial G764 ... 601

Program
- create ... 67

Program branch
- IF ... 535
- SWITCH ... 540
- WHILE ... 538

Program interpretation ... 283

Program list ... 95

Programming
- in ISO mode ... 274
- with smart.Turn ... 98

Programming variables ... 514

Program section code ... 80, 742

Program templates ... 575

R

Radius G87 ... 562

Rapid traverse
- front face G100 ... 442
- lateral surface G110 ... 447
- reducing G48 ... 336
- referencing the machine datum G701 ... 331, 651
- referencing the workpiece datum G0 ... 331
- Y axis G0 ... 650

Read
- interpolation information G904 ... 492

Reading
- configuration data ... 528
- current NC information ... 524
- general NC information ... 525

Reading diagnostic bits ... 523

Reading tool data ... 520

Rear face contour ... 315

Rear-face machining
- example of full-surface machining with counter spindle ... 571
- example of full-surface machining with single spindle ... 573

Recess
- general G23–Geo ... 295
- standard G22–Geo ... 293

Recessing
- recessing cycle G870 ... 373
- recessing G860 ... 366
- repeating G740 ... 368
- repeating G741 ... 368
- repeating recessing cycle G740 ... 368
- repeating recessing cycle G741 ... 368

Recessing cycle G870 ... 373

Recessing G86 ... 561

Recessing G860 ... 366

Recessing unit
- contour recessing in ICP ... 119
- contour recessing with direct contour input ... 122
- parting ... 124
- recess cycle ... 126
- recessing in ICP ... 126
- recess turning in ICP ... 120
- recess turning with direct contour input ... 123
- undercutting, type H, K, U ... 125

Recess turning cycle G869 ... 370

Rechucking the workpiece
- TURN PLUS ... 716

Rectangle
- front face G305-Geo ... 319
- lateral surface G315-Geo ... 327
- XY plane G375-Geo ... 633
- YZ plane G385-Geo ... 642

Reference diameter G120 ... 440

Reference plane
- SURFACE Y section ... 87

Replacement tool ... 94

Resonance, reducing ... 493

Rotary axis ... 65

Roughing
- contour cycle, bidirectional G835 (contour-parallel with neutral tool) ... 364
- contour-parallel G830 ... 362
- face G820 ... 359
- longitudinal G810 ... 356

Roughing unit
- bidirectional in ICP ... 111
- contour-parallel in ICP ... 110
- longitudinal, with direct contour input ... 112
- longitudinal in ICP ... 106
- simultaneous roughing G895 115
- transverse, with direct contour input ... 114
- transverse in ICP ... 108

Running NC blocks with NC start in Program run, Single block mode G999 ... 501

S

Safety clearance for milling G147 ... 349

Safety clearance for turning G47 ... 349

Screen layout
- smart.Turn operating mode ... 69

Search cycle ... 608

Section ... 80
- AUXILIARY BLANK ... 86
- AUXILIARY CONTOUR ... 86
- BLANK ... 86
- CHUCKING EQUIPMENT ... 84
- CONTOUR GROUP ... 85
- END ... 88
- FINISHED PART ... 86
- FRONT ... 86
- FRONT Y ... 86
- LATERAL ... 86
- MACHINING ... 88
- MAGAZINE ... 85
- MANUAL TOOL ... 85
- PROGRAM HEAD ... 82
- REAR SIDE ... 86
- REAR SIDE Y ... 86

- RETURN ... 88
- SUBROUTINE ... 88
- SURFACE Y ... 87
- TURRET ... 85
- VAR ... 89

Separation point G44 ... 306
Setting synchronizing mark G162 ... 548
Shaft machining TURN PLUS ... 709
shaft speed ... 336
Short path in C G154 ... 441
Simultaneous contour finishing G891 ... 384
Simultaneous finishing unit ... 180
Single-point measurement, datum G771 ... 583
Single-point measurement for tool compensation G770 ... 581
Single surface
- XY plane G376-Geo ... 634
- YZ plane G386-Geo ... 645

Single thread cycle G32 ... 403
Skip level ... 541
Skiving (G807) ... 669
Sleeve monitoring G930 ... 503
Slot
- circular on front face G302-/G303-Geo ... 318
- circular on lateral surface G312-/G313-Geo ... 326
- linear on front face G301-Geo ... 317
- linear on front face G791 ... 453
- linear on lateral surface G311-Geo ... 325
- linear on lateral surface G792 ... 454

smart.Turn ... 62
- editor ... 68
- file organization ... 78
- menu structure ... 68
- screen layout ... 69
- unit ... 98

Spec. unit
- C axis OFF ... 230
- C axis ON ... 229

Special output window ... 533
Spec unit
- program beginning ... 228
- program end ... 232
- program section repeat ... 231
- subprogram call ... 230
- Tilt plane ... 233

Speed, constant Gx97 ... 340
Speed limitation G26 ... 336
Spindle override 100 % G919 ... 493
Spindle synchronization G720 ... 551
Standardize C axis G153 ... 441
Starting point
- contour in YZ plane G180-Geo ... 638
- front face contour G100-Geo 315
- lateral surface contour G110-Geo ... 323
- turning contour G0–Geo ... 288

Starting point of contour in XY plane G170-Geo ... 629
Start of pocket/island G308-Geo ... 309
Structured NC program ... 63
Submode AWG ... 683
Subprogram
- call ... 542
- dialog in SP call ... 543
- fundamentals ... 283
- help graphics for SP call ... 544

Surface roughness G10–Geo ... 304
SWITCH..CASE program branch 540
Switch off MCRC G40 ... 341
Switch off protection zone G60. 491
Switch off TRC G40 ... 341
Switch on MCRC G41/G42 ... 342
Switch on TRC G41/G42 ... 342
Synchronous function M97 ... 550
Synchronous start of slides G63 ... 550

T

Tapered API thread G352 ... 408
Tapping ... 426
- universal G31 ... 398

Tapping G36 – Single path ... 422
T command ... 350
- fundamentals ... 91

TCPM ... 494, **722**
Text
- front face, C axis, G306-Geo.. 319
- front face Y G378-Geo ... 635
- lateral surface, C axis, G316-Geo ... 327
- lateral surface Y G388-Geo ... 643

Thread
- general G37–Geo ... 301
- metric ISO G35 ... 407
- simple longitudinal multi-start thread G351 ... 564
- simple longitudinal single-start thread G350 ... 563
- single G32 ... 403
- single path G33 ... 405
- standard G34–Geo ... 300
- tapered API G352 ... 408

Thread cycles
- overview ... 396

Thread milling
- axial G799 ... 439
- XY plane G800 ... 668
- YZ plane G806 ... 669

Thread overrun ... 396
Thread run-in ... 396
Thread runout ... 396
Thread unit
- API thread ... 189
- direct input ... 186
- ICP ... 187
- tapered thread ... 190

Tilted working plane ... 722
Time ... 517
Tool
- editing of entry ... 93
- inserting – T ... 350
- multipoint tool ... 93
- positioning ... 331
- positioning in Y axis ... 650
- replacement tool ... 94

Tool change point
- approaching G14 ... 332
- definition G140 ... 332

Tool command ... 350
Tool Ext form ... 105
Tool form ... 99
Tool programming ... 91
Tool selection
- TURN PLUS ... 704

Tool-tip radius compensation ... 341
Touch probe cycle
- angular measurement ... 620
- automatic mode ... 579
- circular measurement ... 616
- general ... 578, 578
- in-process measurement ... 623
- search cycle ... 608
- single-point measurement ... 581
- two-point measurement ... 589

Tree view ... 71
Turning contour
- basic element ... 288
- form element ... 293

Turning cycle
- contour-based ... 354

TURN PLUS ... 682
- AWG working plan generation 683
- control graphic ... 703
- cutting data ... 707
- example ... 711
- full-surface machining ... 716
- hollowing ... 705
- inside contour ... 707
- list of machining ... 688
- machining information ... 704
- machining sequence ... 685
- machining sequence, editing. 686

recessing.................................. 706
rechucking the workpiece...... 716
shaft machining...................... 709
tool selection........................... 704
turret assignment list............. 704
Turret
setting up a turret list................ 91
TURN PLUS turret assignment list.. 704
Two-point measurement
G17 G777................................. 593
G18 longitudinal G776........... 591
G18 transverse G775............. 589
G19 G778................................. 595

U

Undercut
cycle.. 414
DIN 509 E................................. 297
DIN 509 E with cylinder machining G851..................... 415
DIN 509 F................................. 298
DIN 509 F with cylinder machining G852..................... 416
DIN 76...................................... 298
DIN 76 with cylinder machining G853.. 417
type H...................................... 299
type H G857............................ 420
type K...................................... 299
type K G858............................ 420
type U...................................... 297
type U G856............................ 419
Undercut contour G25–Geo....... 297
Undercut cycle G85..................... 414
Unit.. 98
Unit of measure............................. 65
Unit – thread
overview.................................. 184

V

Variable
automatically calculating G940.... 496
expanded syntax..................... 529
fundamentals.......................... 514
Variables
types... 515
Vibration, reducing...................... 493

W

WHILE... 538
WINDOW...................................... 533
Working plane
tilting G16........................ 648, 649
Workpiece group G99.................. 548
Workpiece transfer
controlled parting with lag error
monitoring G917..................... 554

X

XY plane G17 on front or rear face.. 647
XZ plane G18................................ 647

Y

Y axis
approach tool change point G14.. 650
area milling, finishing G842... 656
area milling, roughing G841... 655
centric polygon – finishing G844.. 658
centric polygon – roughing G843.. 657
circular arc G12, G13............. 654
circular arc G2, G3.................. 653
linear movement G1............... 652
pocket milling – finishing G846.. 664
pocket milling – roughing G845.. 659
rapid traverse G0.................... 650
tool positioning....................... 650
Y-axis contour
fundamentals.......................... 628
YZ plane G19, lateral view / lateral surface.. 647

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 8669 31-0
FAX +49 8669 32-5061
info@heidenhain.de

Technical support FAX +49 8669 32-1000
Measuring systems ☏ +49 8669 31-3104
service.ms-support@heidenhain.de
NC support ☏ +49 8669 31-3101
service.nc-support@heidenhain.de
NC programming ☏ +49 8669 31-3103
service.nc-pgm@heidenhain.de
PLC programming ☏ +49 8669 31-3102
service.plc@heidenhain.de
APP programming ☏ +49 8669 31-3106
service.app@heidenhain.de

www.heidenhain.com

Original document
1118606-27 · Ver07 · SW17 ·10/2022 · H · Printed in Germany