

MTX 15VRS

Standard NC Cycles

Project Planning Manual
R911394940

Edition 03



Title	MTX 15VRS Standard NC Cycles
Type of Documentation	Project Planning Manual
Document Typecode	DOK-MTX***-NC*CYC**V15-PR03-EN-P
Internal File Reference	RS-647a885780f857adc0a80286245f9d8e-3-en-US-3
Change record	Edition 03, 2021-09 Refer to tab. 1-1 "Change record" on page 1
Copyright	© Bosch Rexroth AG 2021 All rights reserved, also regarding any disposal, exploitation, reproduction, editing, distribution, as well as in the event of applications for industrial property rights.
Liability	The specified data is intended for product description purposes only and shall not be deemed to be a guaranteed characteristic unless expressly stipulated in the contract. All rights are reserved with respect to the content of this documentation and the availability of the product.
Editorial Department	Engineering automation systems solution integration HMI, NC controls and support, ReLa (TaDo/MePe)

Table of Contents

	Page
1 About this documentation.....	1
1.1 Validity of the documentation.....	1
1.2 Required and supplementing documentation MTX.....	2
1.2.1 Selection/compilation.....	2
1.2.2 Configuration.....	2
1.2.3 Commissioning.....	3
1.2.4 Operation.....	4
1.2.5 OEM Engineering.....	5
1.2.6 AddOns.....	5
1.3 Using safety instructions.....	5
1.3.1 Structure of the safety instructions.....	5
1.3.2 Explaining signal words and safety alert symbol.....	6
1.3.3 Symbols used.....	7
1.3.4 Explaining the signal alert symbol on the device.....	7
1.4 Names and abbreviations.....	7
1.5 Customer feedback.....	7
2 Important instructions on use.....	9
2.1 Intended use.....	9
2.1.1 Introduction.....	9
2.1.2 Areas of use and application.....	9
2.2 Unintended use.....	10
3 General information.....	11
3.1 Introduction.....	11
3.2 G-code assignment.....	11
3.3 General rules for modal cycle calls.....	12
3.4 User cycles.....	13
3.5 Fixed machine operator cycles.....	14
4 Overview on NC cycles.....	15
4.1 Modal G-codes for cycle calls.....	15
4.2 Non-modal G-codes for cycle calls.....	15
4.3 Specified end user cycles, local.....	17
4.4 Specified end user cycles, modal.....	18
4.5 G-codes for measuring cycles.....	18
4.6 Cycles without G-codes.....	19
4.7 Using RCB labels in cycles.....	20
4.8 Approach starting position in the homing program.....	23
5 Drilling cycles.....	27
5.1 General information.....	27
5.2 G81 Drilling.....	27

Table of Contents

	Page
5.3 G82 Countersinking.....	29
5.4 G83 Deep hole drilling.....	32
5.5 G84 Tapping.....	36
5.6 G85 Reaming.....	39
5.7 G86 Fine boring.....	42
5.8 G87 Back boring.....	45
5.9 G88 Gun drilling.....	48
5.10 G89 Tapping with chip breaking.....	52
6 Hole patterns.....	57
6.1 General information.....	57
6.2 G111 Hole circle, rotary axis.....	57
6.3 G112 Hole circle, main spindle.....	59
6.4 G113 Hole circle, plane.....	63
6.5 G114 Hole pattern, line.....	68
6.6 G115 Hole pattern, matrix.....	72
6.7 G116 Hole pattern, frame.....	76
7 Milling cycles.....	81
7.1 General information.....	81
7.2 G261 Thread milling.....	81
7.3 G262 Thread milling.....	87
7.4 G263 Groove circular milling.....	91
7.5 G266 Face milling, rectangle.....	95
7.6 G267 Face milling, circle.....	100
7.7 G271 Rectangular pocket milling.....	105
7.8 G272 Circular pocket milling.....	109
7.9 G274 Straight slot milling.....	112
7.10 G275 Curved slot milling.....	117
7.11 G276 Rectangular pin milling.....	124
7.12 G277 Circular pin milling.....	130
7.13 G279 Land straight milling.....	134
7.14 G280 Land curved milling.....	139
8 Milling cycles (contour-dependent).....	147
8.1 General notes.....	147
8.2 Defining the contour.....	147
8.3 G273 User-defined contour pocket milling.....	147
8.4 G278 User-defined contour pivot milling.....	156
8.5 G290 Path milling.....	162
8.6 Milling contour pocket with islands xycle.....	168
8.6.1 General information.....	168
8.6.2 G288 Defining island contour.....	168
8.6.3 G289 Defining starting points.....	170
8.6.4 G281 Milling contour pocket with islands.....	172

	Page
9 Engraving.....	181
9.1 General information.....	181
9.2 G201 Linear text engraving.....	185
9.3 G202 Circular text engraving.....	189
10 Turning cycles.....	193
10.1 G160 Longitudinal turning.....	193
10.2 G161 Face turning.....	195
10.3 G162 Tapered turning.....	198
10.4 G165 Thread single step.....	200
10.5 G166 Thread sequence.....	203
10.6 G167 Thread turning.....	209
10.7 G168 Face thread.....	214
10.8 G169 Tapered thread.....	220
10.9 G191 Rounded groove turning.....	226
10.10 G192 Rounded groove turning.....	235
10.11 G196 Thread undercut.....	240
10.12 G197 Undercut Turning.....	245
11 Turning cycles (contour-dependent).....	251
11.1 General notes.....	251
11.2 Defining the Contour.....	251
11.3 G170 Contour finishing.....	252
11.4 G171 Contour roughing, longitudinal.....	254
11.5 G172 Contour roughing, plane.....	259
11.6 G173 Contour roughing, parallel to contour.....	263
11.7 G174 Contour grooving.....	266
12 Contour drafts.....	275
12.1 Contour programming G31.....	275
12.1.1 General information.....	275
12.1.2 Contour programming G31 for milling.....	276
12.1.3 Contour programming G31 for turning.....	288
13 G230 Swiveling	305
14 Technology cycles.....	319
14.1 General information.....	319
14.2 G501 TFS Cycle turning.....	319
14.3 G511 TFS Cycle milling.....	324
14.4 G591 HSC settings.....	328
14.5 G592 Approximated free form surface milling.....	331
14.6 STP static tool placement	333
14.7 SETBQ Set/reset tool/position status bit.....	334

Table of Contents

	Page
14.8 GETBQ Read tool/position status bit.....	337
14.9 INT_TO_STR Format conversion Integer to string.....	339
14.10 REAL_TO_STR Format conversion Real to string.....	340
15 Punching cycles.....	341
15.1 General information.....	341
15.2 G213 Hole circle punching.....	342
15.3 G214 Hole pattern line punching.....	343
15.4 G215 Hole pattern matrix punching.....	346
15.5 G219 Multiple part punching.....	348
15.6 G224 Nibble line.....	352
15.7 G226 Nibble rectangular.....	354
16 Measuring cycles.....	357
17 TCH tool change.....	359
18 Programming instruction for simulation	361
18.1 STOCK/BLOCK Cuboid raw piece (milling)	361
18.2 STOCK/CYLINDER Cuboid raw piece (milling)	362
18.3 STOCK/BAR Bar section (turning).....	365
18.4 STOCK/POLYGON Multi-edge (turning).....	367
18.5 STOCK/SHEET Sheet (punching).....	369
18.6 STOCKIMP Import model.....	370
18.7 STOCK/REMOVE Remove workpieces/models.....	373
18.8 STOCK/RENEW Renew workpiece.....	375
18.9 STOCKSEL Select tool holder.....	376
18.10 STOCKADD Replace/add workpieces/models.....	377
19 Service and support.....	379
Index.....	381

1 About this documentation

Editions of this documentation

Edition	Release date	Note
01	2019-03	First edition
02	2020-10	chapter 4.2 "Non-modal G-codes for cycle calls" on page 15 : Note: G591 is not supported anymore (Defdb00208686) chapter 14.4 "G591 HSC settings" on page 328 : Note: G591 is not supported anymore (Defdb00208686)
03	2021-09	New: G628, G629 (PBI 307916)

Tab. 1-1: Change record

1.1 Validity of the documentation

This documentation describes the use of MTX standard NC cycles.

This documentation supports the user during the following phases:

- Engineering
- Commissioning

Overview on target groups and product phases

The following graphic refers to the bordered activities, product phases and target groups of the present documentation.

The target group "Programmer" can "parameterize", "program, configure and simulate in the product phase Engineering" using this documentation.

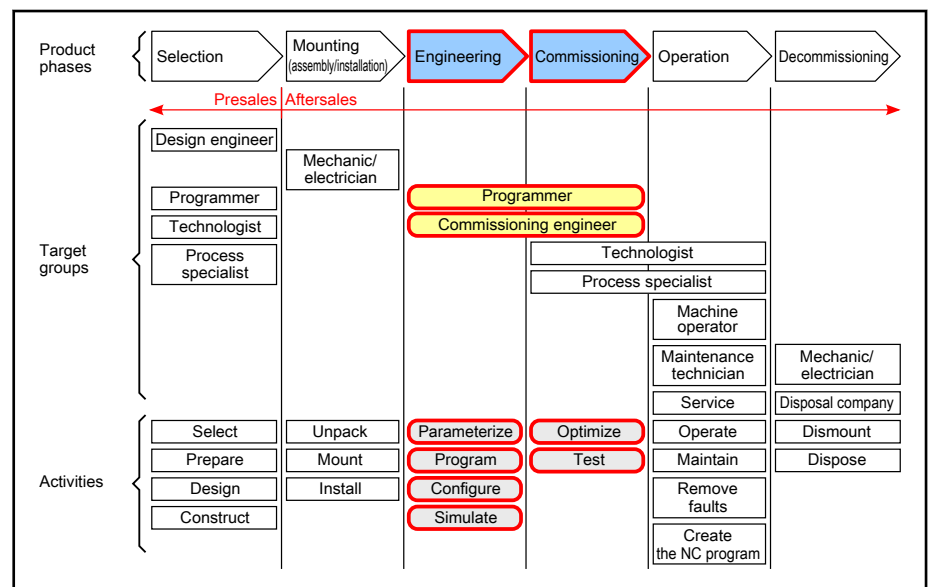


Fig. 1-1: Assigning this documentation to the target groups, product phases and target group activities

1.2 Required and supplementing documentation MTX

1.2.1 Selection/compilation

Documentation titles with type codes and part numbers

MTX 15VRS System Description DOK-MTX***-SYS*DES*V15-PRRS-EN-P, R911397742 This documentation describes the MTX control. It includes the designs, technical data, interfaces as well as the configuration of the control components.
MTX 15VRS SafeLogic System Overview DOK-MTX***-SL**SYS*V15-PRRS-EN-P, R911398637 This documentation describes the use of the safety control SafeLogic in the MTX.
IndraControl XM42 Controls DOK-CONTRL-XM4X*CTRL**-ITRS-EN-P, R911345566 This documentation describes the IndraControl XM42 controls.

RS Corresponding edition
Tab. 1-2: MTX documentation overview - Selecting/composing

1.2.2 Configuration

Documentation titles with type codes and part numbers

MTX 15VRS Machine Parameters DOK-MTX***-MA*PAR**V15-RERS-EN-P, R911400176 This documentation describes structure and the adjustment of available parameters of the MTX. It also includes the functions of the NC configurator and its operation.
MTX 15VRS PLC Interface DOK-MTX***-PLC*INT*V15-PRRS-EN-P, R911400172 This documentation describes the interface signals and the program function blocks for the integrated PLC.
MTX 15VRS Functional Description - Basics DOK-MTX***-NC*F*BA*V15-RERS-EN-P, R911401401 This documentation describes the basic functions of the MTX. The basic commissioning steps and the control functions are provided as description and handling instruction.
MTX 15VRS Functional Description - Extension DOK-MTX***-NC*F*EX*V15-RERS-EN-P, R911393316 This documentation describes the extended functions of the MTX. The basic commissioning steps and the control functions are provided as description and handling instruction.
MTX 15VRS Functional Description - Special Functions DOK-MTX***-NC*F*SP*V15-RERS-EN-P, R911393309 This documentation describes the special functions of the MTX. The basic commissioning steps and the control functions are provided as description and handling instruction.

MTX Free Form Surface Milling

DOK-MTX***-FREEFORM***-APRS-EN-P, R911341435

This documentation describes the free form surface milling process with the MTX control. CNC programs generated by a CAD/CAM system are used as basis for the entire process. Overview over the following topics: Description of the MTX or CNC function for freeform surface milling, strategy during the NC parameterization process, boundary conditions when generating CNC programs with CAM software.

MTX Conversion of MTX Projects

DOK-MTX***-PROCONV****-PRRS-EN-P, R911342484

This documentation provides support during the conversion of MTX 1.x projects to MTX 2G. This project planning manual provides information on how to convert a project and identifies potential difficulties during the conversion.

RS

Corresponding edition

Tab. 1-3:

MTX documentation overview - Configuring

1.2.3 Commissioning

Documentation titles with type codes and part numbers

MTX 15VRS Commissioning

DOK-MTX***-STARTUP*V15-CORS-EN-P, R911393281

This documentation describes the commissioning of the MTX control. Apart from a complete overview, commissioning and configuration of the axes and the user interface as well as the PLC data are described.

IndraWorks 15VRS Basic Libraries, IndraLogic 2G

DOK-IL*2G*-BASLIB**V15-LIRS-EN-P, R911398633

This documentation describes the system-comprehensive PLC libraries.

IndraWorks 15VRS Field Buses

DOK-IWORKS-FB*****V15-APRS-EN-P, R911393284

This documentation describes the field bus and local periphery connections supported by the MLC and MTX systems. The focus of this documentation lies in the configuration, parameterization, commissioning and diagnostics of different periphery connections. It is the basis for the online help.

IndraWorks 14VRS WinStudio 7.4

DOK-IWORKS-WINSTUD*V14-APRS-EN-P, R911341585

This "User Manual and Technical Reference Book" supports the user in achieving the best results with the "Rexroth WinStudio"™ software. The document provides technical information and step-by-step instructions to create web-enabled HMI/SCADA programs.

IndraWorks 15VRS Software Installation

DOK-IWORKS-SOFTINS*V15-CORS-EN-P, R911393450

This documentation describes the IndraWorks installation.

IndraWorks 15VRS Engineering

DOK-IWORKS-ENGINEE*V15-APRS-EN-P, R911393303

This documentation describes the use of IndraWorks in which the Rexroth Engineering tools are integrated. It includes instructions on how to work with IndraWorks and how to operate the oscilloscope function.

IndraWorks 15VRS PLC Programming System IndraLogic 2G

DOK-IWORKS-IL2GPRO*V15-APRS-EN-P, R911396137

This documentation describes the PLC programming tool IndraLogic 2G and its use. The documentation includes the basic use, first steps, visualization, menu items and editors.

IndraWorks 15VRS HMI

DOK-MLC***-HMI****V15-APRS-EN-P, R911399270

This documentation describes the functions, configuration and operation of the user interfaces IndraWorks HMI Engineering and IndraWorks HMI Operation.

MTX 15VRS Shape Cutting Technology

DOK-MTX***-TECHCUT*V15-CORS-EN-P, R911399147

This documentation describes the shape cutting technology used by the machines to cut materials using an NC-controlled tool head.

MTX-integrated CAD/CAM System for Shape Cutting

DOK-MTX***-CAD*TECHCUT-CO01-EN-P, R911393320

This documentation describes the mode of operation and the area of application of the CAD/CAM nesting software Lantek Expert Inside.

RS

Corresponding edition

Tab. 1-4:

MTX documentation overview - Commissioning

1.2.4 Operation

Documentation titles with type codes and part numbers

MTX 12VRS Block Pre-Run

DOK-MTX***-BLK*RUN*V12-APRS-EN-P, R911334379

This documentation explains to the machine manufacturer how to setup the "Block pre-run" function at the machine for the end user.

MTX 15VRS Programming Manual

DOK-MTX***-NC**PRO*V15-RERS-EN-P, R911393318

This documentation describes the standard programming of the MTX control. Apart from the basics of NC programming, the use of NC functions according to DIN 66025 as well as the NC functions with high-level language syntax and CPL functions are described.

MTX 15VRS Standard NC Operation

DOK-MTX***-NC*OP***V15-APRS-EN-P, R911393314

This documentation describes the operation of the standard user interface of the NC control of the MTX. It includes the operation of the interface, the NC program development as well as the tool management.

MTX 15VRS Multitouch

DOK-MTX***-MULTI***V15-APRS-EN-P, R911393311

The Multitouch user interface of the NC control MTX is described in this documentation.

MTX 15VRS Standard NC Cycles

DOK-MTX***-NC*CYC**V15-PRRS-EN-P, R911394940

This documentation describes the application of the standard cycles of the different technologies for the MTX control.

MTX 15VRS NC Simulation

DOK-MTX***-NC*SIM**V15-APRS-EN-P, R911393273

This documentation describes the NC simulation for the MTX control.

MTX 15VRS Measuring Functions

DOK-MTX***-MES*FUN*V15-APRS-EN-P, R91194938

This documentation describes the measuring cycles of the MTX control.

RS

Corresponding edition

Tab. 1-5:

MTX documentation overview - Operating

1.2.5 OEM Engineering

Documentation titles with type codes and part numbers

MTX 13VRS Automation Interface DOK-MTX***-AUT*INT*V13-APRS-EN-P, R911337274 This documentation describes the script-based access to IndraWorks project data via the interface of the Automation Interface. Different objects including code examples are described. The Automation Builder is also described in this manual.
MTX 15VRS OPC Communication DOK-MTX***-OPC*COM*V15-PRRS-EN-P, R911399272 This documentation describes the syntax and the structure of the items for the communication with Bosch Rexroth devices.
IndraWorks OPC UA Communication DOK-IWORKS-OPC*UA*****-APRS-EN-P, R911379309 This documentation describes the OPC UA communication of the MLC and MTX control systems.

RS Corresponding edition
Tab. 1-6: MTX documentation overview - OEM engineering

1.2.6 AddOns

Documentation titles with type codes and part numbers

MTX 11VRS Action Recorder DOK-MTX***-ACR*****V11-APxx-EN-P, R911329943 This documentation describes the MTX action recorder. It includes the installation and commissioning as well as interface signals, application and operation.
MTX 15VRS Efficiency Workbench MTX cta, MTX ega DOK-MTX***-EWB*****V15-APRS-EN-P, R911400178 This documentation describes the mode of operation and the area of application of the analysis tool MTX cta and MTX ega.
MTX Remote Condition Monitoring DOK-MTX***-RCM*****V01-APRS-EN-P, R911334383 This documentation describes the operation of the Remote Condition Monitoring System.
MTX visIREC User Documentation DOK-MTX***-VISIREC*V01-APRS-EN-P, R911344242 This documentation describes the analysis tool visIREC used to optimize the free form surface milling process. 2D or 4D display of path-related data. 2D or 4D display of coordinate-related data. Analysis of critical areas (path and orientation deviation). Comparing the programmed NC blocks to the interpolated NC blocks.

xx / RS Corresponding edition
Tab. 1-7: MTX documentation overview - AddOns

1.3 Using safety instructions

1.3.1 Structure of the safety instructions

The safety instructions are structured as follows:

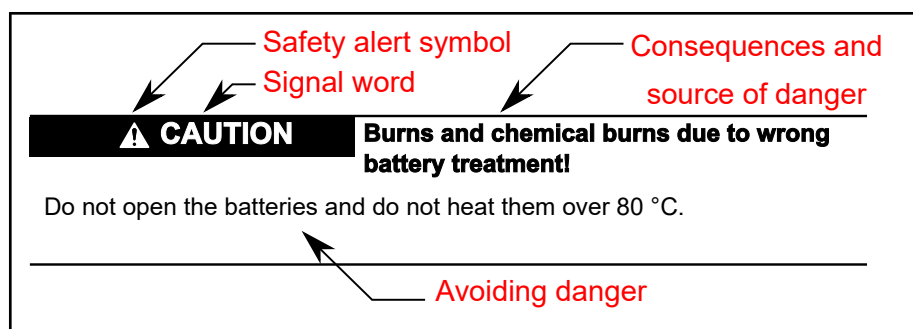


Fig. 1-2: Structure of the safety instructions

1.3.2 Explaining signal words and safety alert symbol

The safety instructions in this documentation contain specific signal words (danger, warning, caution, notice) and, if necessary, a safety alert symbol (according to ANSI Z535.6-2006).

The signal word draws attention to the safety instruction and indicates the risk potential.

The safety alert symbol (triangular safety reflector with exclamation marks), preceding the signal words Danger, Warning, Caution indicates hazards for persons.

DANGER

In case of non-compliance with this safety instruction, death or serious injury **will** occur.

WARNING

In case of non-compliance with this safety instruction, death or serious injury **can** occur.

CAUTION

In case of non-compliance with this safety instruction, minor or moderate injury can occur.

NOTICE

In case of non-compliance with this safety instruction, material damage can occur.

1.3.3 Symbols used

Pointers are displayed as follows:



This is a note.

Tips are displayed as follows:



This is a tip.

1.3.4 Explaining the signal alert symbol on the device



If this symbol is on your device, you have to observe the documentation on the device. The respective documentation informs on the type of hazard as well as the steps required to avoid this hazard.

1.4 Names and abbreviations

Term	Explanation
IWE	IndraWorks Engineering
IWO	IndraWorks Operation
NC	Numerical Control
OEM	Original Equipment Manufacturer
Profibus	Communication connection
Sercos	Communication connection

Tab. 1-8: Terms and abbreviations

1.5 Customer feedback

Customer requests, comments or suggestions for improvement are of great importance to us. Please email your feedback on the documentations to Feedback.Documentation@boschrexroth.de. Directly insert comments in the electronic PDF document and send the PDF file to Bosch Rexroth.

2 Important instructions on use

2.1 Intended use

2.1.1 Introduction

Bosch Rexroth products are developed and manufactured according to the state-of-the-art. The products are tested prior to delivery to ensure operating safety and reliability.

The products may only be used as intended. If they are not used as intended, situations occur that result in damage to property or injury to persons.



Bosch Rexroth shall not assume any warranty, liability or payment of damages in case of damage resulting from a non-intended use of the products; the user shall solely bear all risks from unintended use of the products.

Before using Bosch Rexroth products, the following requirements have to be met to guarantee the intended use of the products:

- Anyone handling Bosch Rexroth products in any way is obliged to read and consent to the relevant safety instructions and the intended use.
- Hardware products may not be altered and have to remain in their original state; i.e. no structural changes are permitted. It is not permitted to decompile software products or alter source codes.
- Do not install damaged or defective products or use them in operation.
- It has to be ensured that the products have been installed as described in the relevant documentation.



Ensure that the data present in the control or entered or read in by the user is correct before applying it to exclude unwanted axis motion. It can be the following invalid or old data:

- Part programs
- ZO tables
- Compensation tables
- Tool tables
- Permanent CPL variables
- Remanent PLC data
- Permanent system data

2.1.2 Areas of use and application

The MTX control is used to

- program contour and machining technology (path feed, spindle speed, tool change) of a workpiece.
- guide a machining tool along a programmed path.

Feed drives, spindles and auxiliary axes of a machine tool are activated via Sercos interface.



This additionally requires I/O components for the integrated PLC which - together with the actual CNC - control the machining process as a whole and also monitors this process with regard to technical safety.

It may only be operated with the explicitly specified hardware component configurations and combinations and only with the software and firmware specified in the appropriate documents and functional descriptions.

The CNC system MTX provides the perfect CNC system solution for cutting and forming for the following technologies:

- Rotate
- Milling
- Drilling
- Grinding
- Bending
- Nibbling
- Punching
- Contour cutting
- Handling

2.2 Unintended use

Using the CNC system MTX outside the previously specified areas of application or under operating conditions other than the conditions described in the documentation and the specified technical data, is defined as "unintended use".

The CNC system MTX must not be used if...

- it is subjected to operating conditions not corresponding to the specified ambient conditions. Operation under water, under extreme temperature fluctuations or under extreme maximum temperatures is prohibited.
- Furthermore, the CNC system MTX shall not be used for applications not expressly approved of by Bosch Rexroth. Therefore, please read the information given in the general safety instructions!
- The CNC system MTX may not be used in systems or machines connected to the internet via an unsecure network connection. Otherwise, malfunctions or a control failure can result due to unauthorized access.

3 General information

3.1 Introduction

NC cycles are an effective tool for the NC workpiece programming. They allow the programming of selected geometric elements and technological sequences simply by using cycle calls and parameter specifications.

The standard cycles are part of the firmware. It is not possible to change these cycles in the firmware. Superimposing of standard cycle files is possible (e.g. as hotfix with cycle files). However, it should be avoided as standard cycles are subject to change with each new release/version due to fault removal or function extensions.

The folder reserved for superimposing of standard cycles in the NC file system is \usr\bosch.



After a new version or a new release has been installed, standard cycle files contained in folder \usr\bosch or a different folder contained in the search routine for subroutine files, should be deleted. Thus, new corrections and function extensions of the installation are provided.

3.2 G-code assignment

Pre-assigned G-code ranges for cycles

Cycle groups	G-code range		Remark
Reserved by the control	G00 - G79		Relevant G-functions
Drilling cycles (part 1)	G80 - G89		
Reserved by the control	G90 - G109		Relevant G-functions
Hole patterns	G110 - G129		
Reserved by the control	G130 - G159		Partially assigned by control commands (3D radius correction, placements).
Turning cycles (part 1)	G160 - G179		
Drilling cycles (part 2)	G180 - G189		Reserved
Turning cycles (part 2)	G190 - G199		
Engraving cycles	G200 - G209		Reserved
Reserved by the control	G210 - G259		For future extensions
Milling cycles	G260 - G299		
Measuring cycles	G300 - G349		Reserved for future extensions
Fixed local end user cycles	G350 - G368	local	Can be used for the end user without parameter change.
Fixed modal end user cycles	G370 - G388	modal	Can be used for the end user without parameter change.
OEM user cycles	G390 - G449	local	Range reserved for OEM cycles.
USER user cycles	G450 - G499	modal	Range reserved for end user cycles.
Program assistance cycles (TFS)	G500 - G599		

General information

Cycle groups	G-code range		Remark
Measuring in space	G600 - G699		Reserved for future extensions
GNP cycles	G700 - G899		

Tab. 3-1: Pre-assignment of G-codes for cycles

The ranges G390 to G499 are available for the assignment of individual G-codes.



The G-code ranges listed above are reserved for the respective cycles in the MTX. Additional cycles can also be added for a new version/release. OEM and users should only use cycles in their reserved G-code ranges.

If the OEM or user uses arbitrary names for the cycles, it is the responsibility of OEM or user to ensure that there is no double use of the existing CNC key terms.

3.3 General rules for modal cycle calls

A modal cycle is executed at the position programmed in the same NC block N110. Only the position is programmed in the next NC block N120. It is first positioned and then the cycle is processed. This continues until the modal call is deleted with G80.

Examples: *Example 1:*

```
N100 G0 X0 Y0 Z100 S2500 M3
N110 X0 G81(IX Z,SL2,DT-5,RL2) F250 ; Set the parameters and 1st call
N120 X10 ; 2nd call
N120 X20 ; 3rd call
N130 G80 Z100 M5 ; Leave contour, delete modal call
```

If no position is programmed in the NC block N110, a modal cycle is only executed in the next NC block N120 in which a position was programmed.

Example 2:

```
N100 G0 X0 Y0 Z100 S2500 M3
N110 G81(IX Z,SL2,DT-5,RL2) F250 ; Set the parameters
N120 X10 ; 1st call
N120 X20 ; 2nd call
N130 G80 Z100 M5 ; Leave contour, delete modal call
```

Individual parameters cannot be modified with this type of programming. The parameter setting programmed again completely. It is not necessary to delete the modal function between the individual bores with G80.

Example 3:

```
N100 G0 X0 Y0 Z100 S2500 M3
N110 X0 G81(IX Z,SL2,DT-5,RL2) F250 ; Set the parameters and
; 1st call depth -5 Retract height 2
N120 X10 G81(DT-10,RL30) ; 2nd call depth -10 Retract height 30
N120 X20 G81(DT-5,RL2) ; 3rd call depth -5 Retract height 2
N130 X30 ; 4th call depth -5 Retract height 2
N140 G80 Z100 M5 ; Leave contour, delete modal call
```

If no parameters are specified in the cycle call, the SD parameters are used. Thus, individual parameters can also be modified. SD parameters are permanent and channel-specific.

Example 4:

```
N100 G0 X0 Y0 Z100 S2500 M3
1 SD.SysCyc.SL=2 : SD.SysCyc.DT=-5: SD.SysCyc.RL=2: REM Set the parameters
```

```

N110 X0 G81 F250 ; 1st depth call -5 Retract height 2
2 SD.SysCyc.DT=-10 : SD.SysCyc.RL=30
N120 X10 ; 2nd call depth -10 Retract height 30
3 SD.SysCyc.DT=-5 : SD.SysCyc.RL=2
N120 X20 ; 3rd call depth -5 Retract height 2
N130 X30 ; 4th call depth -5 Retract height 2
N140 G80 Z100 M5 ; Leave contour, delete modal call

```



When programming the cycles via system data, all parameters are considered as obligatory. Once used, the system data is used again for the next cycle call. It is the NC programmer's responsibility to enter a correct value for each system date.



Due to runtime optimization reasons, the cycles do not use a "WAIT". Thus, it is possible that the system variables "SD.SysCyc.xxx" have a different value during runtime in the variable display than during processing of cycles.

If this effect is to be avoided, the user has to insert WAIT instructions.

Optionally, the system date can also be written with the NC command SSDE. Block preparation is not stopped.

Code	Description	System date value during cycle processing
1 SD.SysCyc.DTC = 10	Written for block preparation	Undetermined
1 WAIT 2 SD.SysCyc.DTC = 10	Block preparation and block execution are synchronized by the "WAIT"	Determined 10
N10 SSDE[SD.SysCyc.DTC = 10]	System date is written for block execution	Determined 10

3.4 User cycles

The machine manufacturer (OEM) and the end user can create their own cycles. To use the MTX cycle functions (e.g. cycle export), the following rules have to be complied with:

Reserved directories on the control

The cycles of the machine manufacturer are to be saved in the data structure of the control under \usr\mtb.

The cycles of the end user are to be saved in the data structure of the control under \usr\user.

Reserved directories on the PC or operator panel

Store the input screens, images and texts of the machine manufacturer in the project under ...\"Visualization device" (e.g. VPP40\OEM\config\cycles).

Store the input screens, images and texts of the end user in the project under ...\"Visualization device" (e.g. VPP40\user\config\cycles).

There are various options to create user cycles. To create user cycles and input masks, refer to the instruction given in the "Commissioning" manual, see [chapter 1.2 "Required and supplementing documentation MTX" on page 2](#).

Cycles with addressed parameters:

Existing standard cycles can be copied, customized and used as user cycles. In this case, use the addressed parameters allowed in the cycles (if necessa-

ry, additional user parameters). If a user cycle with the same name is created, the standard cycle with the same name is no longer active.



The cycle used is determined by the search path. Only overwrite standard cycles in exceptional cases.

G-commands from the groups, which have not yet been implemented, may not be used, as these are reserved for future extensions of the standard cycles.

Cycles with position parameters:

Also create own cycles with a G-command (G390 to G499) or with own names without G-command. In this case, the parameters are to be provided as unaddressed position parameters (P1 to Pn).

3.5 Fixed machine operator cycles

Rules for fixed machine operator cycles

A series of cycles are fixedly defined for the machine operator. The operator can create cycles in a program code without changing the configuration data of the control. The following rules are to be observed:

G-codes G370 to G388 are reserved for modal cycles.

G-codes G350 to G368 are reserved for local cycles.

A fixed machine operator cycle contains up to ten transfer parameters. It is up to the operator whether all parameters are used.

There are no default input masks for fixed machine operator cycles. If masks are to be defined for these cycles, they have to be described in the respective cycle header definition file.

The NC program name of the cycle is **USRCYCxxx**.

xxx is the respective number of the G-code (e.g. G382 has the NC program name USRCYC382).



The content of the cycle depends on the machine operator. The cycle is not protected against modifications.

Save the NC program in the search path.

4 Overview on NC cycles

4.1 Modal G-codes for cycle calls

The maximum number of available modal cycles is 64.

G-code	Name	Default SR name	No. of Parameter
Drilling technology			
G80	Delete modal call		
G81	Drilling	STC_G81	5
G82	Countersinking	STC_G82	10
G83	Deep hole drilling	STC_G83	13
G84	Tapping Without compensating chuck	STC_G84	7
G85	Reaming	STC_G85	6
G86	Fine boring	STC_G86	8
G87	Back boring	STC_G87	8
G88	Gun drilling	STC_G88	13
G89	Rigid tapping with chip breaking	STC_G89	9
Technology Turning			
G165	Thread single cut	STC_G165	5
Milling technology			
G261	Thread milling	STC_G261	15
G262	Drill thread milling	STC_G262	16
G263	Groove circular milling	STC_G263	12
G266	Face milling, rectangle	STC_G266	17
G267	Face milling, circle	STC_G267	14
G271	Rectangular pocket milling	STC_G271	20
G272	Circular pocket milling	STC_G272	16
G274	Straight slot milling	STC_G274	16
G275	Curved slot milling	STC_G275	18
G276	Rectangular pin milling	STC_G276	20
G277	Circular spigot milling	STC_G277	16
G279	Straight land milling	STC_G279	18
G280	Curved land milling	STC_G280	20

4.2 Non-modal G-codes for cycle calls

The maximum number of available non-modal cycles is 64.

Overview on NC cycles

G-code	Name	Default SR name	No. of Parameter
Hole patterns			
G111	Hole circle, rotary axis	STC_G111	n.r.
G112	Hole circle, main spindle	STC_G112	n.r.
G113	Hole circle, plane	STC_G113	n.r.
G114	Hole pattern, line	STC_G114	n.r.
G115	Hole pattern, matrix	STC_G115	n.r.
G116	Hole pattern, frame	STC_G116	n.r.
Technology turning			
G160	Longitudinal turning	STC_G160	n.r.
G161	Face turning	STC_G161	n.r.
G162	Tapered turning	STC_G162	n.r.
G166	Thread sequence	STC_G166	n.r.
G167	Thread turning	STC_G167	n.r.
G168	Face thread	STC_G168	n.r.
G169	Tapered thread	STC_G169	n.r.
G170	Contour finishing		n.r.
G171	Contour roughing, longitudinal		n.r.
G172	Contour roughing, plane		n.r.
G173	Contour roughing, parallel to contour		n.r.
G174	Contour grooving		n.r.
G191	Groove turning	STC_G191	n.r.
G192	Rounded groove turning	STC_G192	n.r.
G196	Thread undercut, turning	STC_G196	n.r.
G197	Undercut turning	STC_G197	n.r.
Milling technology			
G201	Linear text engraving	STC_G201	n.r.
G202	Circular text engraving	STC_G202	n.r.
G273	User-defined contour pocket milling		n.r.
G278	User-defined spigot milling		n.r.
G281	Contour pocket with islands		n.r.
G288	Island definition of contour pocket with islands		n.r.
G289	Starting point specification for contour pocket with islands		n.r.
G290	Path milling		n.r.
Punching/nibbling			
G213	Hole circle punching	STC_G213	n.r.

G-code	Name	Default SR name	No. of Parameter
G214	Hole pattern line punching	STC_G214	n.r.
G215	Hole pattern matrix punching	STC_G215	n.r.
G219	Multiple part punching	STC_G219	n.r.
G224	Nibble line	STC_G224	n.r.
G226	Nibble rectangular	STC_G226	n.r.
Special functions			
G230	Swiveling		n.r.
G501	Tool change (turning)	STC_G501	n.r.
G511	Tool change (milling)	STC_G511	n.r.
G591 *)	HSC settings	STC_G591	n.r.
G592	Approximated free form surface milling	STC_G592	n.r.

*) The cycle G591 is not supported anymore in MTX 14VRS and above.

4.3 Specified end user cycles, local

G-codes G350 to G368 are specified for local end user cycles.

G-code	Name	Default SR name	No. of Parameter
G350		STC_G350	
G351		STC_G351	
G352		STC_G352	
G353		STC_G353	
G354		STC_G354	
G355		STC_G355	
G356		STC_G356	
G357		STC_G357	
G358		STC_G358	
G359		STC_G359	
G360		STC_G360	
G361		STC_G361	
G362		STC_G362	
G363		STC_G363	
G364		STC_G364	
G365		STC_G365	
G366		STC_G366	
G367		STC_G367	
G368		STC_G368	

4.4 Specified end user cycles, modal

G-codes G370 to G388 are specified for modal end user cycles.

G-code	Name	Default SR name	No. of Parameter
G370		STC_G370	
G371		STC_G371	
G372		STC_G372	
G373		STC_G373	
G374		STC_G374	
G375		STC_G375	
G376		STC_G376	
G377		STC_G377	
G378		STC_G378	
G379		STC_G379	
G380		STC_G380	
G381		STC_G381	
G382		STC_G382	
G383		STC_G383	
G384		STC_G384	
G385		STC_G385	
G386		STC_G386	
G387		STC_G387	
G388		STC_G388	

4.5 G-codes for measuring cycles

The G-codes G300 - G349 and G540 - G549 are specified for measuring cycles.

G-code	Name	Default SR name	No. of Parameter
G301	Probe tool length	STP_G301	n.r.
G303	Probe tool radius	STP_G303	n.r.
G311	Calibrate probe - Tool length	STP_G311	n.r.
G315	Calibrate probe - Tool radius	STP_G315	n.r.
G317	Calibrate probe - Point	STP_G317	n.r.
G318	Calibrating the probe - Adjustment ring	STP_G318	n.r.
G319	Calibrate probe - Spigot	STP_G319	n.r.
G321	Probe point	STP_G321	n.r.
G323	Check workpiece availability	STP_G323	n.r.
G324	Probe boring	STP_G324	n.r.

G-code	Name	Default SR name	No. of Parameter
G327	Probe spigot	STP_G327	n.r.
G328	Ball head measuring	STP_G328	n.r.
G329	Probe land	STP_G329	n.r.
G331	Probe groove	STP_G331	n.r.
G333	Probe angle	STP_G333	n.r.
G541	Tolerance monitoring	STP_G541	n.r.
G542	Average	STP_G542	n.r.
G543	Percentage compensation	STP_G543	n.r.
G628	Calculating the orientation of a plane	STP_G628	n.r.
G629	Determining the center point of a calibrated ball	STP_G629	n.r.

For a detailed description, refer to the documentation "MTX xxVRS Measuring Functions" (DOK-MTX***-MES*FUN*Vxx-APRS-EN-P).

4.6 Cycles without G-codes

The G-codes G300 - G349 and G540 - G549 are specified for measuring cycles.

NC-code	Relev. First parameter	Name	Default SR name
SETBQ	n.r.	Set/reset tool/position status bit	SETBQ
GETBQ	n.r.	Read tool/position status bit	GETBQ
STP	n.r.	Static tool placement	STP
INT_TO_STR	n.r.	Format conversion Integer to string	INT_TO_STR
REAL_TO_STR	n.r.	Format conversion Real to string	REAL_TO_STR
TCH	n.r.	Tool change	TCH
STOCK	BLOCK	Cuboidal raw piece (milling)	STOCK
	CYLINDER	Cylindrical raw piece (milling)	
	BAR	Bar section (turning)	
	POLYGON	Multi-edge (turning)	
	SHEET	Sheet (punching)	
	REMOVE	Remove workpieces/models	
	RENEW	Renew workpiece	
STOCKIMP	n.r.	Import model	STOCKIMP
STOCKSEL	n.r.	Select workpiece holder	STOCKSEL
STOCKADD	n.r.	Replace/add workpieces/models	STOCKADD

4.7 Using RCB labels in cycles

To continue the program with re-entry after the program has been canceled, RCB labels have been programmed in the cycles.

 If an NC program is terminated during processing, the program can be continued at the last, predefined point. By programming the NC function "RCB(1)", this position has been specified within the cycles.

The behavior in the cycles is controlled via the channel-dependent system variable "SD.SysCyc[\$].RCB" in the calling user program as follows.

System variable	Value	Effect	
SD.SysCyc[\$].RCB	0	no label is set	set in the calling user program
	1	a label is only set at the start or the end of a cycle	
	2	additionally sets a label on each machining plane	

Tab. 4-1: Relevant system variable

 The state of the system variable SD.SysChSRun.LockTargBlock is saved when calling the cycle. The state can sporadically be influenced by the cycle and can be reset to default.

Example of user program with cycle call G271 (Rectangle pocket milling)

```
ZOS(1) RCB(0)
G54 G17
T20 M6
1 SD.SysCyc.RCB=0
G0 X0 Y0 Z15 S1000 M3
Z10
G271 (CV1, ORI2, LE60, WI40, R10, UL0, DT-15, CD8, FI800, RSW2, RSL2, FF300) F600
X0
G80
```

User program with cycle call G271

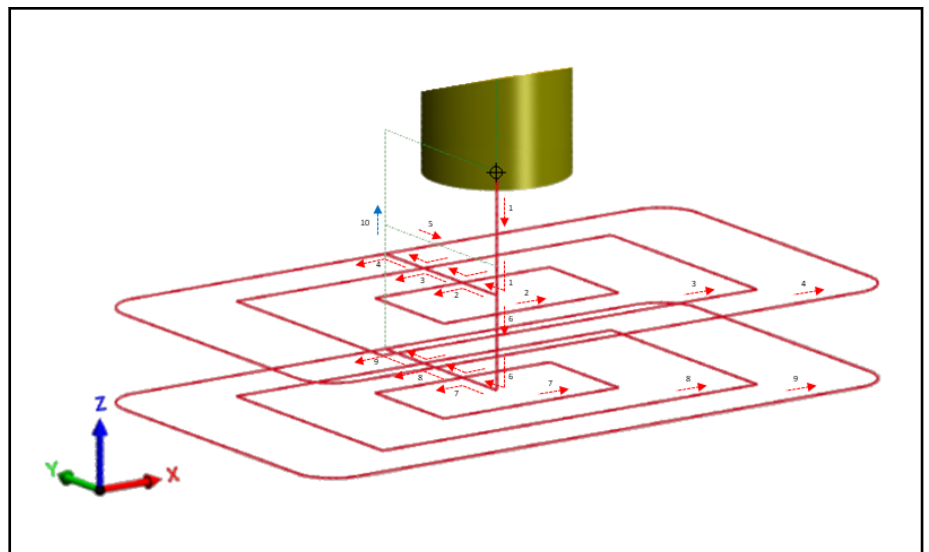


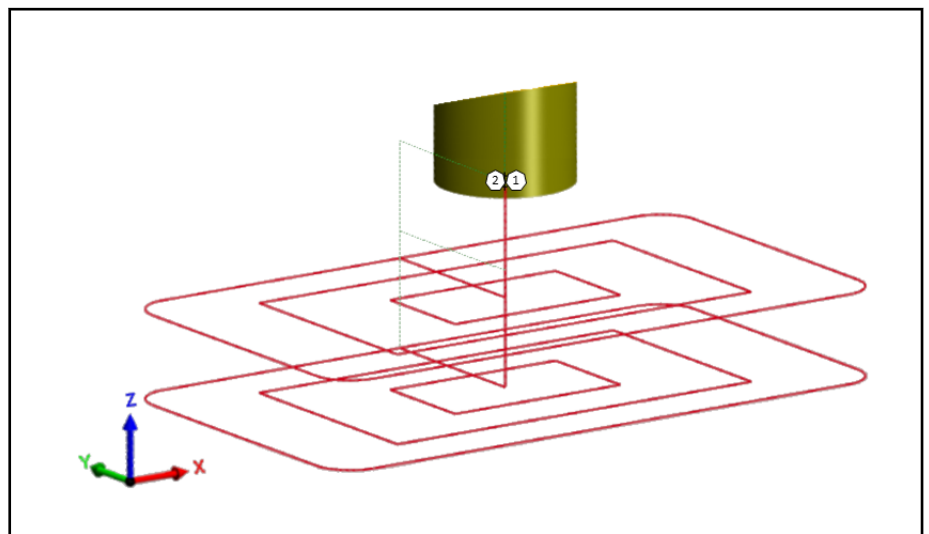
Fig. 4-1: Motion sequence with RCB=0

Motion sequence with RCB=1 before the cycle call

...

```
1 SD.SysCyc.RCB=1
G0 X0 Y0 Z15 S1000 M3
Z10
G271 (CV1, ORI2, LE60, WI40, R10, UL0, DT-15, CD8, FI800, RSW2, RSL2, FF300) F600
```

...



1 Re-entry positions at the start
2 Re-entry positions at the end

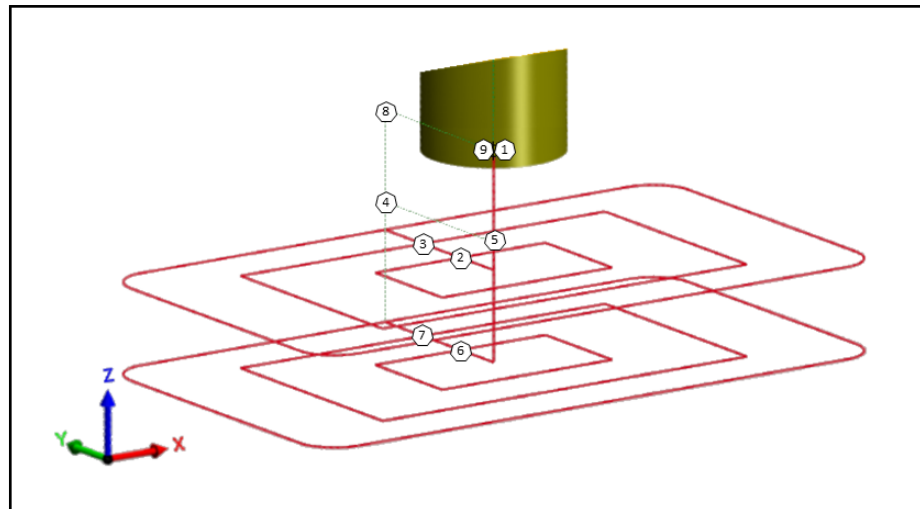
Fig. 4-2: Re-entry labels with RCB=1

Motion sequence with RCB=2 before the cycle call

...

```
1 SD.SysCyc.RCB=2
G0 X0 Y0 Z15 S1000 M3
Z10
G271 (CV1, ORI2, LE60, WI40, R10, UL0, DT-15, CD8, FI800, RSW2, RSL2, FF300) F600
```

...



1 ... 9 Re-entry positions

Fig. 4-3: Re-entry labels with RCB=2

"Re-entry" is possible for the following cycles.

Cycle	Name
G81	Drilling
G82	Countersinking
G83	Deep hole drilling
G84	Tapping
G85	Reaming
G86	Fine boring
G87	Back boring
G88	Gun drilling
G89	Rigid tapping with chip breaking
G160	Longitudinal turning
G161	Face turning
G162	Tapered turning
G165	Thread single cut
G166	Thread sequence
G167	Thread turning
G168	Face thread
G169	Tapered thread
G191	Groove turning
G192	Rounded groove turning
G196	Thread undercut, turning

Cycle	Name
G197	Undercut turning
G261	Thread milling
G262	Drill and thread milling
G263	Groove circular milling
G266	Face milling, rectangle
G267	Face milling, circle
G271	Rectangular pocket
G272	Circular pocket milling
G274	Straight slot milling
G275	Curved slot milling
G276	Rectangular pin milling
G277	Circular spigot milling
G279	Straight land milling
G280	Curved land milling
G201	Linear engraving
G202	Circle engraving

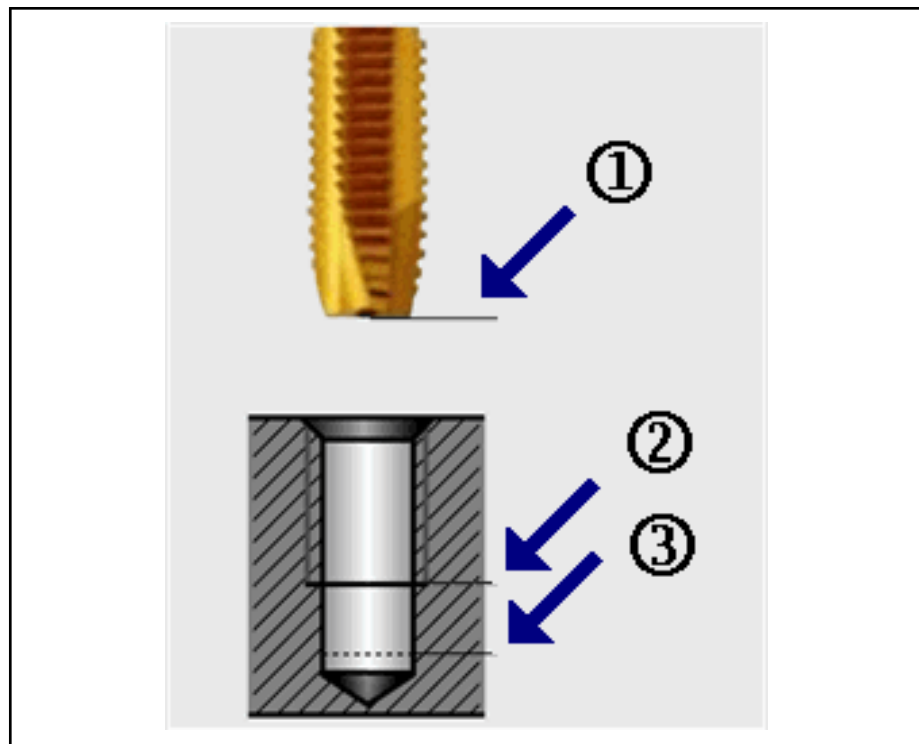
Tab. 4-2: Cycles with re-entry



If modal cycles are called indirectly by hole pattern cycles (G111...G116), re-entry actions cannot be used for modal cycles.

4.8 Approach starting position in the homing program

The cycle STC_HOME supports the user by automatically traversing the start position of the cycle in a customized homing program after the program has been canceled in a standard cycle.



- 1 Starting position
- 2 Abort position
- 3 Thread depth

Fig. 4-4: Example of start position of standard cycle (tapping G84)

Ensure that the following conditions are met:

- The user has to write a user-defined homing program to approach the channel home position.
- The homing program of the user has to be registered as asynchronous subroutine (ASUP).
- On the operator panel, a key to start the homing program has to be configured.
- The "STC_Home" cycle has to be called in the homing program of the machine manufacturer if the cycle with the ability for homing has been active during cancelation.

Basic procedure

1. Press <Home> button.
Homing program start via PLC.
2. Processing of actions defined by the machine manufacturer in part 1 of the homing program.
3. If a cycle in which homing is possible is active, the cycle STC_Home is started.
Travelling to starting position of the cycle.
4. If required, a subroutine to be defined by the user, e.g. "HOME-USRCH01" can be called in part 2 of the homing program (e.g. in case of recesses and use of a T-groove cutter).
5. Optionally, it can be tested for an active tool that has to be unloaded.

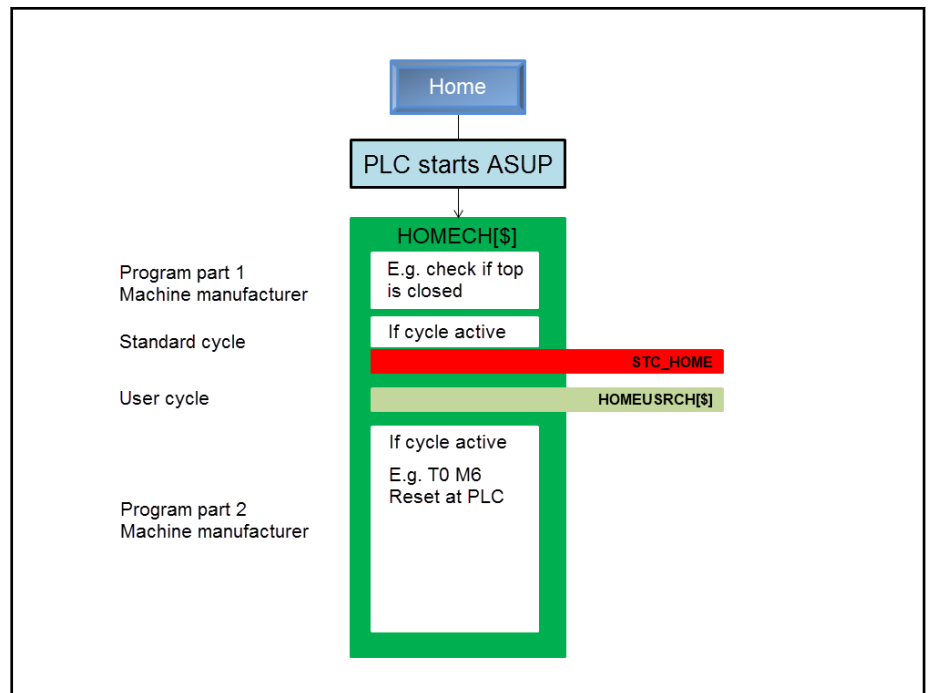


Fig. 4-5: Sequence of homing
Example of a homing program:

```

; ----- Home Program -----
1 SETWARN("HOMING: HOMECH01 was started")
; ----- MTB Part 1 -----
1 ABD% = PLC(1,,37,1) : REM PLC Command Query Cover referenced
1 IF (ABD% AND 2) = 0 THEN
1   SETERROR("Cover not callibrated, Reference cover at first and start Homing again")
1 ENDIF
; ----- Cycle activ -----
1 IF SD.SysCyc.CycName<>"" THEN
1   CALL STC_HOME
1 ENDIF
; ----- MTB Part 2 -----
1 CALL HOMEUSRCH01
; ----- Query Tool active -----
1 IF SD.SysTch[1].EXTERNAL_SP_EMPTY=TRUE THEN
1   T[0]
1 ENDIF
; -----
1 WAIT
1 SETWARN("HOMING: HOMECH01 is finished")
1 WAIT(,5000)
1 CLARWARN()
M31 ;RESET to PLC

```

"Homing" is possible for the following cycles.

Cycle	Name
G81	Drilling
G82	Countersinking
G83	Deep hole drilling
G84	Tapping
G85	Reaming
G86	Fine boring
G87	Back boring

Overview on NC cycles

Cycle	Name
G88	Gun drilling
G89	Rigid tapping with chip breaking
G261	Thread milling
G262	Drill and thread milling
G263	Groove circular milling
G201	Linear engraving
G202	Circle engraving

Tab. 4-3: Cycles with homing function

5 Drilling cycles

5.1 General information

The drilling cycles described in the following are modal and thus also suitable for processing on point patterns. If the cycles are given without parameter specification, the parameters of the assigned system variables are used.

5.2 G81 Drilling

Syntax G81(IX,SL,DT,DW,RL)
Brief description The modal cycle G81 performs simple drilling with ensured chip removal or centering (e.g. with NC spot drill).

Overview

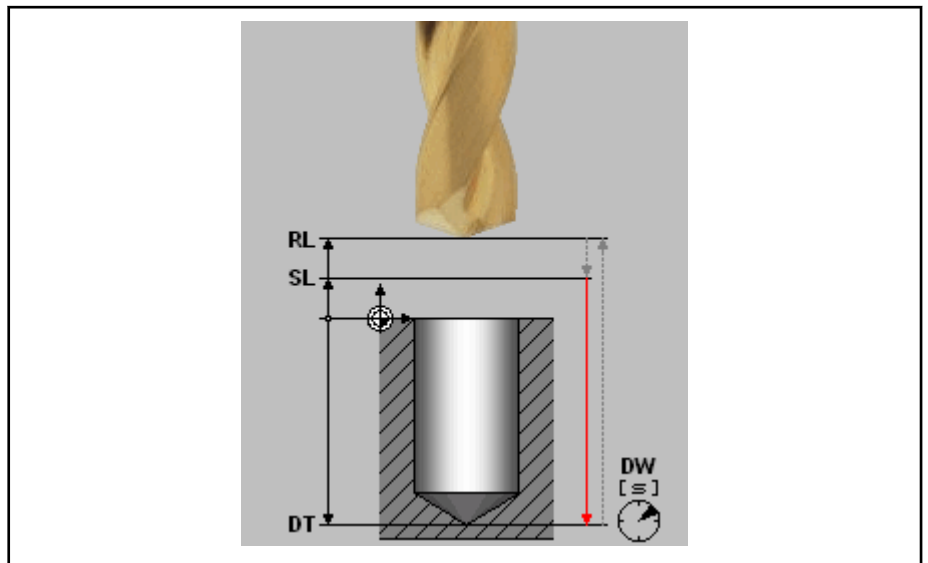


Fig. 5-1: G81 Drilling cycle

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
IX	Drill axis	Optional	String	Depending on the active plane G17 = Z, G18 = Z, G19 = X
SL	Safety level	Mandatory	Absolute	
DT	Drilling depth	Mandatory	Absolute	0
DW	Dwell time	Optional	≥ 0 Seconds	0
RL	Retraction height	Optional	Absolute	SL

Tab. 5-1: Parameter call

Description

IX Drill axis

Name of drill axis

If IX is not specified and if no drilling axis is available, the main axis of the interpolation plane to the drilling axis, i.e. when turning in G18 becomes the z-axis

SL Safety level

	Machining in feed starts at this absolute position
	DT Drilling depth
	Absolute position of the final depth to be reached
	DW Dwell time
	Period the driller "waits" at the lowest point
	RL Retract height
	This absolute position is the final position of the cycle
Special features	Note that a modal cycle is not executed with the cycle call during programming, but only after the next traversing command.
Procedure/processing	

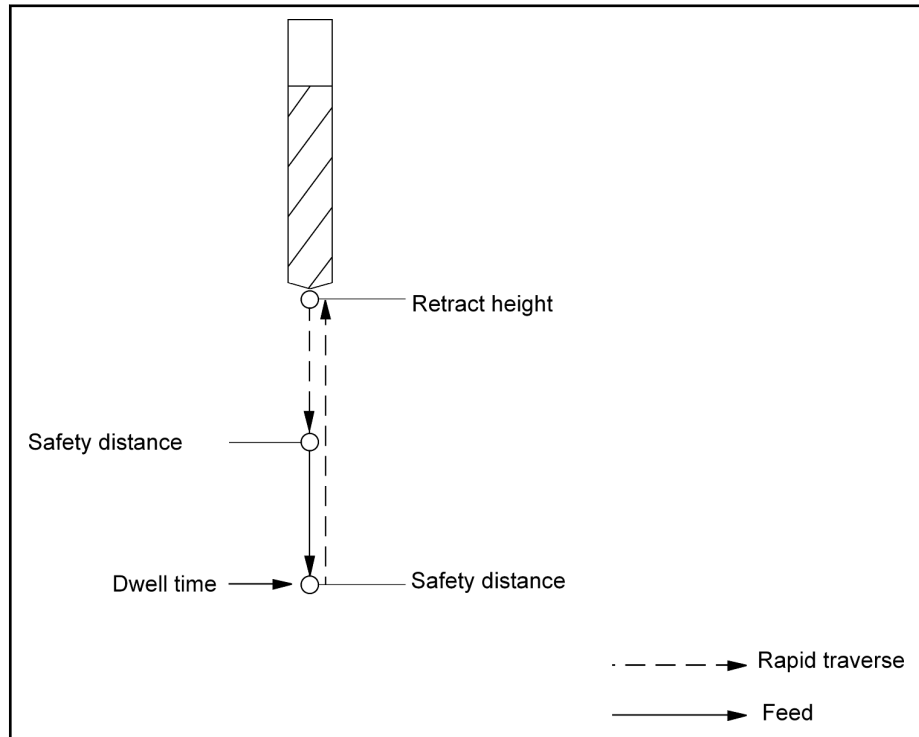


Fig. 5-2: Motion sequence G81 Drilling

From the last programmed position of the drill axis, it is positioned at the safety level in rapid traverse. From there, it is traveled to the drilling depth with feed. The feed value (feed/min for G94 or feed/revolution for G95) has to be programmed before or in the NC block of the cycle call at the latest. If a dwell time (optional) was specified, time elapses in seconds once the drilling depth is reached. Subsequently, the retract height is approached in rapid traverse.

Example

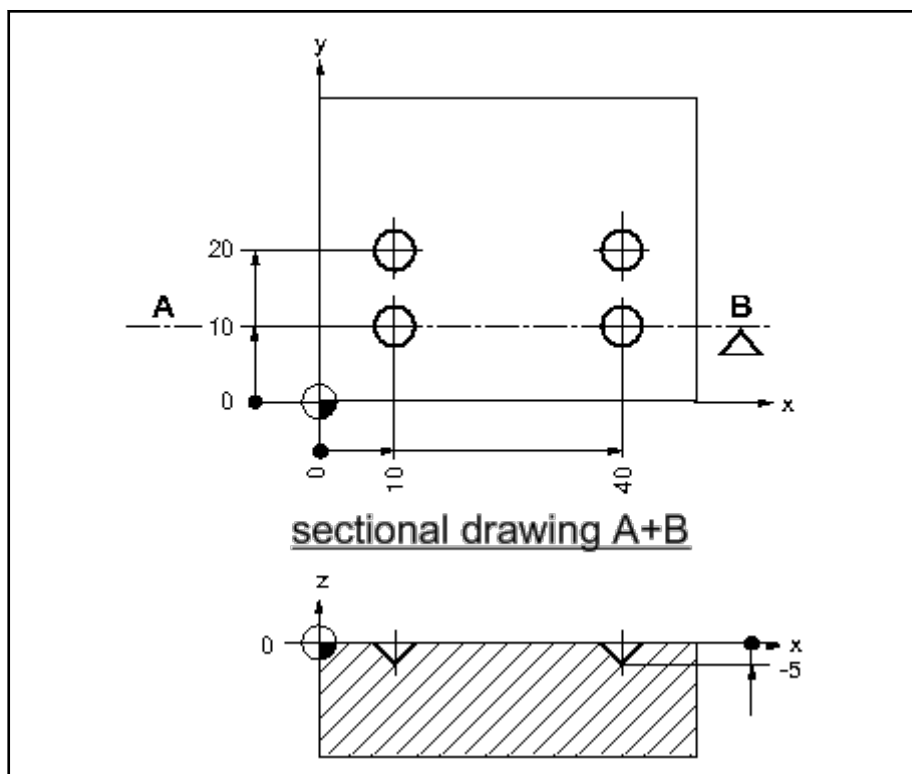


Fig. 5-3: Example of G81 Drilling

Program for G81 Drilling

```

N10 T1 M6                ; Tool change
N20 G17 G0 X10 Y10 Z100 S2500 M3 ; Prepositioning
N30 G81 (SL2, DT-5, DW1, RL10) F300 ; Cycle definition + feed
N40 X10 Y10              ; 1st cycle call
N50 Y20                  ; 2nd cycle call
N60 X40                  ; 3rd cycle call
N70 Y10                  ; 4th cycle call
N80 G80 Z100 M5          ; Leave contour, delete modal call

```

5.3 G82 Countersinking

Syntax	G82(IX,SL,DT,DW,MS,MFI,FD,FS,FF,RL)
Brief description	The modal cycle G82 splits one drilling in two chamfers and is suitable for example for centering with subsequent drilling or drilling with subland drills.

Overview

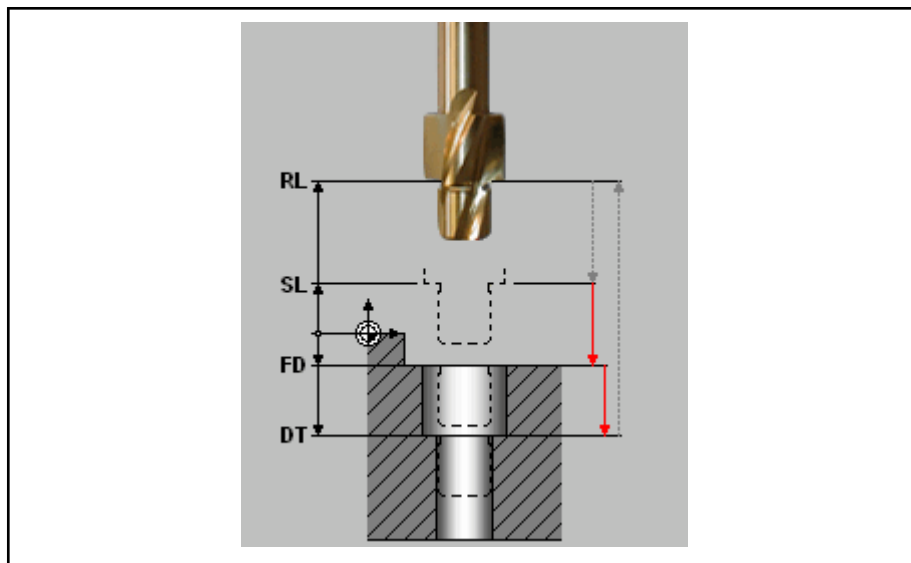


Fig. 5-4: G82 Countersinking cycle

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
IX	Drill axis	Optional	String Absolute	Depending on the active plane G17 = Z, G18 = X, G19 = Y
SL	Safety level	Mandatory	Absolute	
DT	Countersink depth	Mandatory	Absolute	
DW	Dwell time	Optional	≥ 0 Seconds	0
MS	Countersink speed	Optional	>0	Current speed
MFI	Countersink feed	Optional	>0	Current feed
FD	Entry depth	Optional	Absolute	No drilling level for entering
FS	Entry speed	Optional	>0	MS
FF	Entry feed	Optional	>0	MFI
RL	Retraction height	Optional	Absolute	SL

Tab. 5-2: Parameter call

Description

IX Drill axis

Name of drill axis

If IX is not specified and if no drilling axis is available, the main axis of the interpolation plane to the drilling axis, i.e. when turning in G18 becomes the z-axis

SL Safety level

Machining in feed starts at this absolute position

DT Countersink depth

Absolute position of the final depth to be reached

DW Dwell time

Duration the drilling machine "waits" at the lowest point

MS Countersink speed and MFI Countersink speed

Speed and feed during "countersinking"

FD Entry depth, FS Entry speed & FF Entry feed

It travels with entry speed and entry feed up to the entry depth

RL Retract height

This absolute position is the final position of the cycle

Special features Note that a modal cycle is not executed with the cycle call during programming, but only after the next traversing command.

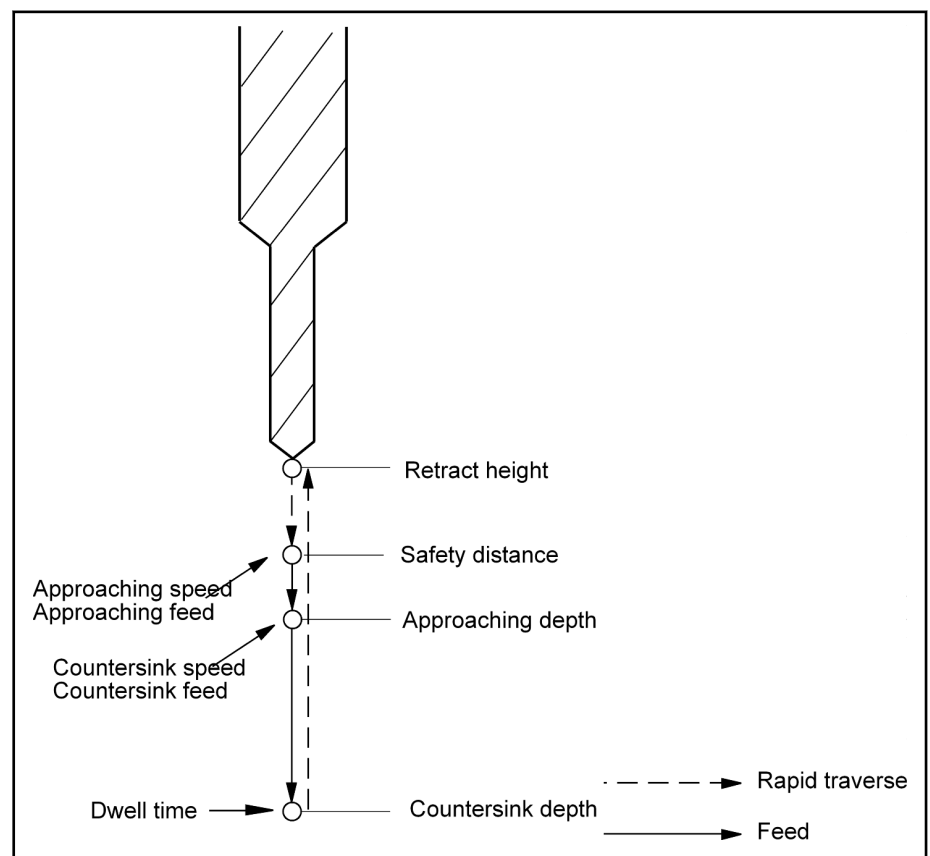
Procedure/processing

Fig. 5-5: Motion sequence G82 Countersinking

From the last programmed position of the drill axis, it is positioned at the safety level in rapid traverse. From there, it travels to the entry depth with entry feed and entry speed. This step cannot be performed without specifying the entry depth. Subsequently, the tool is traveled to the countersink depth with the countersink feed and at countersink speed. If no countersink feed is transferred in the cycle call, program a feed value prior to or no later than in the NC block of the cycle call. If a dwell time (option) is specified, it elapses in seconds once the countersink depth is reached. Subsequently, the retract height is approached in rapid traverse.

Example

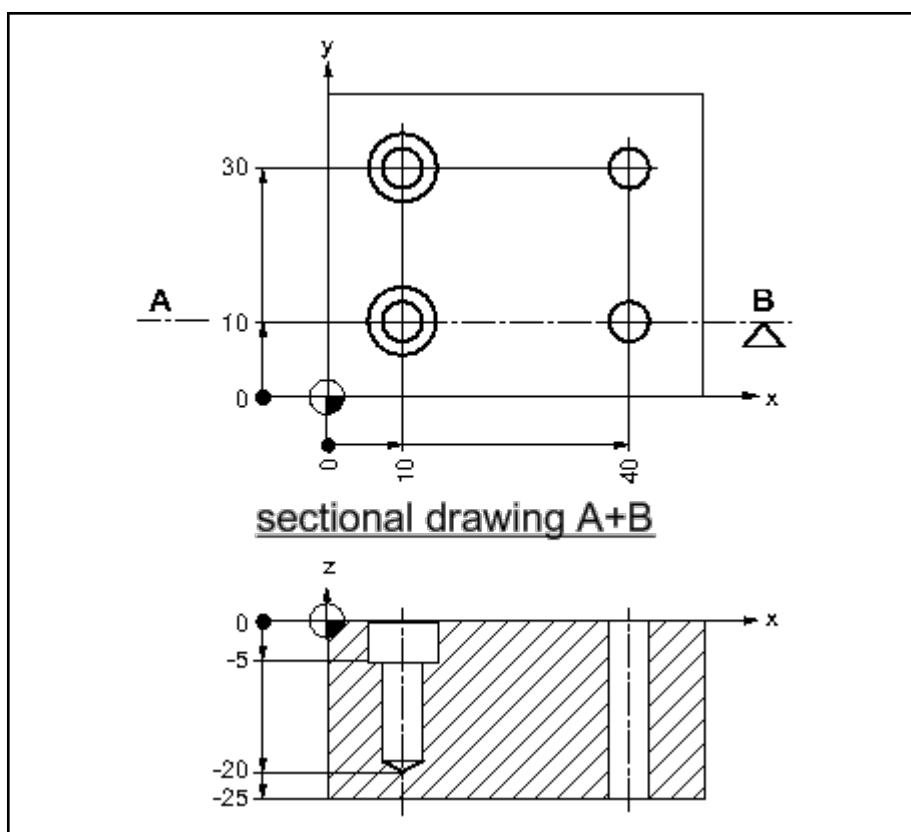


Fig. 5-6: Example of G82 Countersinking
Program for G82 Countersinking

```

N10 T1 M6 ; Tool change (step drill)
N20 G17 G0 X10 Y10 Z100 M3 ; Prepositioning
N30 G82 (SL2, DT-20, FD-15, FS1500, FF300, MS1200, MFI250, DW1, RL10) ; Cycle definition
N40 X10 Y10 ; 1st cycle call
N60 Y30 ; 2nd cycle call
N80 G80 Z100 M5 ; Leave contour, delete modal call

```

5.4 G83 Deep hole drilling

Syntax G83(IX,SL,DT,CD,RD,DW,RL,LD,DEG,CDM,FD,FS,FF)

Brief description Modal cycle G83 is suitable for drilling deeps holes for which chip breaking and chip removal has to be ensured.

Overview

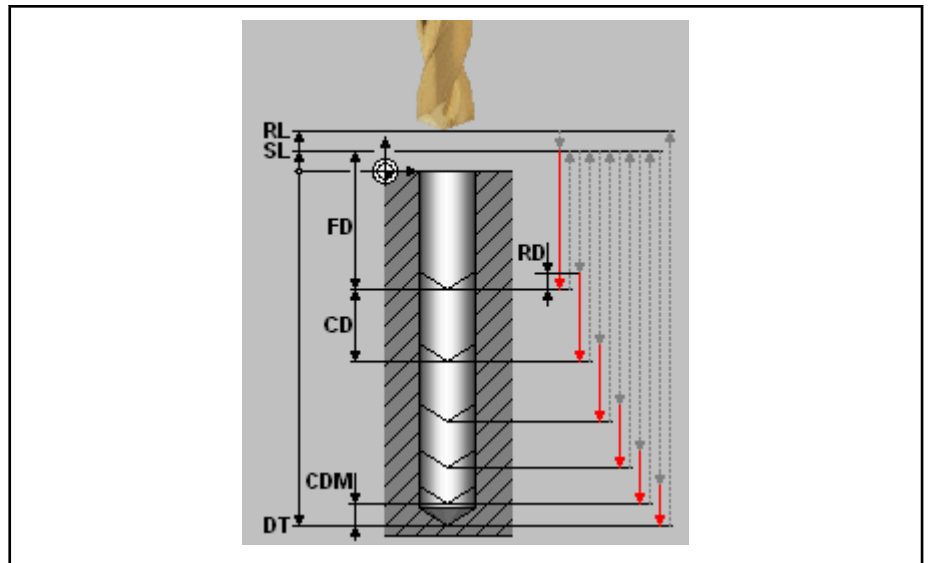


Fig. 5-7: G83 Deep hole drilling cycle

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
IX	Drill axis	Optional	String	Depending on the active plane G17 = Z, G18 = X, G19 = Y
SL	Safety level	Mandatory	Absolute	
DT	Drilling depth	Mandatory	Absolute	
CD	Infeed depth	Mandatory	> 0 Incremental	
RD	Approach distance/retracting path	Optional	>= 0 Incremental	1 mm
DW	Dwell time	Optional	>= 0 Seconds	
RL	Retraction height	Optional	Absolute	SL
LD	Chip breaking	Optional	Chip removal (0)/ Chip breaking (1)	Chip removal (0)
DEG	Degression	Optional Specify together with CDM		Constant in-feed depth (0)
CDM	Minimum in-feed depth	Mandatory if DEG < 0	> 0	
FD	Initial infeed depth	Optional	> 0	CD

Parameter	Designation	Properties	Values	Pre-assignment
FS	Starting speed	Optional	>0	Current speed
FF	Starting feed	Optional	>0	Current feed

Tab. 5-3: Parameter call

DEG	Effect
> 0:	Degression amount
0	Constant infeed depth
-1 < DEG < 0:	Degression factor

Tab. 5-4: Meaning of the parameter DEG - degression

Description	IX Countersink axis Name of drill axis If IX is not specified and if no drilling axis is available, the main axis of the interpolation plane to the drilling axis, i.e. when turning in G18 becomes the z-axis
	SL Safety level Machining in feed starts at this absolute position
	DT Drilling depth Absolute position of the final depth to be reached
	CD Infeed depth Infeed depth for every infeed
	RD approach distance/retract height Retract distance after each infeed
	DW Dwell time Period the driller "waits" at the lowest point
	RL Retract height This absolute position is the final position of the cycle
	LD Chip breaking Chip removal and chip breaking
	DEG Degression Reducing infeed depths. Specify together with CDM
	CDM Minimum infeed depth To be infeed by at least this value
	FD First infeed depth This absolute depth is approached at first infeed
	FS Starting speed and FF Starting feed Speed and feed for the first drilling stroke
Special features	Note that a modal cycle is not executed with the cycle call during programming, but only after the next traversing command.

Procedure/processing

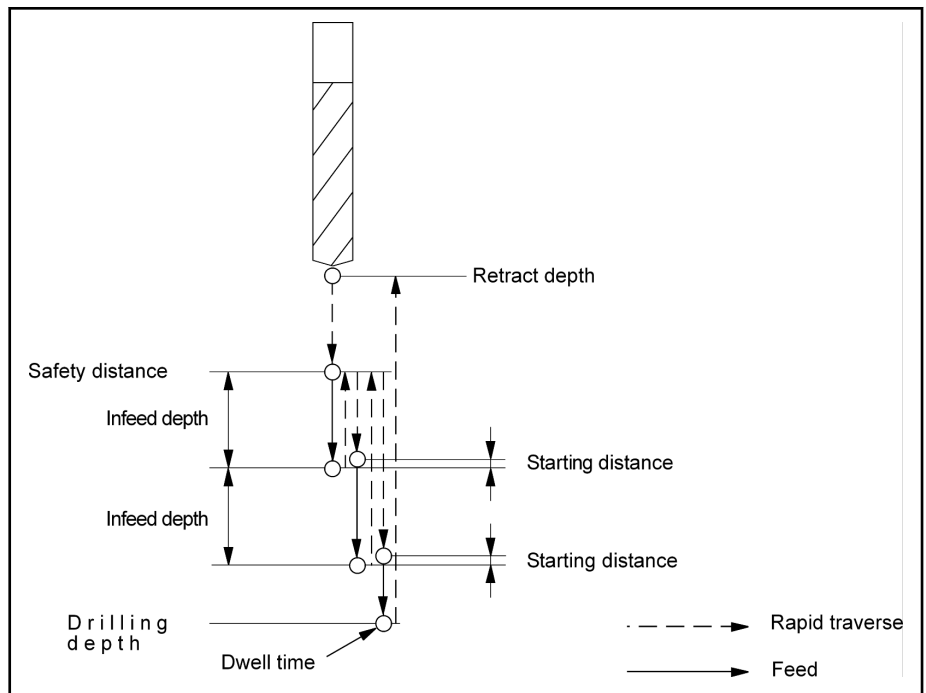


Fig. 5-8: Motion sequence G83 Deep hole drilling (chip removal)

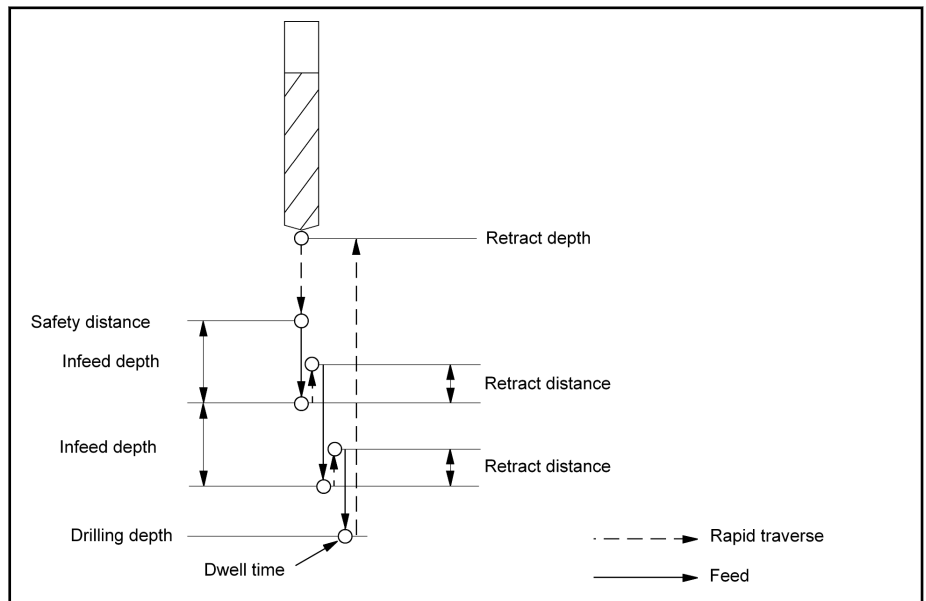


Fig. 5-9: Motion sequence G83 Deep hole drilling (chip breaking)

From the last programmed position of the drill axis, it is positioned at the safety level in rapid traverse. From there, the drill drills to the first infeed depth with feed. If "Chip removal (0)" was programmed in LD, it is positioned to the safety level with rapid traverse. Subsequently, it travels in rapid traverse to the respective infeed depth reduced by the starting distance. If "chip breaking" was selected in LD, the tool is retracted by the retract distance. This process is repeated until the drilling depth has been reached. Program the feed value prior to or not later than in the NC block of the cycle call. If a dwell time (optional) was specified, time elapses in seconds once the drilling depth is reached. Subsequently, the retract height is approached in rapid traverse.

Example

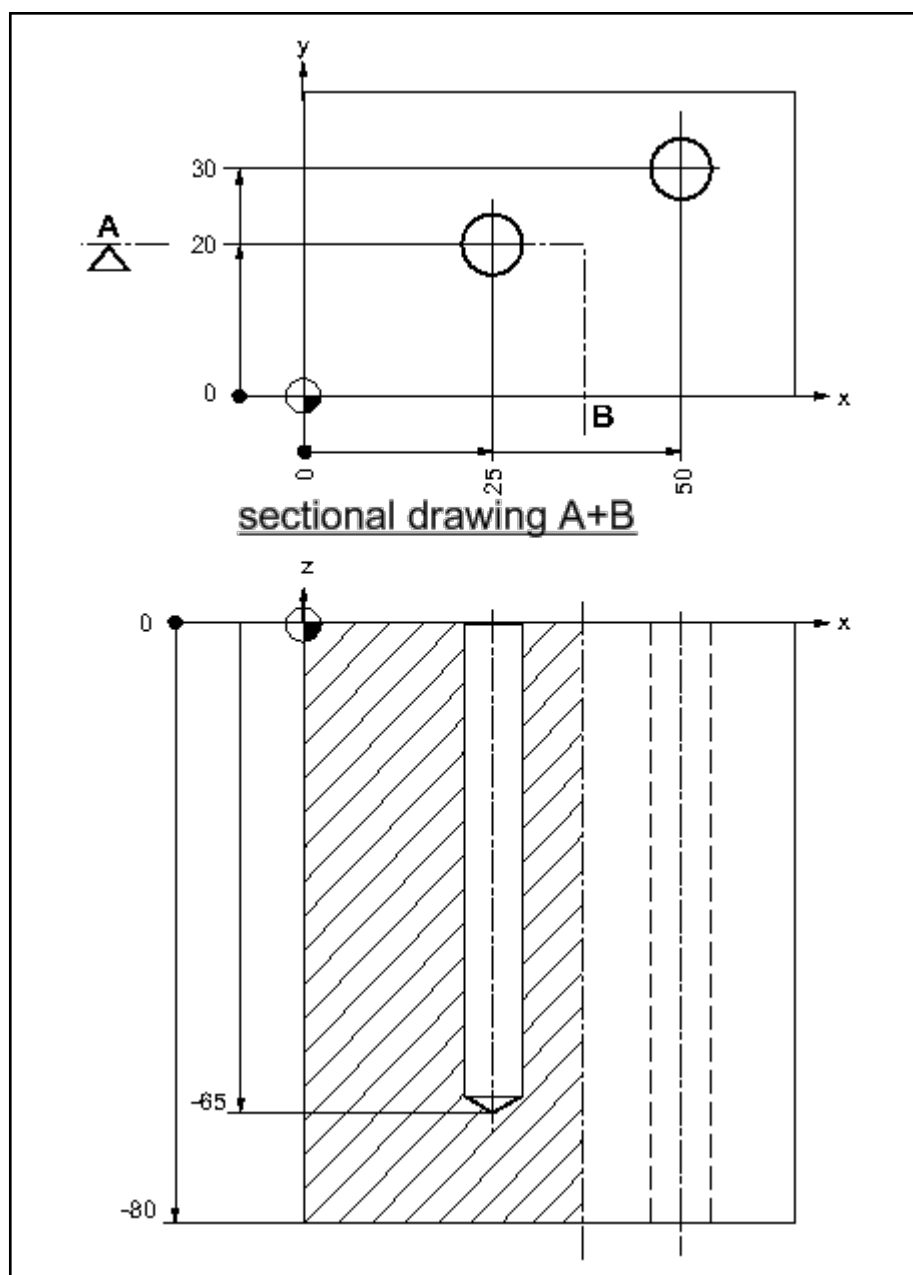


Fig. 5-10: Example of G83 Deep hole drilling

Program for G83 Deep hole drilling

```

N10 T1 M6 ; Tool change
N20 G17 G0 X25 Y20 Z100 S2500 M3 ; Prepositioning
N30 G83 (SL2, DT-65, CD20, RD1, FD30, DW1, DEG-0.2, CDM10, RL10) F300 ; Cycle definition + feed
; 1st cycle call
N40 X25 Y20 ; 1st cycle call
N50 G83 (DT-85) ; New drilling depth
N60 X50 Y30 ; 2nd cycle call
N70 G80 Z100 M5 ; Leave contour, delete modal call

```

5.5 G84 Tapping

Syntax G84(IX,SL,DT,DW,TP,MS,RS,RL)

Brief description Modal cycle G84 is suitable for making tapings, where chip breaking and chip removal are ensured. A rigid tap and a position-controlled spindle mode are used.

Overview

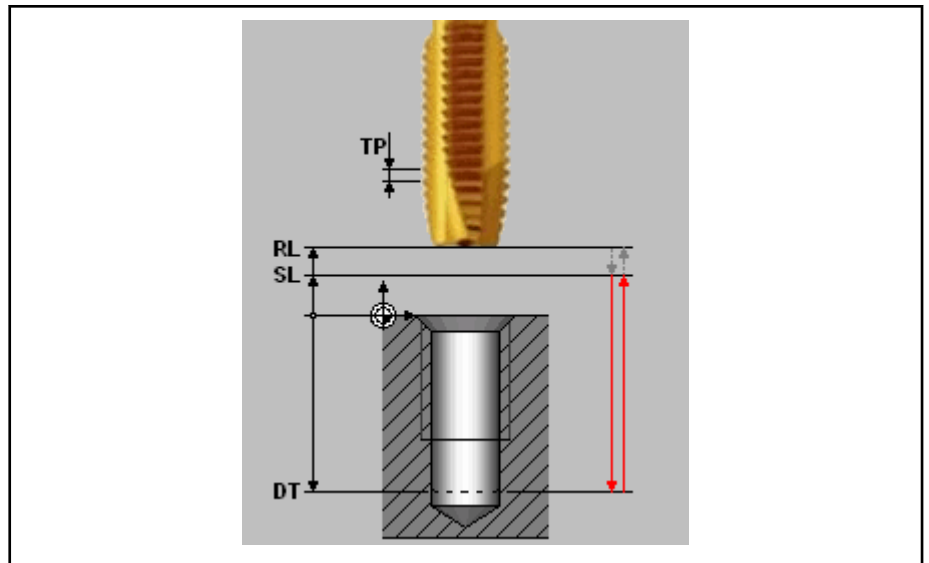


Fig. 5-11: Cycle G84 Tapping

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
IX	Drill axis	Optional	String	Depending on the active plane G17 = Z, G18 = Y, G19 = Z
SL	Safety level	Mandatory	Absolute	
DT	Thread depth	Mandatory	Absolute	
TP	Thread pitch	Mandatory	>0 right-hand thread <0 left-hand thread	
MS	Speed	Optional	>0	Current speed
RS	Speed for re-tracting	Optional		MS
RL	Retraction height	Optional	Absolute	SL

Tab. 5-5: Parameter call

Description

IX Drill axis

Name of drill axis

If IX is not specified and if no drilling axis is available, the main axis of the interpolation plane to the drilling axis, i.e. when turning in G18 becomes the z-axis

SL Safety level

Tapping starts at this absolute position

DT Thread depth

Absolute position of the final depth to be reached

TP Thread pitch

Pitch of thread

MS speed

Speed at thread cutting

RS Speed for retracting

Speed when leaving the thread

RL Retract height

This absolute position is the final position of the cycle

Special features

Note that a modal cycle is not executed with the cycle call during programming, but only after the next traversing command.

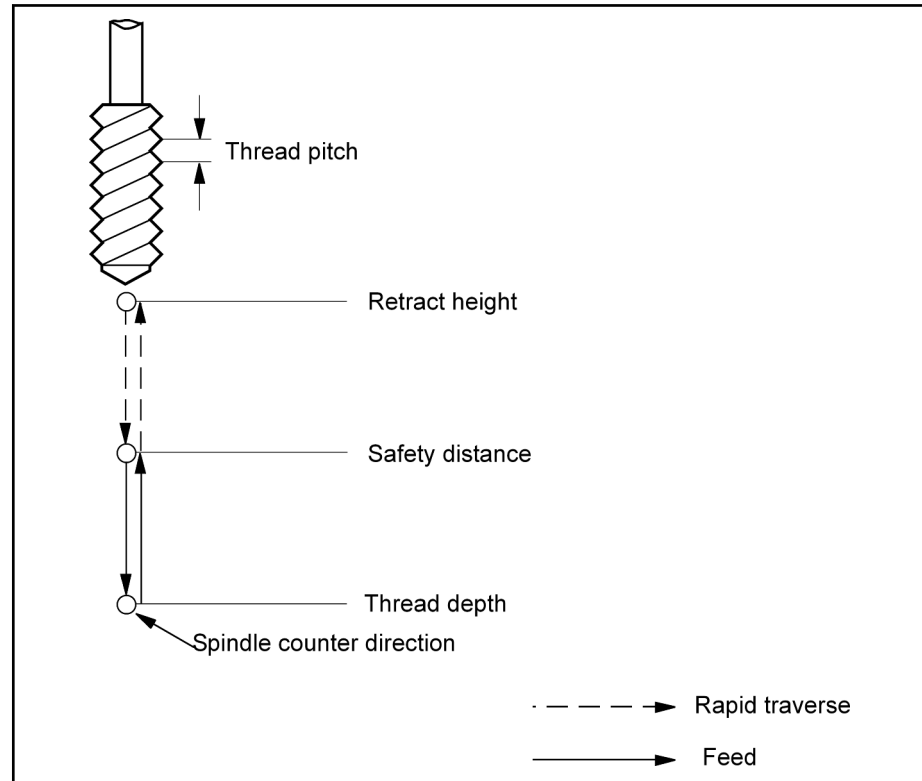
Procedure/processing

Fig. 5-12: Motion sequence G84 Tapping

From the last programmed position of the drill axis, it is positioned at the safety level in rapid traverse. From there, it travels with the previously programmed speed and the specified thread pitch to the thread depth. Program the reference spindle before the cycle is called with TSP. A left-hand thread can be created by specifying a negative pitch and a right-hand thread by specifying a positive pitch. If a dwell time (optional) is specified in seconds, it elapses once the thread depth is reached. The tool is returned to the safety level in opposing direction at the same speed or, if specified, at the speed used for retracting. Finally, the retract height is approached in rapid traverse.

Example

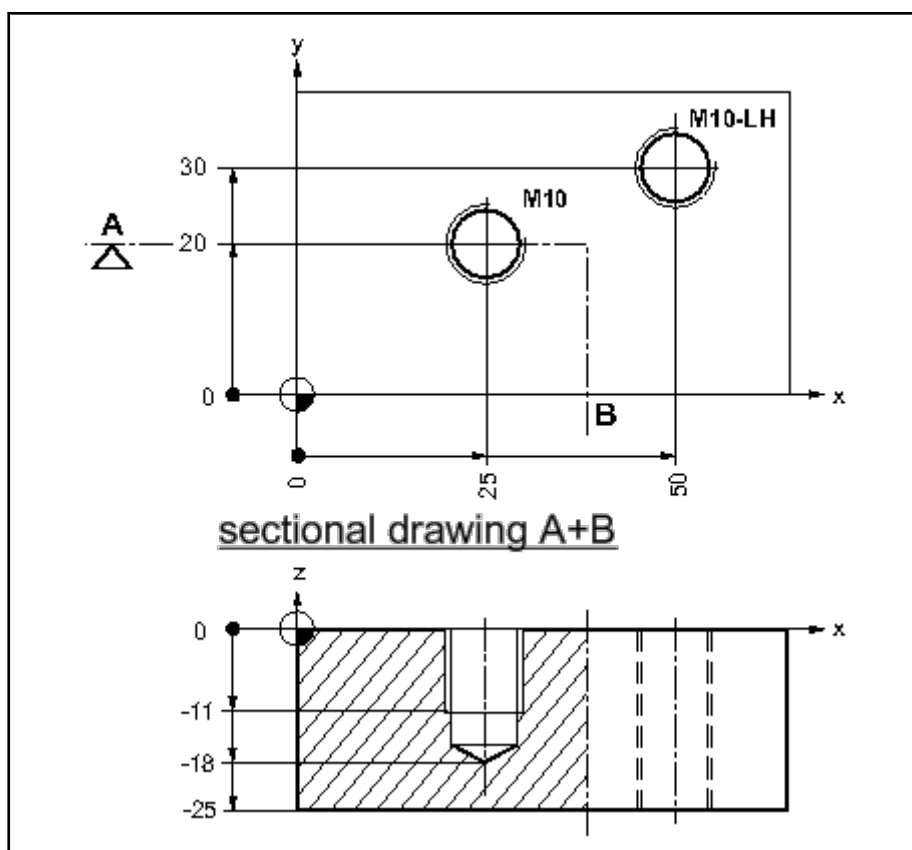


Fig. 5-13: Movement sequence

Program for G84 Tapping

```

N10 T1 M6 ; Tool change
N20 G17 G0 X25 Y20 Z100 ; Prepositioning
N30 TSP (GRP1) ; Reference spindle group 1
N40 G84 (SL2, DT-15, TP1.5, MS300, RS600, RL10) ; Cycle definition
N50 X25 Y20 ; 1st cycle call
N60 G84 (DT-30) ; New thread depth
N70 X50 Y30 ; 2nd cycle call
N80 G80 Z100 M5 ; Leave contour, delete modal call

```

5.6 G85 Reaming

Syntax G85(IX,SL,DT,DW,RFF,RL)

Brief description The modal cycle G85 finishes the finely toleranced holes with a reamer.

Overview

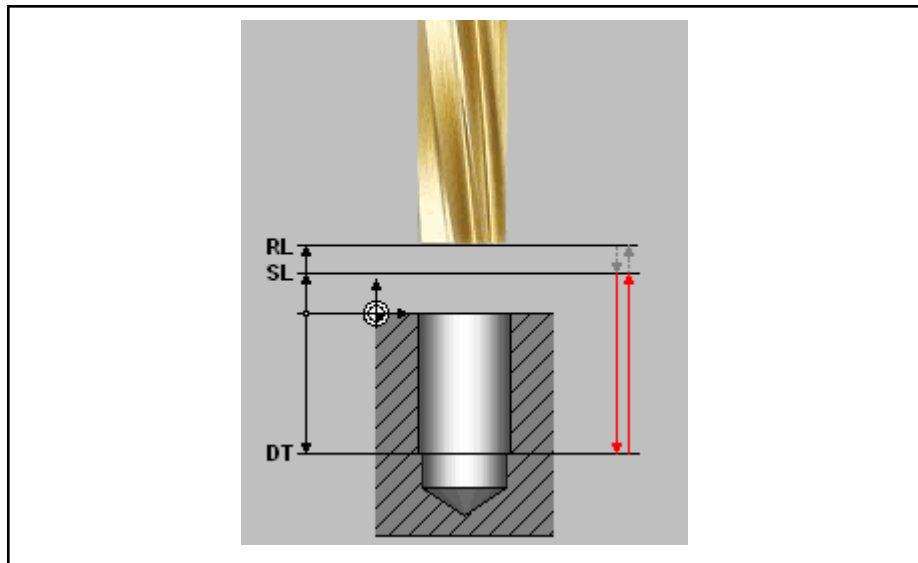


Fig. 5-14: Cycle G85 Reaming

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
IX	Drill axis	Optional	String	Depending on the active plane G17 = Z, G18 = X, G19 = Y
SL	Safety level	Mandatory	Absolute	
DT	Final depth	Mandatory	Absolute	
DW	Dwell time	Optional	≥ 0 Seconds	0
RFF	Velocity factor for retracting	Optional	≥ 1	1
RL	Retraction height	Optional	Absolute	SL

Tab. 5-6: Parameter call

Description

IX Drill axis

Name of drill axis

If IX is not specified and if no drilling axis is available, the main axis of the interpolation plane to the drilling axis, i.e. when turning in G18 becomes the z-axis

SL Safety level

Machining in feed starts at this absolute position

DT Final depth

Absolute position of the final depth to be reached

DW Dwell time

Period the driller "waits" at the lowest point

RFF velocity factor for the retraction

The velocity increases by this factor when leaving the hole

RL Retract height

This absolute position is the final position of the cycle

Special features

Note that a modal cycle is not executed with the cycle call during programming, but only after the next traversing command.

Procedure/processing

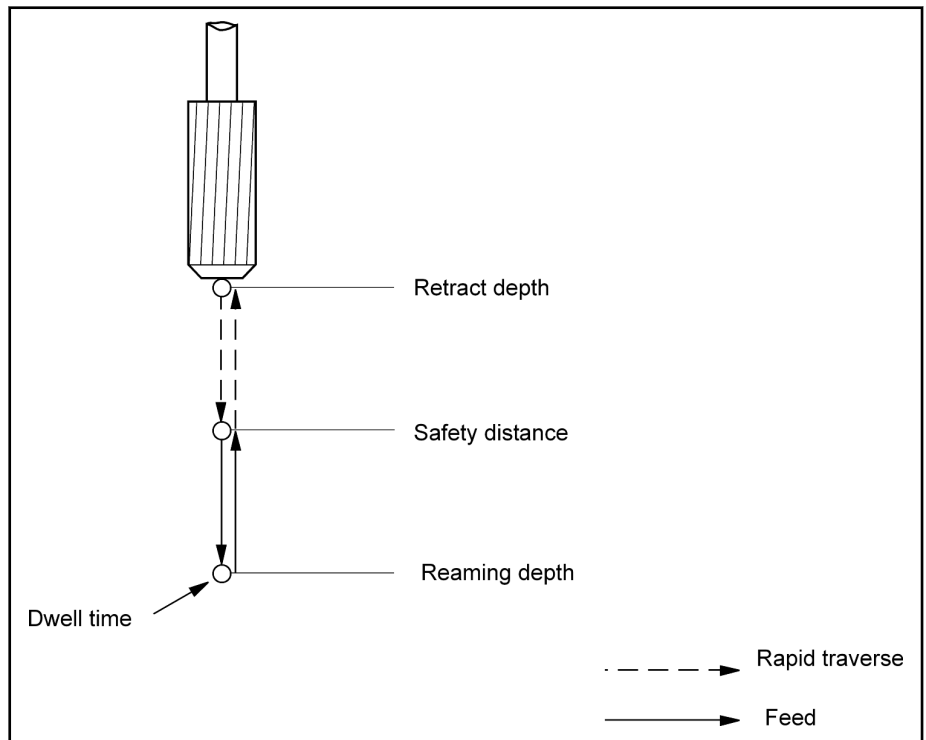


Fig. 5-15: Motion sequence G85 reaming

From the last programmed position of the reamer, it is positioned at the safety level in rapid traverse. From there, it is traveled down to the final depth with feed. Program the feed value prior to or not later than in the NC block of the cycle call. If a dwell time (optional) is specified in seconds, it elapses once the final depth is reached. The tool travels to the safety level with the same feed or, if a velocity factor for retraction is specified, with an increased feed and travels to the retract height in rapid traverse.

Example

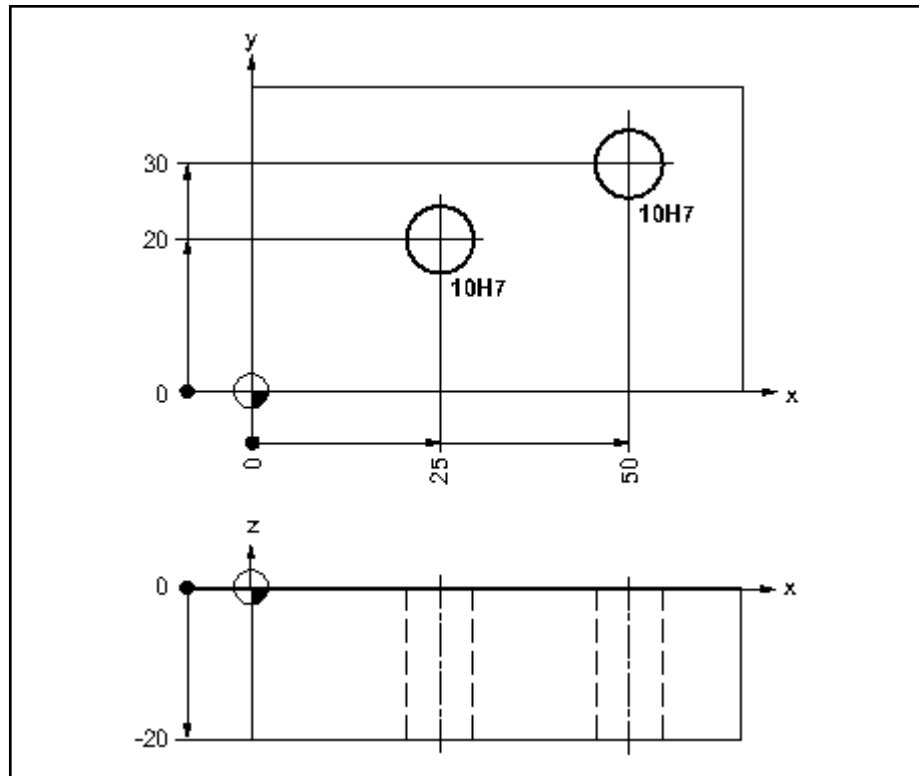


Fig. 5-16: Example of G85 Reaming
Program for G85 Reaming

```

N10 T1 M6 ; Tool change
N20 G17 G0 X25 Y20 Z100 S1000 M3 ; Prepositioning
N30 G85 (SL2, DT-25, DW1, RFF5, RL10) F150 ; Cycle definition + feed
N40 X25 Y20 ; 1st cycle call
N50 X50 Y30 ; 2nd cycle call
N60 G80 Z100 M5 ; Leave contour, delete modal call

```

5.7 G86 Fine boring

Syntax G86(IX,SL,DT,DW,OA,SX,RD,RL)

Brief description Modal cycle G86 is suitable for fine boring with a boring bar. This is especially useful for finely tolerance holes.

Overview

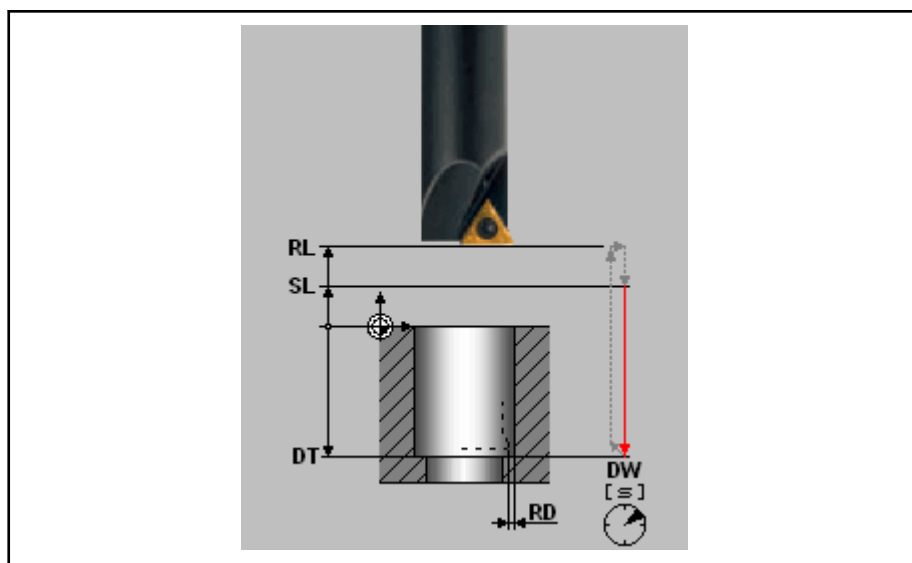


Fig. 5-17: Cycle G86 Fine boring

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
IX	Drill axis	Optional	String	Depending on the active plane G17 = Z, G18 = X, G19 = Y
SL	Safety level	Mandatory	Absolute	
DT	Drilling depth	Mandatory	Absolute	
DW	Dwell time	Optional	≥ 0 Seconds	0
OA	Orientation angle	Optional	Absolute Degree	0
SX	Retract axis	Mandatory	String	
RD	Retract path	Optional	≥ 0 Incremental	0.2 mm
RL	Retraction height	Optional	Absolute	SL

Tab. 5-7: Parameter call

Description

IX Drill axis

Name of drill axis

If IX is not specified and if no drilling axis is available, the main axis of the interpolation plane to the drilling axis, i.e. when turning in G18 becomes the z-axis

SL Safety level

Machining in feed starts at this absolute position

DT Drilling depth

Absolute position of the final depth to be reached

DW Dwell time

Period the driller "waits" at the lowest point

OA Orientation angle

Before leaving, the spindle is stopped on this angular position

SX Retract axis

The tool is retracted before leaving using this axis

RD Retract distance

The tool is axially and radially retracted by this value

RL Retract height

This absolute position is the final position of the cycle

Special features

Note that a modal cycle is not executed with the cycle call during programing, but only after the next traversing command.

Procedure/processing

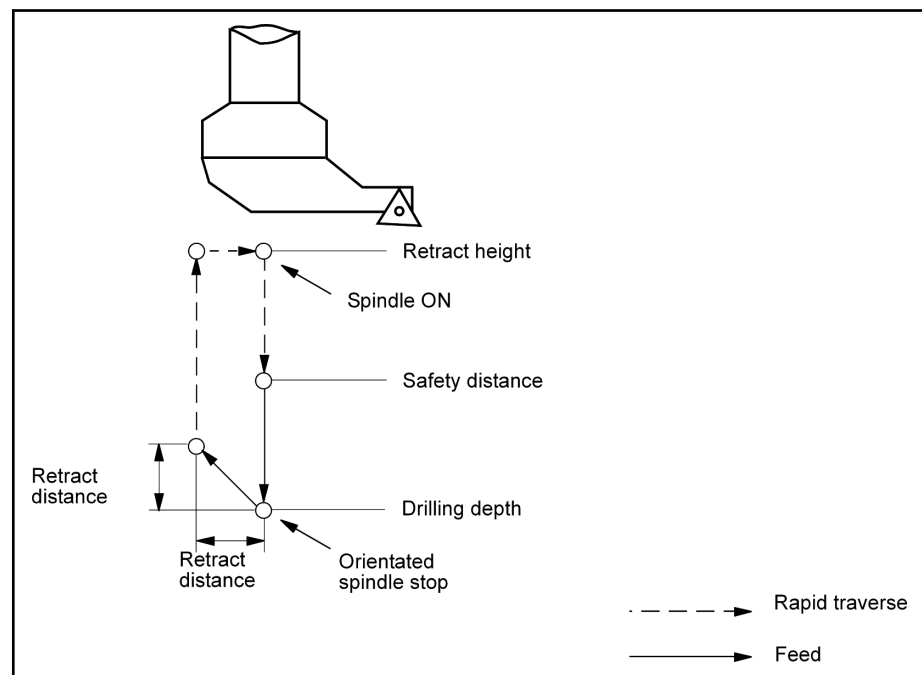


Fig. 5-18: Motion sequence G86 Fine boring

From the last programmed position of the drill axis, it is positioned at the safety level in rapid traverse. From there, it is traveled to the drilling depth with feed. Program the feed value prior to or not later than in the NC block of the cycle call. If a dwell time (option) was specified in seconds, it elapses once the drilling depth is reached. The spindle is stopped with the orientation angle and the retract path is traveled in the programmed retract axis and the drill axis to move out of the edge. Subsequently, the retract height and the center of the hole can be approached in rapid traverse.

Example

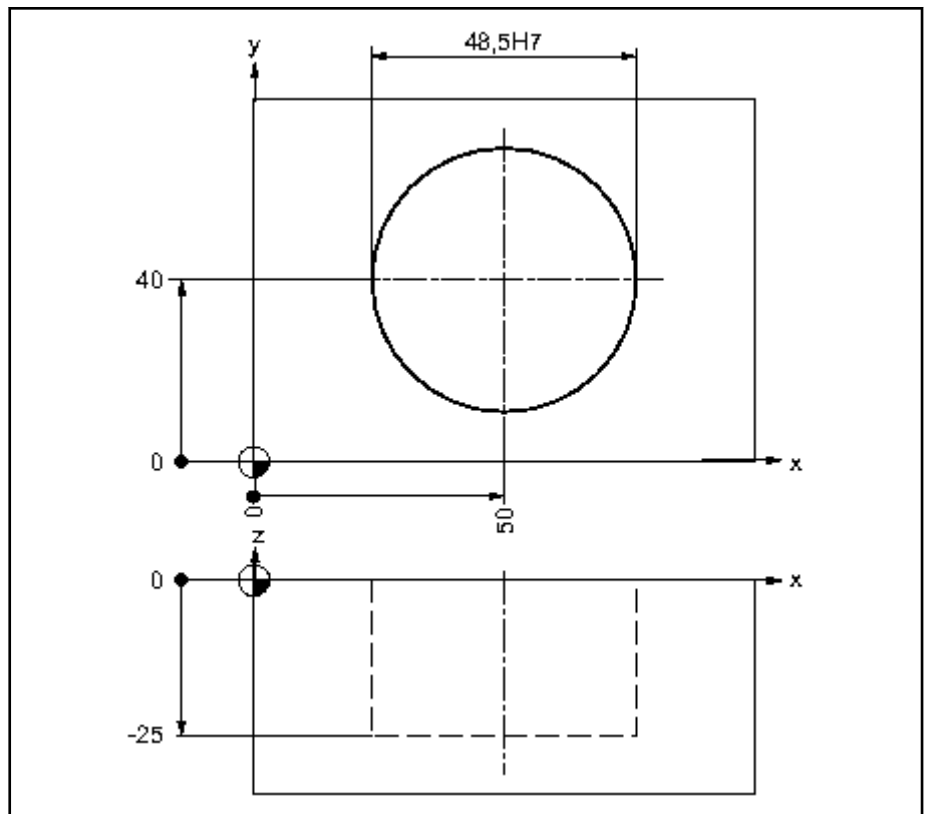


Fig. 5-19: Example of G86 Fine boring

Program for G86 Fine boring

```

N10 T1 M6                ; Tool change
N20 G17 G0 X25 Y40 Z100 S2500 M3 ; Prepositioning
N30 G86 (SL2, DT-25, DW1, OA0, SX X, RD0.1, RL10) F300
                           ; Cycle definition + feed
N40 X50 Y10              ; Cycle call
N50 G80 Z100 M5          ; Leave contour, delete modal call

```

5.8 G87 Back boring

Syntax G87(IX,SL,DT,DW,OA,SX,RD,RL)

Brief description The modal cycle G87 is suitable for back boring/backward countersinking by a tool with a fixed edge.

Overview

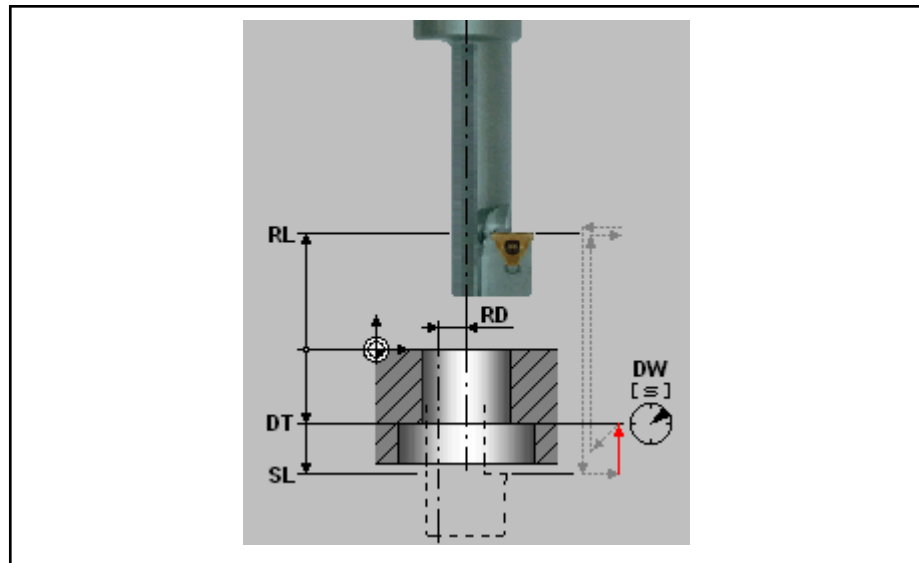


Fig. 5-20: Cycle G87 Back boring

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
IX	Drill axis	Optional	String	Depending on the active plane G17 = Z, G18 = X, G19 = Y
SL	Safety level	Mandatory	Absolute	
DT	Final depth	Mandatory	Absolute	
DW	Dwell time	Optional	≥ 0 Seconds	0
OA	Orientation angle	Optional	Absolute Degree	0
SX	Retract axis	Mandatory	String	
RD	Retract path	Mandatory	≥ 0 Incremental	
RL	Retraction height	Optional	Absolute	Initial position of drilling axis in front of the cycle

Tab. 5-8: Parameter call

Description

IX Drill axis

Name of drill axis

If IX is not specified and if no drilling axis is available, the main axis of the interpolation plane to the drilling axis, i.e. when turning in G18 becomes the z-axis

SL Safety level

Machining in feed starts at this absolute position

DT Final depth

Absolute depth position where the back boring ends

DW Dwell time

Period the driller "waits" at the lowest point

OA Orientation angle

Before leaving, the spindle is stopped on this angular position

SX Retract axis

The tool is retracted before leaving using this axis

RD Retract distance

The tool is axially and radially retracted by this value

RL Retract height

This absolute position is the final position of the cycle.

Special features

Note that a modal cycle is not executed with the cycle call during programming, but only after the next traversing command.

Procedure/processing

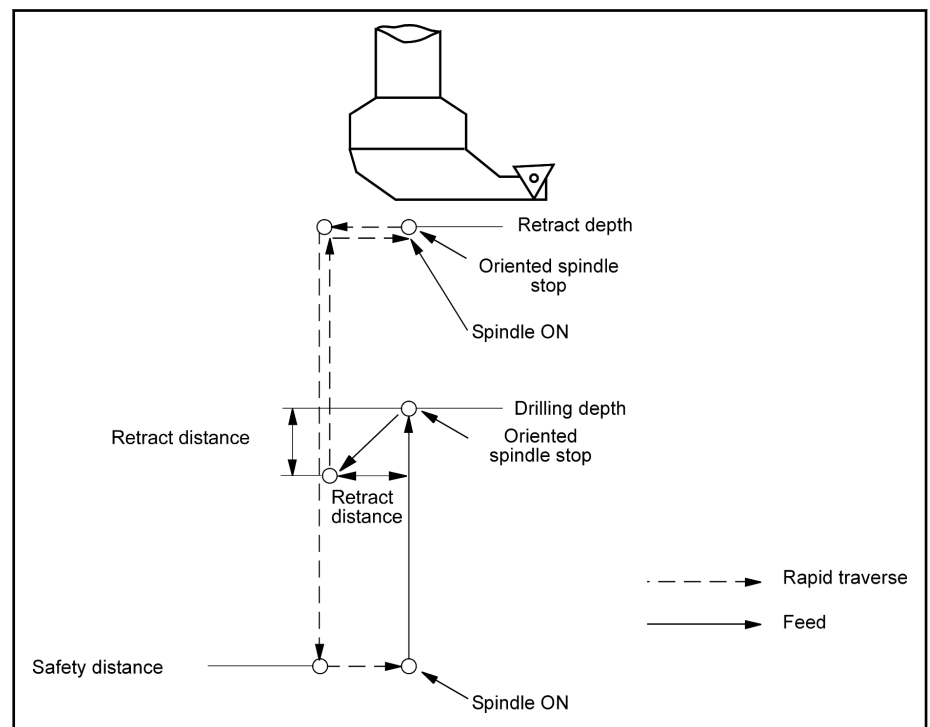


Fig. 5-21: Motion sequence G87 Back boring

Starting from the previously programmed position of the drill axis, the spindle is stopped with the orientation angle and the retract path is traveled in the programmed retract axis. Positioning at the safety level is implemented in rapid traverse and with oriented spindle. Subsequently, the retract path is traveled along the retract axis so that the spindle has now returned to the center of the bore. From there, it is traveled down to the final depth with feed. Program the feed value prior to or not later than in the NC block of the cycle call. If a dwell time (optional) is specified in seconds, it elapses once the final depth is reached. The spindle is stopped with the orientation angle and the retract path is traveled in the programmed retract axis and the drill axis to move out of the edge. Subsequently, the retract height and the center of the hole can be approached in rapid traverse.

Example

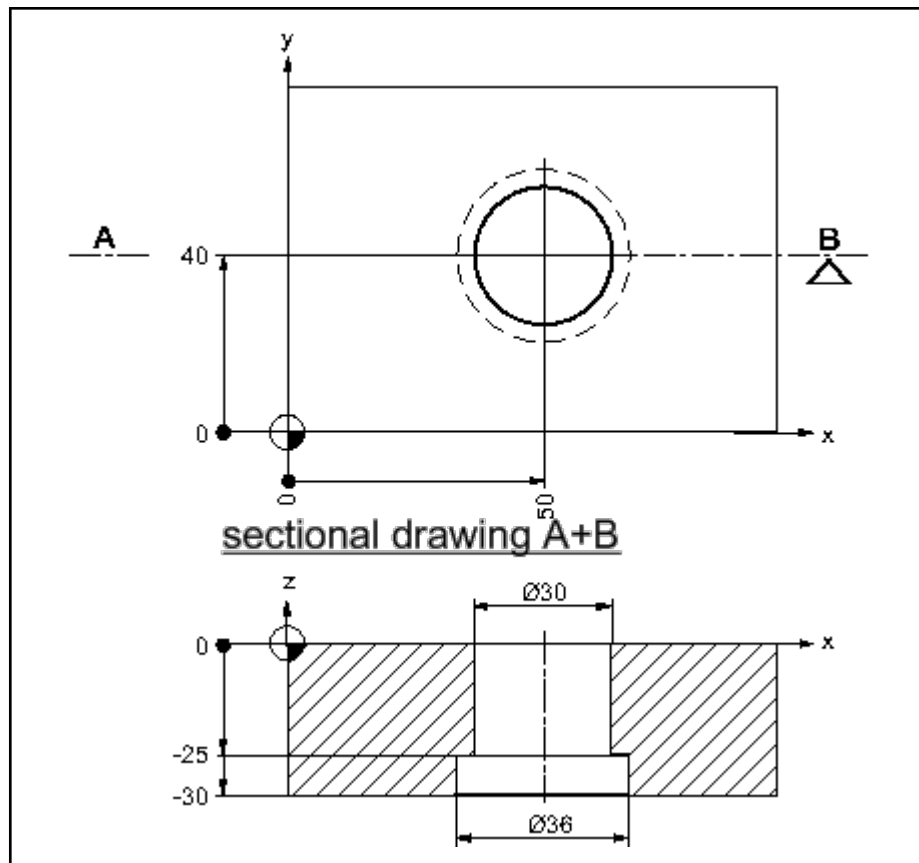


Fig. 5-22: Example of G87 Back boring
Program for G87 Back boring

```

N10 T1 M6 ; Tool change
N20 G17 G0 X25 Y40 Z100 S2500 M3 ; Prepositioning
N30 G87 (SL-32, DT-25, DW1, OA0, SX X, RD4, RL10) F300

N40 X50 Y10 ; Cycle call
N50 G80 Z100 M5 ; Leave contour, delete modal call

```

5.9 G88 Gun drilling

Syntax G88(IX,SL,DT,ST,MS,MFI,FD,FS,FF,C_ON,C_OF,RL,FP)

Brief description The modal cycle G88 is used to machine deep holes with a deep-hole drill cooled from the inside.

Overview

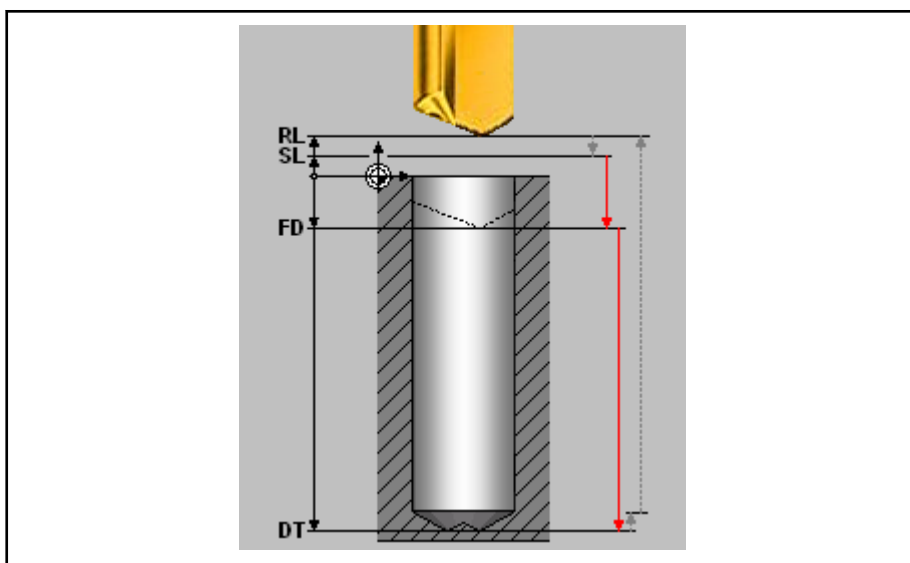


Fig. 5-23: Cycle G88 Gun drilling

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
IX	Drill axis	Optional	String	Depending on the active plane G17 = Z, G18 = X, G19 = Y
SL	Safety level	Mandatory	Absolute	
DT	Drilling depth	Mandatory	Absolute	
ST	Stabilization time	Optional	≥ 0 Seconds	0
MS	Drill speed	Mandatory	> 0	
MFI	Drill feed	Optional	> 0 Feed/min	Current feed
FD	Entry depth	Mandatory	Absolute	
FS	Entry speed	Mandatory	> 0	
FF	Entry feed	Mandatory	> 0 Feed/min	
C_ON	Coolant ON	Mandatory	String Auxiliary function	
C_OF	Coolant OFF	Mandatory	String Auxiliary function	

RL	Retraction height	Optional	Absolute	SL
FP	Start of drill speed and drill feed	Optional	50..100, Percent of MS/MFI	100

Tab. 5-9: Parameter call

Description	IX Drill axis
	Name of drill axis If IX is not specified and if no drilling axis is available, the main axis of the interpolation plane to the drilling axis, i.e. when turning in G18 becomes the z-axis
	SL Safety level
	Machining in feed starts at this absolute position
	DT Drilling depth
	Absolute position of the final depth to be reached
	ST Stabilization time
	The time the drill requires to stabilize
	MS drill speed & MFI drill feed
	Speed and feed after stabilization of the spot drill for drilling Specify MFI in feed/min
	FD Entry depth, FS Entry speed & FF Entry feed
	It travels with entry speed and entry feed up to the entry depth in the pilot bore Specify FF in feed/min
	C_ON coolant on or C_OF coolant off
	Switching on/off the coolant
	RL Retract height
	This absolute position is the final position of the cycle
	FP Start of drill speed and drill feed in %
	Reduced speed and feed at the beginning of the drilling. They increase longitudinally of the first 10% percent of the drill stroke linearly to MS and MFI
Special features	Note that a modal cycle is not executed with the cycle call during programming, but only after the next traversing command.

Procedure/processing

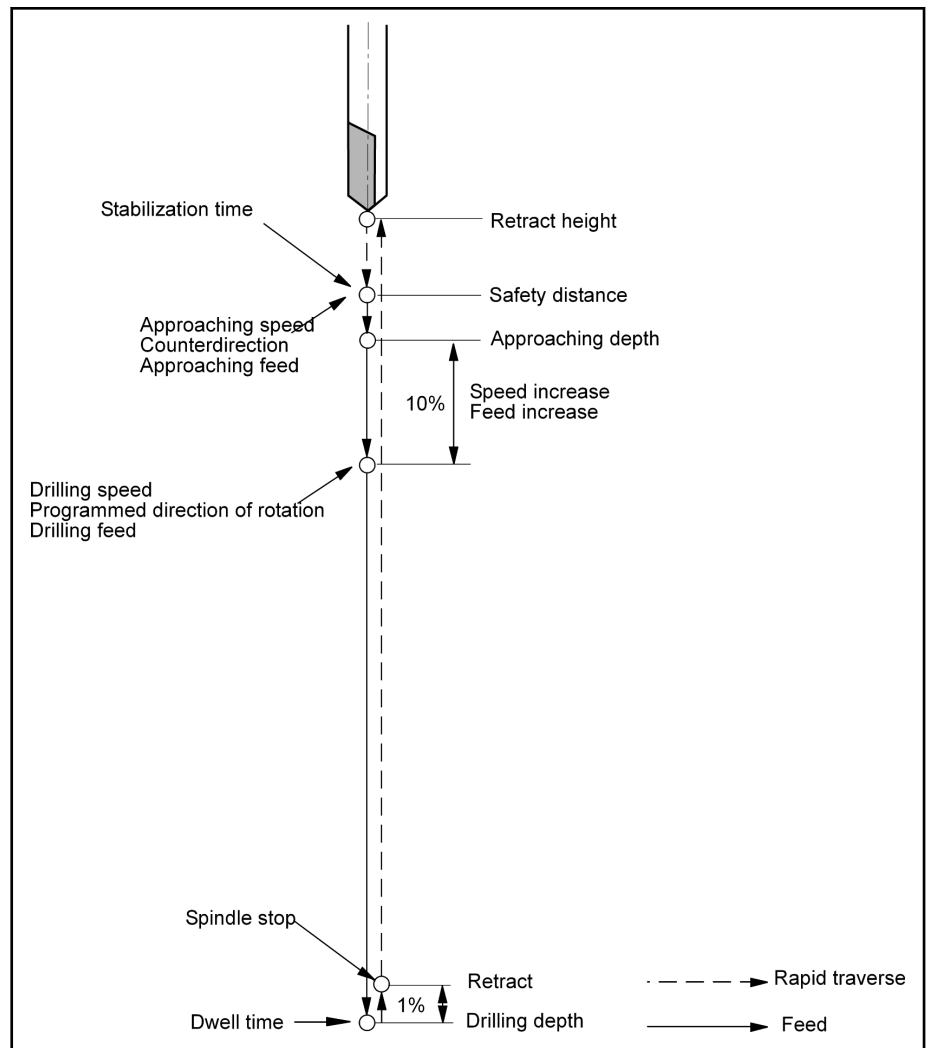


Fig. 5-24: Motion sequence G88 Gun drilling

From the last programmed position of the drill axis, it is positioned at the safety level in rapid traverse. If a stabilization time is specified, the drill is stabilized by switching on the coolant for this time with the spindle stationary and then the coolant is switched off again. The drill travels in opposing direction with entry speed and entry feed to the entry depth of the pilot bore. There, the coolant is switched on and the spindle is started in cutting direction at the starting drill speed resulted by FP. The starting speed and starting feed are now linearly increased to drill speed and drill feed while the first 10% of the drilling stroke is traveled. Once the drilling depth has been reached, the tool is retracted in rapid traverse by 1% of the drilling depth, the coolant is switched off and, with the spindle stationary, the retract height is approached in rapid traverse.

Example

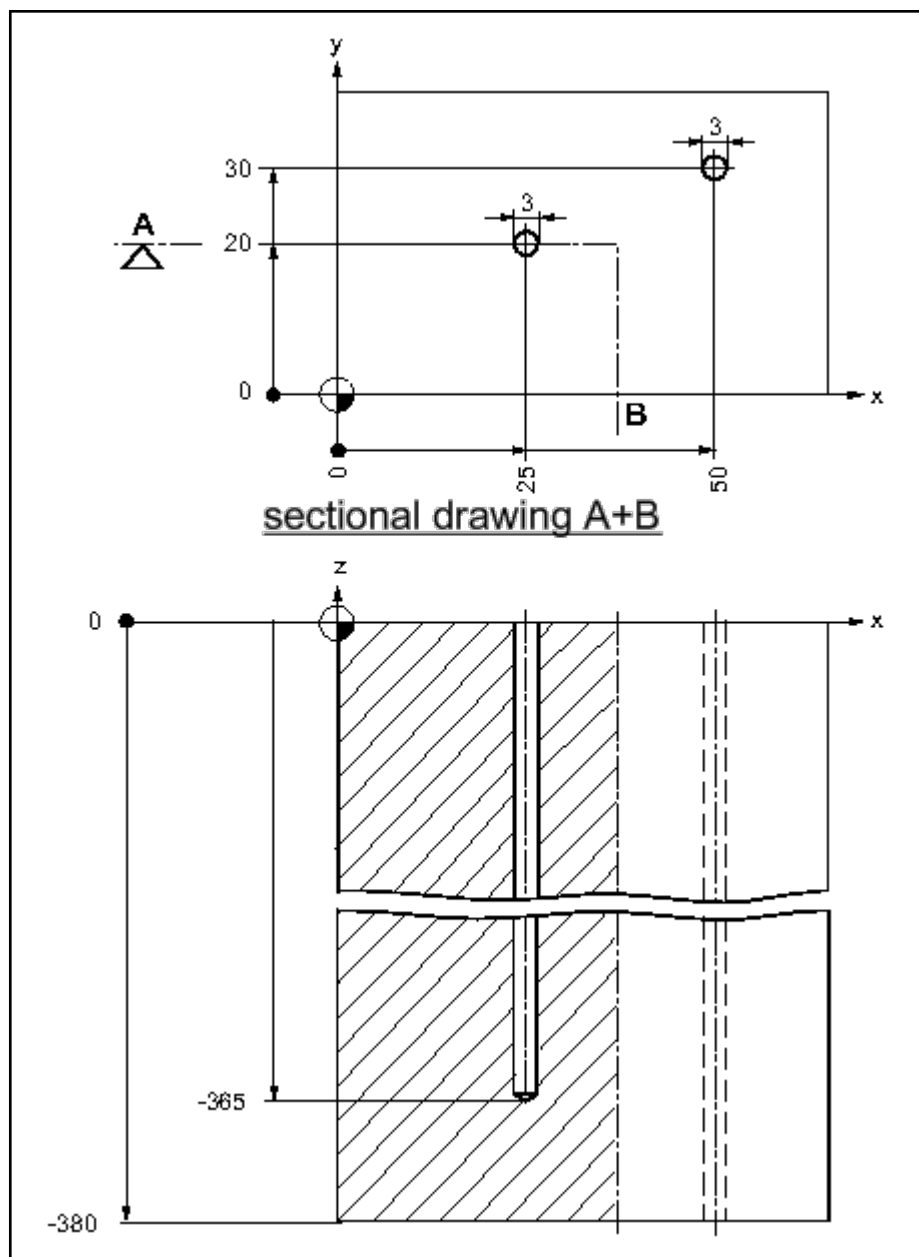


Fig. 5-25: Example

Program for G88 Gun drilling

```

N10 T1 M6 ; Tool change (step drill)
N20 G17 G0 X25 Y20 Z100 S0 M3 ; Prepositioning
N30 G88 (SL2, FD-15, DT-365, RL10, ST5, FS50, FF250, MS5000, MFI500, FP80, C_ON M8, C_OF M9)
N40 X25 Y20 ; 1st cycle call
N50 G88 (DT-383) ; New drilling depth
N60 X50 Y30 ; 2nd cycle call
N80 G80 Z100 M5 ; Leave contour, delete modal call

```

5.10 G89 Tapping with chip breaking

Syntax G89(IX,SL,DT,CD,RD,DW,TP,MS,RS,RL,LD)

Brief description Modal cycle G89 is suitable for machining tappings where chip braking and chip removal have to be ensured. A rigid tap and a position-controlled spindle mode are used.

Overview

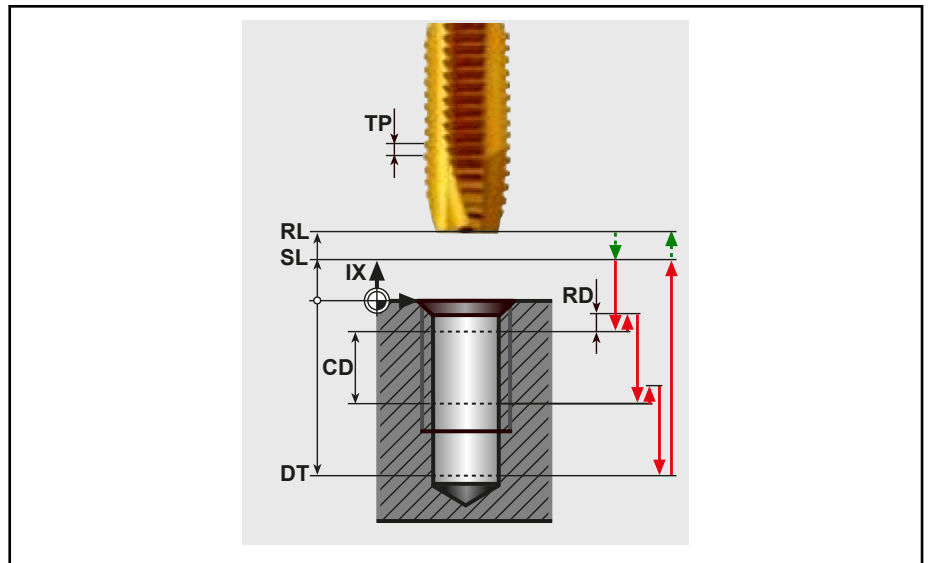


Fig. 5-26: Cycle G89 Tapping with chip breaking

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
IX	Drill axis	Optional	String	Depending on the active plane G17 = Z, G18 = X, G19 = Y
SL	Safety level	Mandatory	Absolute	
DT	Thread depth	Mandatory	Absolute	
CD	Infeed depth	Mandatory	> 0 Incremental	
DD	Approach distance/retracting path	Optional	>0 Incremental	TP amount
TP	Thread pitch	Mandatory	>0 right-hand thread <0 left-hand thread	
MS	Speed	Mandatory	>0	Current speed
RS	Speed for retracting	Optional	>0	MS
RL	Retraction height	Optional	Absolute	SL
LD	Chip breaking	Optional	Chip removal (0)/Chip breaking (1)	Chip removal (0)

Tab. 5-10: Parameter call

Description

IX Drill axis

Name of drill axis

If IX is not specified and if no drilling axis is available, the main axis of the interpolation plane to the drilling axis, i.e. when turning in G18 becomes the z-axis

SL Safety level

Tapping starts at this absolute position

DT Thread depth

Absolute position of the final depth to be reached

CD Infeed depth

Maximum infeed depth up to chip breaking

LD Retract distance

Distance the drill rotates backwards up to chip breaking

TP Thread pitch

Pitch of thread

MS speed

Speed for thread cutting

RS Speed for retracting

Speed to leave the thread

RL Retract height

This absolute position is the final position of the cycle

LD Chip breaking

Chip removal and chip breaking

Special features The reference spindle should be programmed before the cycle is called with TSP.

A left-hand thread can be created by specifying a negative pitch and a right-hand thread by specifying a positive pitch.

Procedure/processing

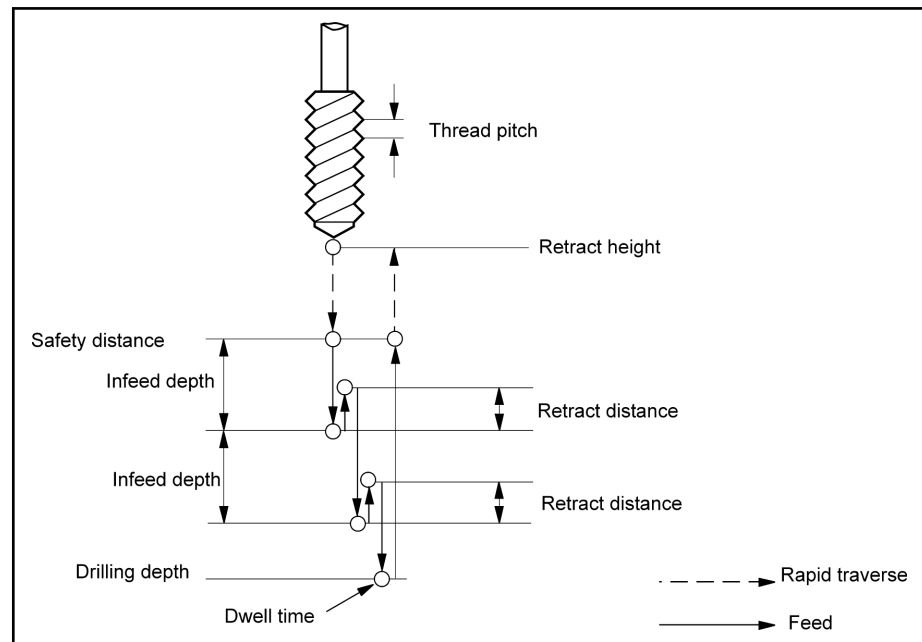


Fig. 5-27: Motion sequence G89 Tapping with chip breaking

From the last programmed position of the drill axis, it is positioned at the safety level in rapid traverse. From there, it travels down the infeed depth at

the previously programmed speed and at the specified thread pitch. If "Chip removal (0)" was programmed in LD, it is positioned to the safety level at retracting speed. Subsequently, it is traveled at the same speed to the respective infeed depth reduced by the starting distance. If chip breaking (1) was selected in LD, the tool is retracted by the retract distance at retracting speed. The drilling steps including counterclockwise turning is repeated until the thread depth is reached. The spindle returns in opposing direction to the safety level at the same speed, or if specified, at the speed for retraction. Finally, the retract height is approached in rapid traverse.

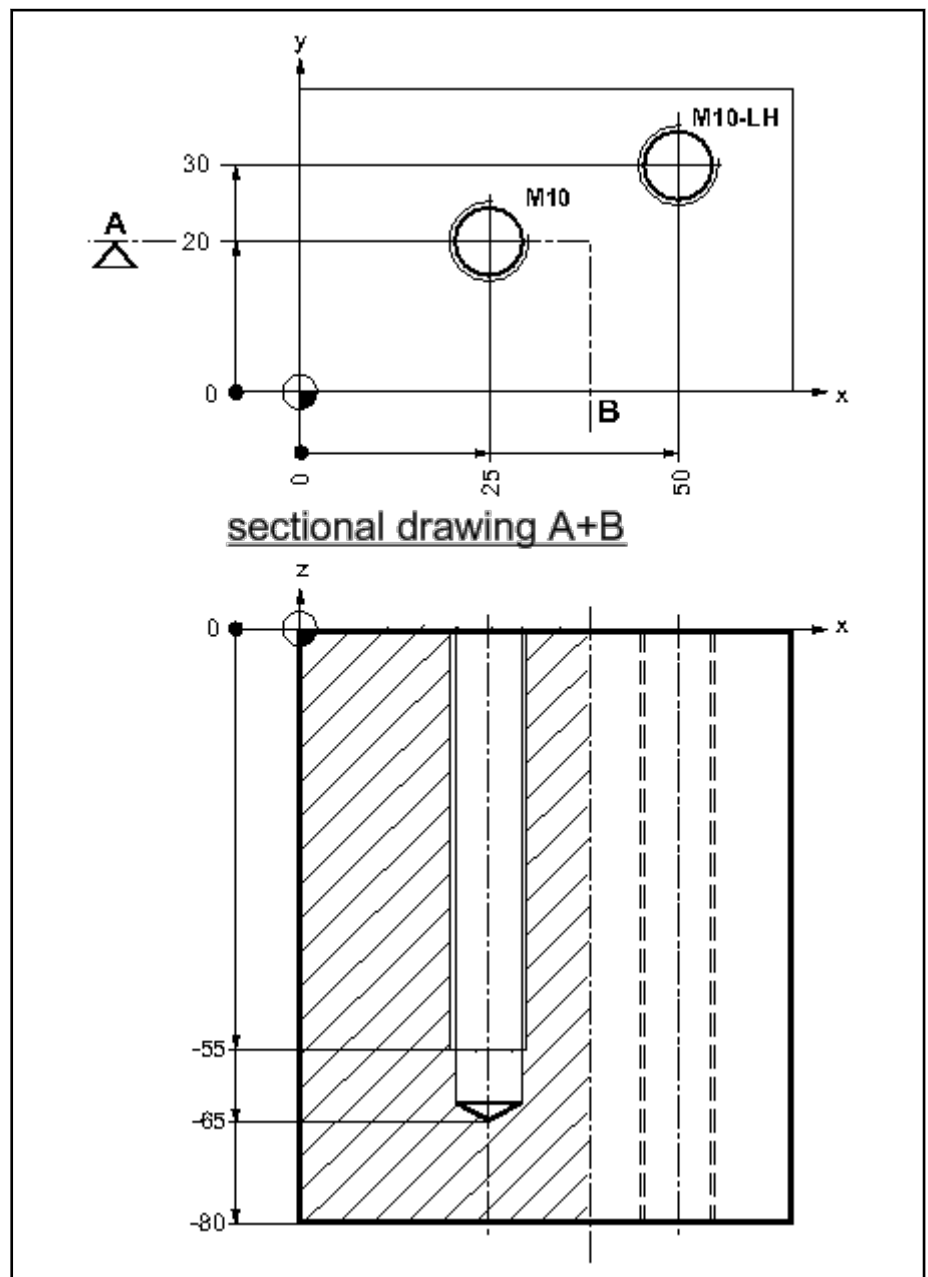
Example

Fig. 5-28: Example of G89 Tapping with chip breaking
Program for G89 Tapping with chip breaking

```

N10 T1 M6                ; tool change
N20 G17 G0 X25 Y20 Z100  ; prepositioning
N30 TSP (GRP1)           ; reference spindle group 1
N40 G89 (SL2, DT-59, CD15, RD1, TP1.5, MS300, RS600, RL10, LD1)

```

Drilling cycles

```

; cycle definition
N50 X25 Y20 ; 1. cycle call
N60 G89 (DT-85) ; new thread depth
N70 X50 Y30 ; 2. cycle call
N80 G80 Z100 M5 ; leave contour, delete modal call
```

6 Hole patterns

6.1 General information

Modal cycles can be performed simply with the hole patterns described in the following.

6.2 G111 Hole circle, rotary axis

Syntax G111(RX,NH,BA,LA)

Brief description The cycle positions a rotary axis repeated in the same increments. At each angular position, a previously programmed modal is executed. Thus, a partial circle with the radius of the radial initial position results.

Overview figure

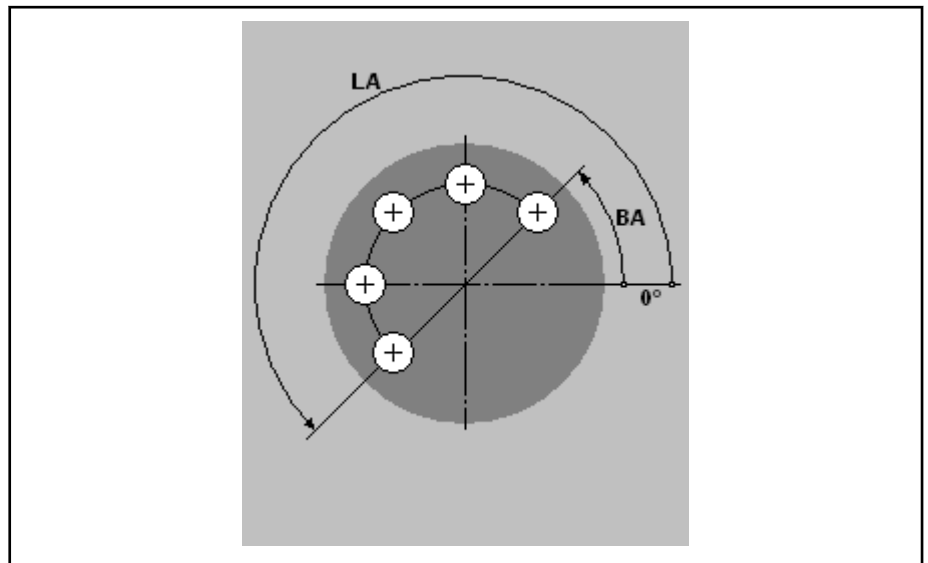


Fig. 6-1: Cycle "G111 Hole Pattern, Rotary Axis"

Parameters

Parameters	Name	Properties	Values	Pre-assignment
RX	Rotary axis	Mandatory	String	
NH	Number of drillings	Mandatory	Integer ≥1	
BA	Starting angle	Optional	Degrees	0
LA	End angle	Optional	Degrees	Default setting is the uniform distribution across the full circle.

Tab. 6-1: Parameter call

Description

Special features



Parameter change. Parameter "BA" for the starting angle was renamed and called "FA" up to MTX 10VRS. Calls with old parameter names are compatible with the new ones. But if changes were made via the input helps of the NC program editor, the respective parameters have to be named manually.

Hole patterns

Procedure/processing

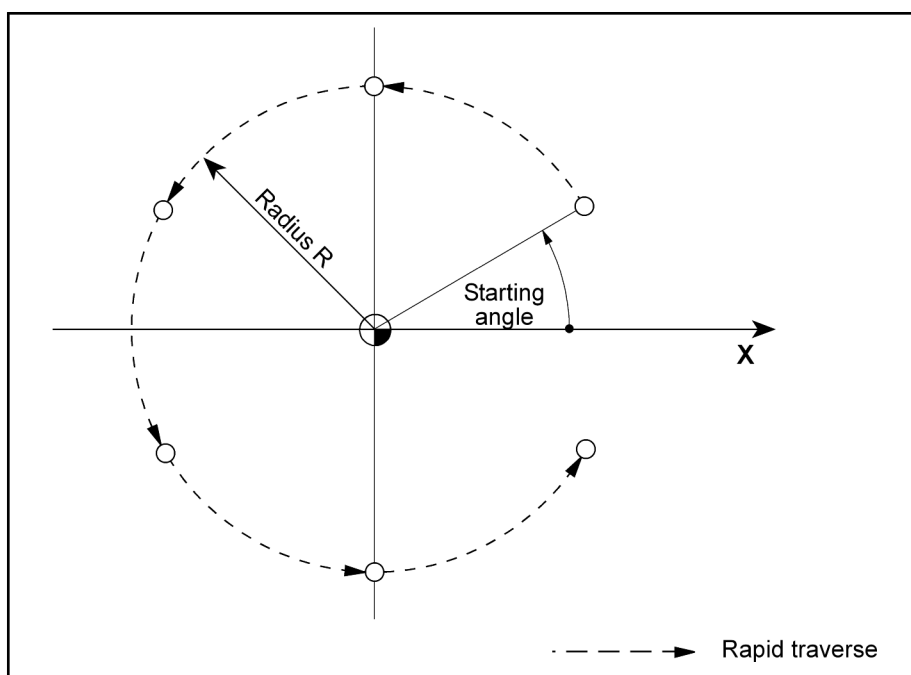


Fig. 6-2: Motion sequence "G111 Hole pattern, totary axis" (full circle)

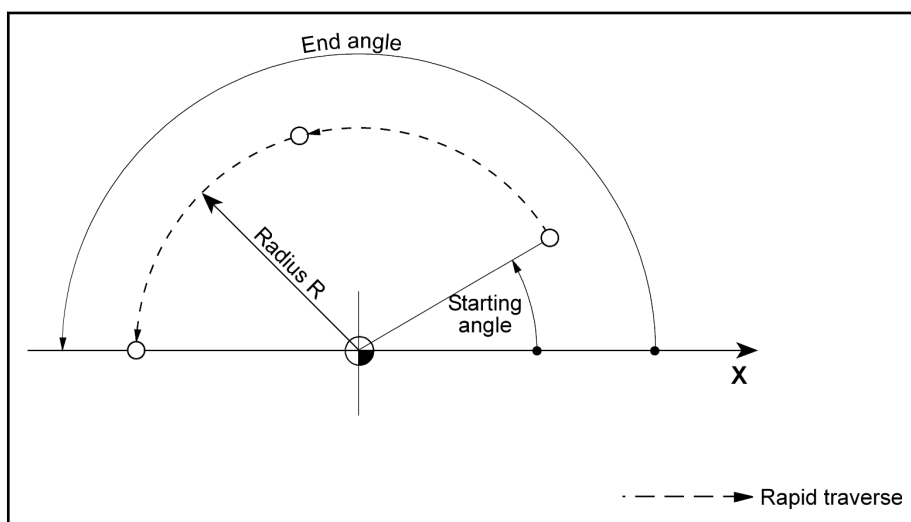


Fig. 6-3: Motion sequence "G111 Hole pattern, rotary axis" (partial circle)

The modal drilling or milling cycle programmed before the cycle call is processed at each position resulted from the input parameters. If no end angle is specified, the angular positions are equally distributed on a full circle and approached in positive direction of the rotary axis. If the end angle is specified, the distribution is different. If the starting angle is bigger than the end angle, it is switched to negative direction.

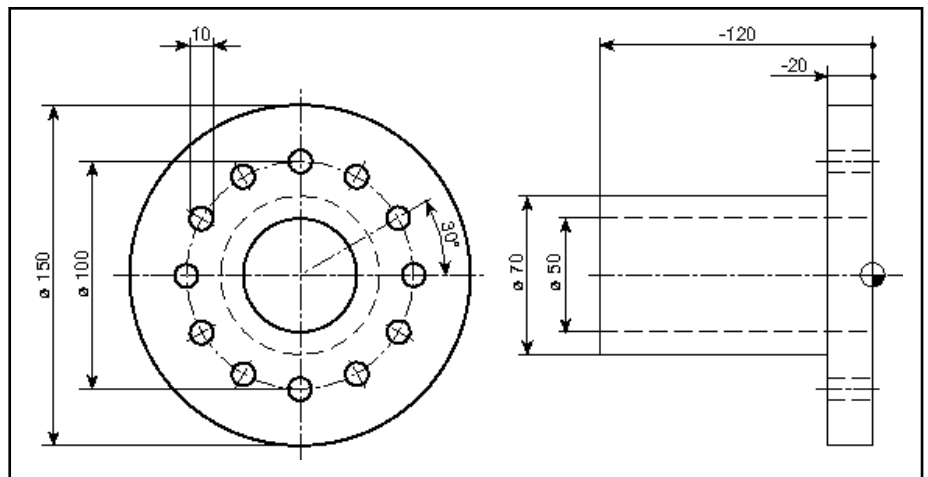
Example 1

Fig. 6-4: Example of G111 Hole Circle, Rotary Axis (full circle)

Program for G111 Hole circle, rotary axis (full circle)

```

N10 T1 M6 ; Tool change
N20 G17 G0 X50 Y50 Z100 S2500 M3 ; Prepositioning
N30 G81 (IX=Z,SL2,DT-22,DW1,RL10) F300 ; Cycle definition + feed
N40 G111 (RX=C,NH12,BA0) ; Cycle definition
N50 G80 Z100 M5 ; Leave contour, delete modal call

```

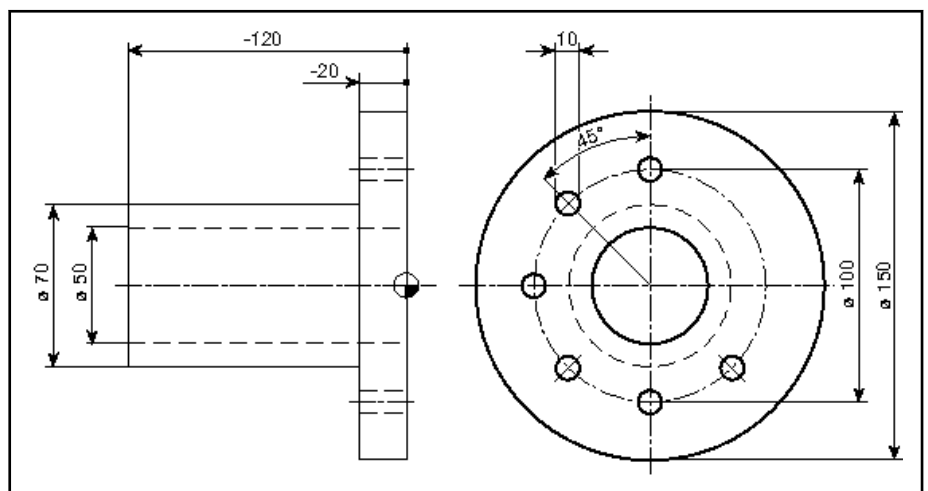
Example 2:

Fig. 6-5: Example of G111 Hole Circle, Rotary Axis (partial circle)

Program for G111 Hole circle, rotary axis (partial circle)

```

N10 T1 M6 ; Tool change
N20 G17 G0 X50 Y50 Z100 S2500 M3 ; Prepositioning
N30 G81 (IX=Z,SL2,DT-22,DW1,RL10) F300 ; Cycle definition + feed
N40 G111 (RX=C,NH6,BA90,LA315) ; Cycle definition
N50 G80 Z100 M5 ; Leave contour, delete modal call

```

6.3 G112 Hole circle, main spindle

Syntax G112(SP,SO,NH,BA,LA,B_ON,B_OF)

Brief description The cycle orients a spindle or a spindle group repeated in the same increments. At each angular position, a previously programmed modal is executed. Thus, a partial circle with the radius of the radial initial position results.

Hole patterns

Overview figure

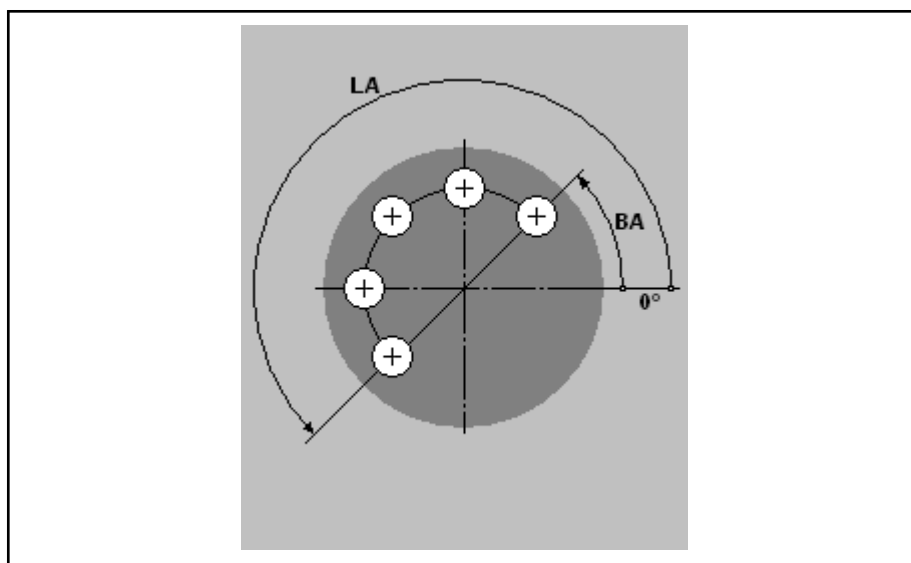


Fig. 6-6: Cycle "G112 Hole circle, main spindle"

Parameters

Parameters	Name	Properties	Values	Pre-assignment
SP	Spindle identifier	Mandatory	String	
SO	Spindle orientation	Mandatory Auxiliary function	String	
NH	Number of drillings	Mandatory	Integer ≥ 1	
BA	Starting angle	Optional Degrees		0
LA	End angle	Optional Degrees		Default setting is the uniform distribution across the full circle.
B_ON	Spindle brake ON	Optional Auxiliary function	String	
B_OF	Spindle brake OFF	Optional Auxiliary function	String	

Tab. 6-2: Parameter call

Description



Parameter change. Parameter "BA" for the starting angle was renamed and called "FA" up to MTX 10VRS. Calls with old parameter names are compatible with the new ones. But if changes were made via the input helps of the NC program editor, the respective parameters have to be named manually.

Special features

Procedure/processing

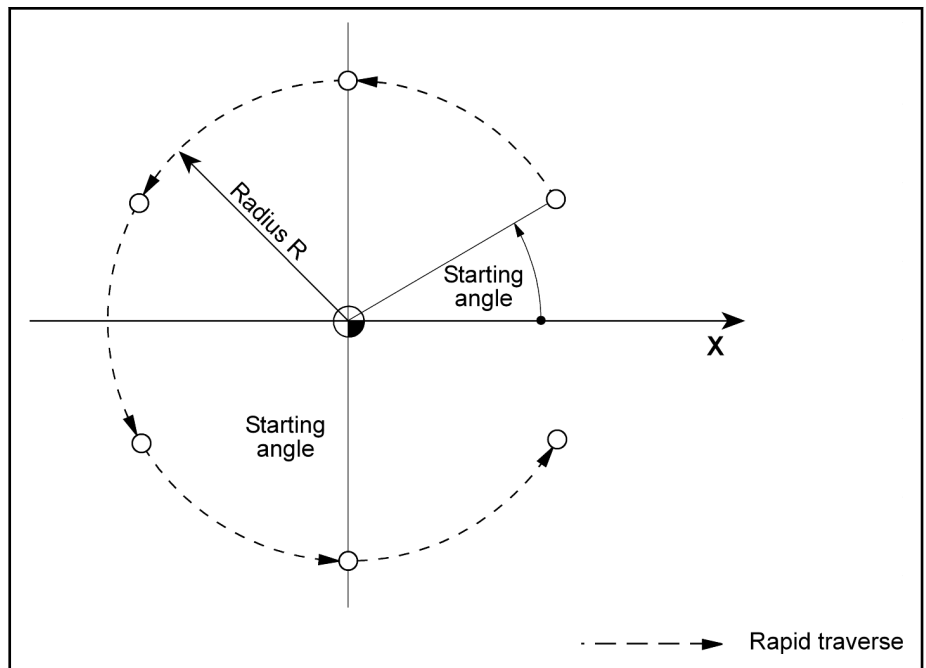


Fig. 6-7: Motion sequence "G112 Hole circle, main spindle" (full circle)

The modal drilling or milling cycle programmed before the cycle call is processed at each spindle position resulted from the input parameters. If no end angle is specified, the angular positions are equally distributed on a full circle and approached in positive direction of the spindle. If the end angle is specified, the distribution is different. If the starting angle is bigger than the end angle, it is switched to negative direction. It is possible for the spindle brake to clamp the spindle during machining using the optional input of the auxiliary functions.

Example 1

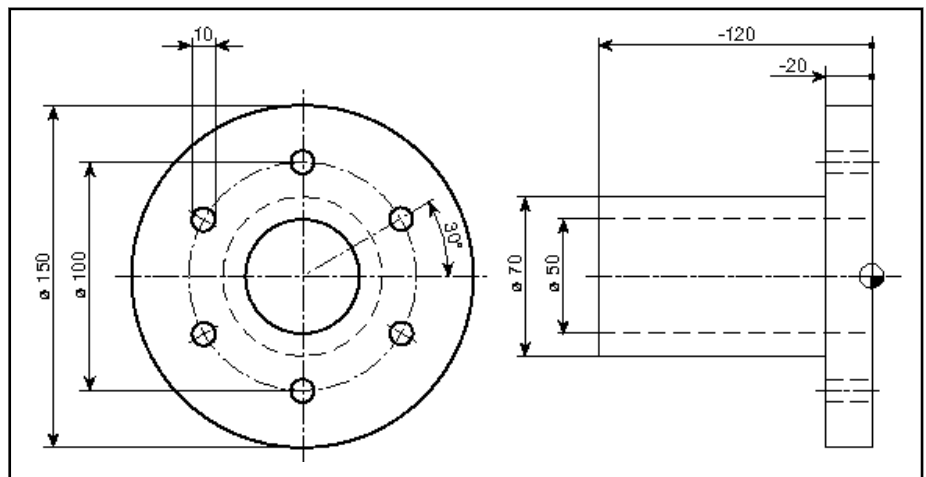


Fig. 6-8: Example of G112 Hole circle, main spindle (full circle)

Program for G112 Hole circle, main spindle (full circle)

```

N10 T1 M6 ; Tool change
N20 G18 G0 X50 Y50 Z100 M5 ; Prepositioning
N30 G81 (IX=Z,SL2,DT-22,DW1,RL10) F300 ; Cycle definition + feed
N40 G112 (SP=S,SO=M19,NH6,BA30,B_ON=M10,B_OF=M11) ; Cycle definition
N50 G80 Z100 M5 ; Leave contour, delete modal call

```

Hole patterns

Example 2:

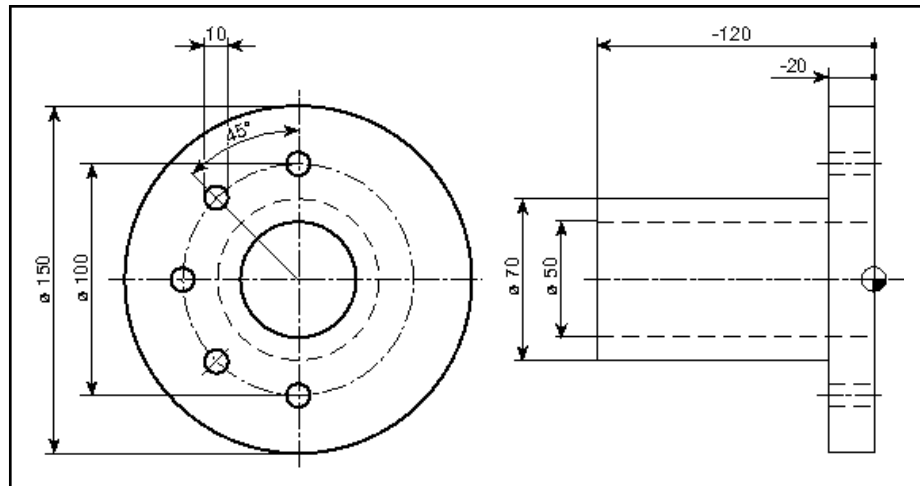


Fig. 6-9: Example of G112 Hole circle, main spindle (partial circle)
 Program for G112 Hole circle, main spindle (partial circle)

```

N10 T1 M6 ; Tool change
N20 G18 G0 X50 Y50 Z100 M5 ; Prepositioning
N30 G81 (IX=Z, SL2, DT-22, DW1, RL10) F300 ; Cycle definition + feed
N40 G112 (SP=S, SO=M19, NH5, BA90, LA270, B_ON=M10, B_OF=M11) ; Cycle definition
N50 G80 Z100 M5 ; Leave contour, delete modal call
  
```

6.4 G113 Hole circle, plane

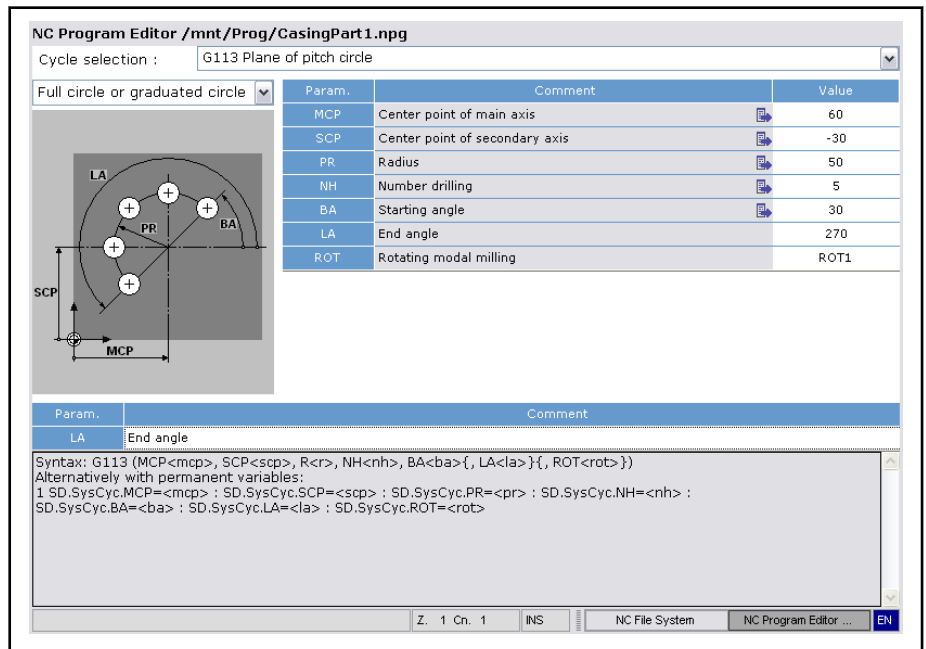


Fig. 6-10: G113 Hole circle, plane

G113		Hole circle, plane	G113(MCP,SCP,PR,NH,BA,LA,ROT)		
MCP	SD.SysCyc.MCP	Center point: master axis	Real	Mandatory	
SCP	SD.SysCyc.SCP	Center point: slave axis	Real	Mandatory	
R	SD.SysCyc.PR	Radius	Real	Mandatory	>0
NH	SD.SysCyc.NH	Number of drillings	Integer	Mandatory	>=1
BA	SD.SysCyc.BA	Starting angle	Real	Optional	Degrees 0 is preset
LA	SD.SysCyc.LA	End angle	Real	Optional	Degrees Default setting is the uniform distribution across the full circle.
ROT	SD.SysCyc.ROT	Rotating modal milling	Integer	Optional	1 Parallel shift 2 Rotation by incremental angle 1 is the default setting

Hole patterns

Sequence diagram:

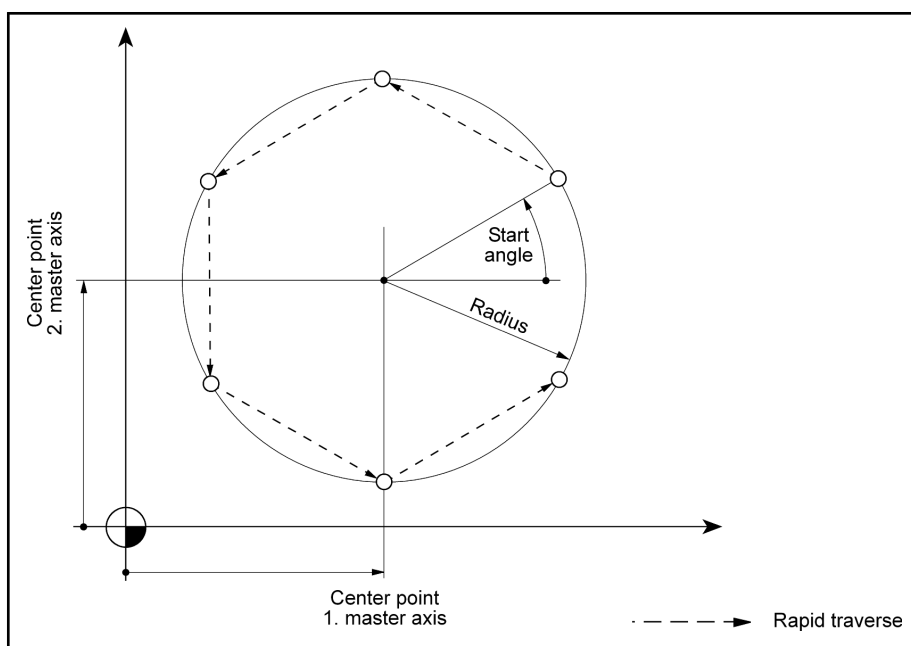


Fig. 6-11: Motion sequence "G113 Hole circle, plane" (full circle)

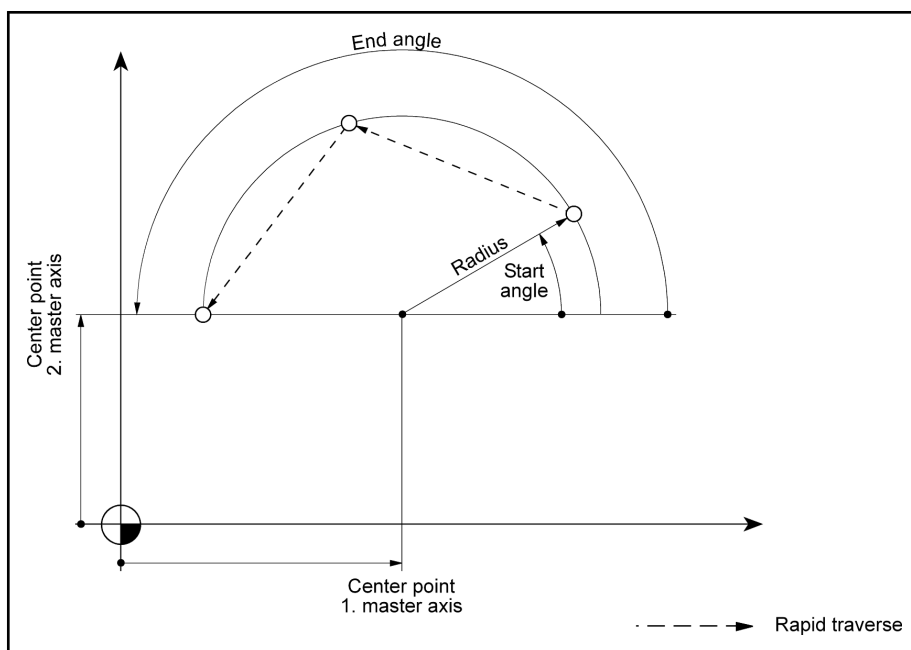


Fig. 6-12: Motion sequence "G113 Hole circle, plane" (partial circle)

Sequence: The modal drilling cycle programmed before the cycle call is processed in the interpolation plane at each position resulted from the input parameters. If no end angle is specified, a full circle is assumed and it is always traveled in positive direction. By entering the end angle, only a partial circle can be manufactured. If the starting angle is larger than the end angle, it is traveled in negative direction. Thus, a hole circle can also be manufactured in negative direction.



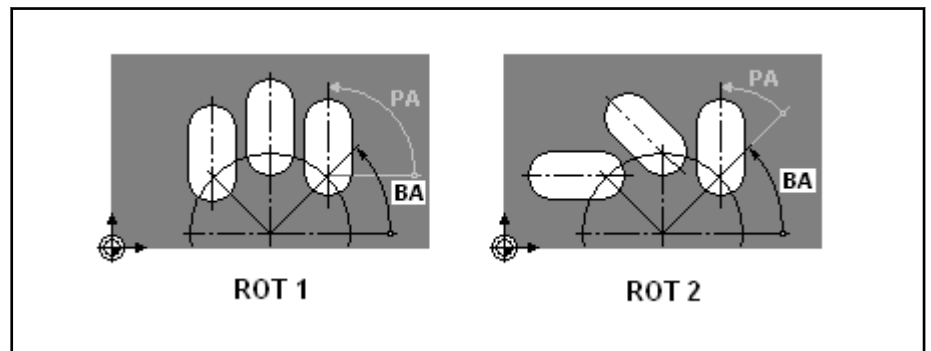
Parameter change. As compared with MTX 10VRS, some of the parameters were renamed:

- Parameter "FA" is now "BA".
- Parameter "R" is now "PR".

Calls with old parameter names are compatible with the new ones. But if changes were made via the input helps of the NC program editor, the respective parameters have to be renamed manually.

Diameter dimension programming (DIA instruction) is only possible for the x-axis.

Optionally, modal milling cycles can be rotated by the incremental angle. Rotating is triggered with ROT 2 and applies to all default milling cycles except for G267 Face milling, circle and G272 Circular pocket milling. The following figure illustrates the effect of rotation for the cycle "G274 Straight slot milling".



- ROT 1** The milling patterns are arranged at the point positions moved in parallel. The orientation angle of the milling cycles (here: PA) are absolute angles in relation to the master axis of the plane.
- ROT 2** The milling patterns are rotated along with the incremental angle. The orientation angles of the milling cycles are an offset of the angular pitch at each position.

Fig. 6-13: Mode of action of the parameter ROT

Hole patterns

Example 1:

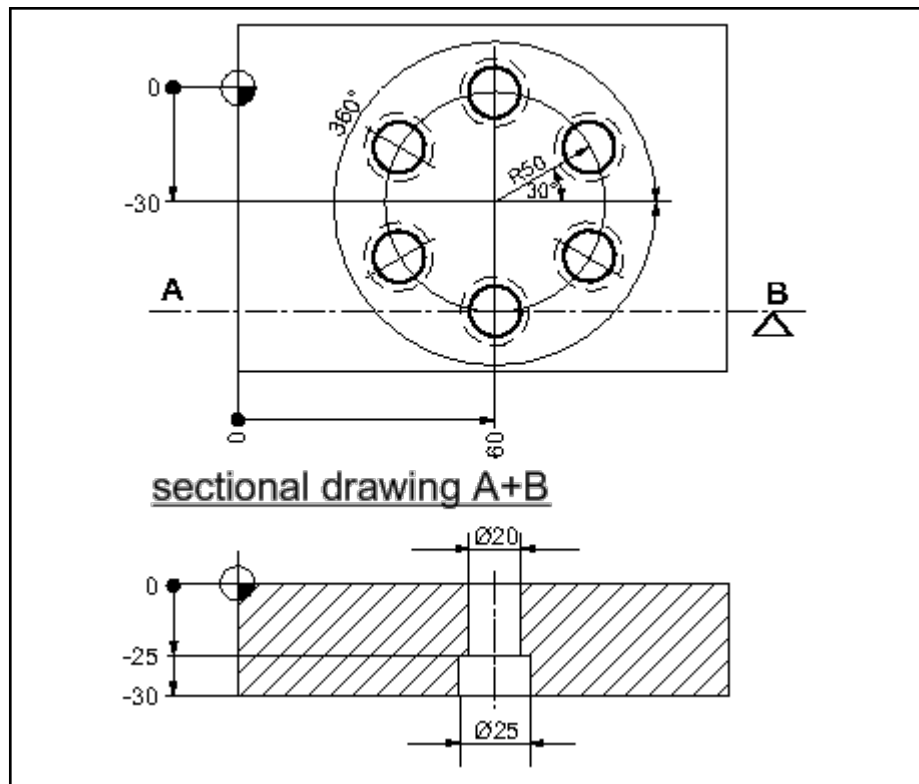


Fig. 6-14: Example of G113 Hole circle, plane (full circle)

Program for G113 Hole circle, plane (full circle)

```

N10 T1 M6 ; Tool change
N20 G17 G0 X60 Y-30 Z100 S2500 M3 ; Prepositioning
N30 G87 (IX=Z, SL-32, DT-25, DW1, OA0, SX=X, RD1, RL10) F300 ; Cycle definition + feed
N40 G113 (MCP60, SCP-30, PR50, NH6, BA30) ; Cycle definition
N50 G80 Z100 M5 ; Leave contour, delete modal call

```


Example 2:

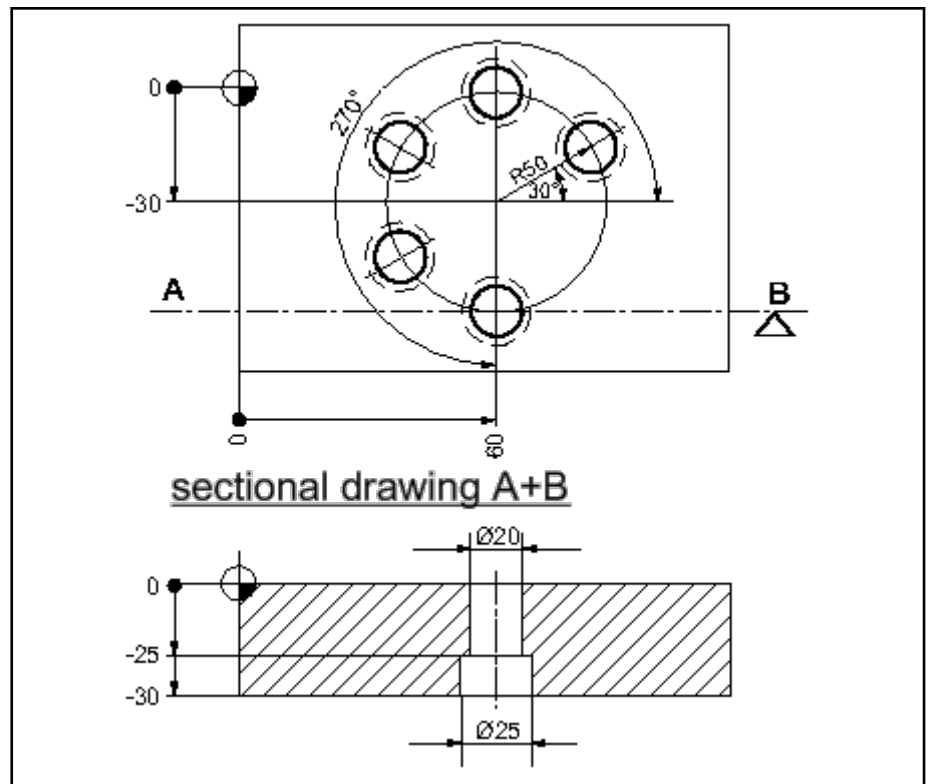


Fig. 6-15: Example of G113 Hole circle, plane (partial circle)

Program for G113 Hole circle, plane (partial circle)

```

N10 T1 M6 ; Tool change
N20 G17 G0 X60 Y-30 Z100 S2500 M3 ; Prepositioning
N30 G87 (IX=Z,SL-32,DT-25,DW1,OA0,SX=X,RD1,RL10) F300 ; Cycle definition + feed
N40 G113 (MCP60,SCP-30,PR50,NH5,BA30,LA270) ; Cycle definition
N50 G80 Z100 M5 ; Leave contour, delete modal call

```

6.5 G114 Hole pattern, line

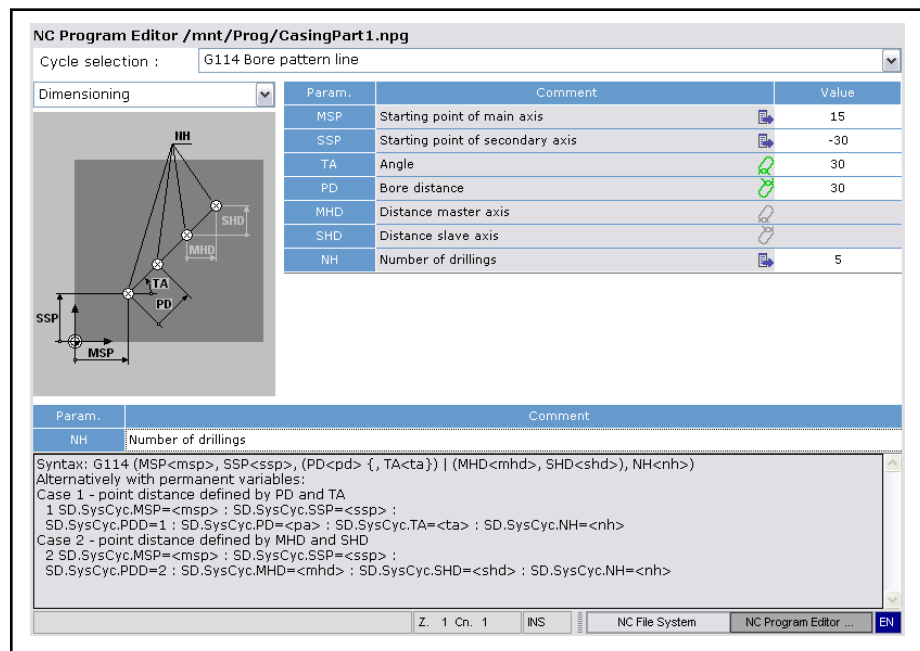


Fig. 6-16: G114 Hole pattern, line

G114		Hole pattern, line	G114(MSP,SSP,TA,PD,MHD,SHD,NH)		
MSP	SD.SysCyc.MSP	Starting point: master axis	Real	Mandatory	
SSP	SD.SysCyc.SSP	Starting point: slave axis	Real	Mandatory	
TA	SD.SysCyc.TA	Angle	Real	Optional Specify in pairs with PD and alternatively to MHD and SHD	Degrees 0 is preset
PD	SD.SysCyc.PD	Hole distance	Real	Specify in pairs with TA and alternatively to MHD and SHD	<>0
MHD	SD.Sy-sCyc.MHD	Distance of master axis	Real	Specify in pairs with SHD and alternatively to TA and PD	
SHD	SD.SysCyc.SHD	Distance of slave axis	Real	Specify in pairs with MHD and alternatively to TA and PD	
NH	SD.SysCyc.NH	Number of drillings	Integer	Mandatory	>=1
PDD	SD.SysCyc.PDD	Dimensioning type	Integer	Only for SD parameters	1 - Dimensioning TA and PD 2 - Dimensioning MHD and SHD

Sequence diagram:

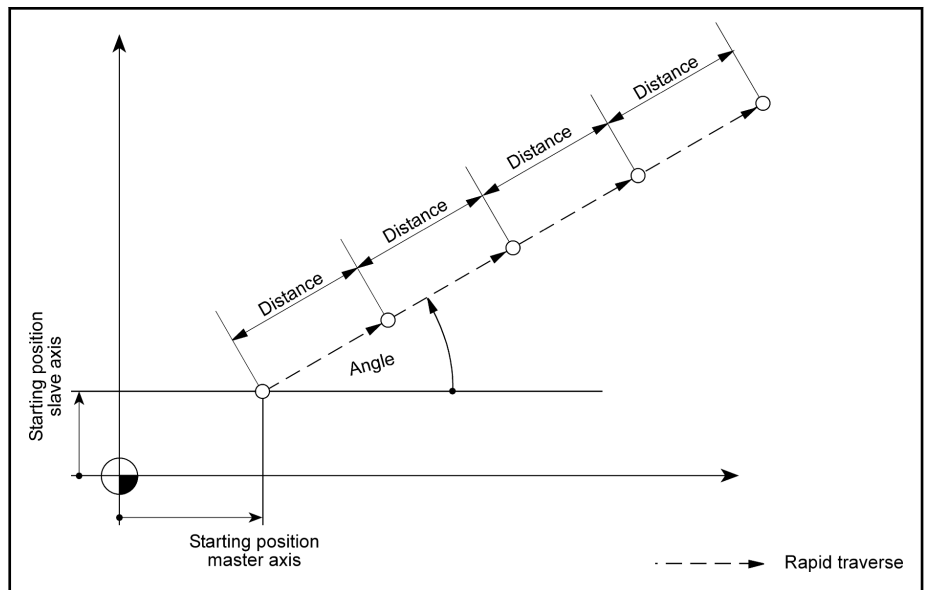


Fig. 6-17: Motion sequence "G114 Hole pattern, line" (hole distance and angle)

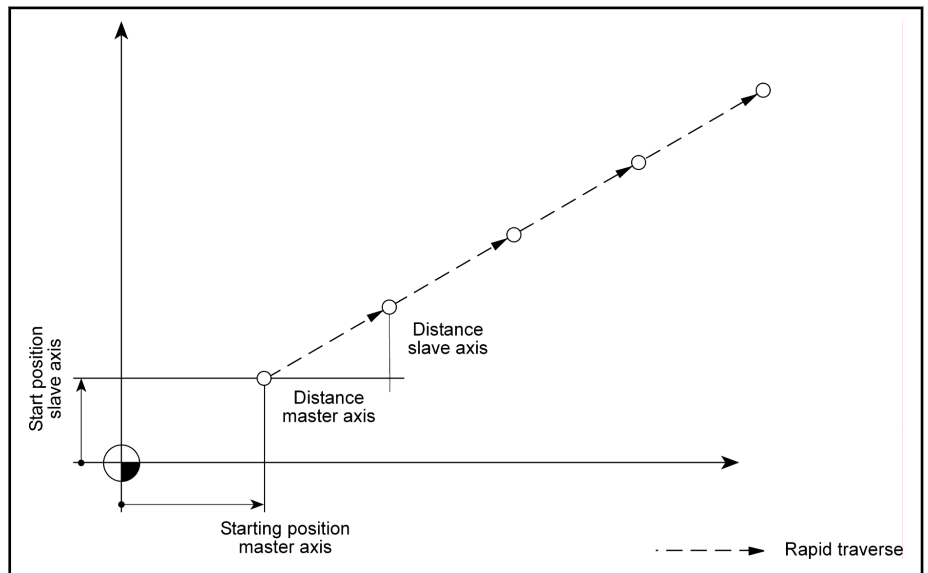


Fig. 6-18: Motion sequence for G114 Hole pattern, line (axis-related hole distance)

Sequence: The modal drilling cycle programmed before the cycle call is processed in the interpolation plane at each position resulted from the input parameters. Define the cycle using the parameters Angle (TA) and Hole Distance (PD) or the axis-related distances (MHD and SHD). To change the machining direction, specify the last drilling position in both cases as starting point (MSP and SSP). For dimensioning type (PA and PD), increase the angle by 180°. For dimensioning type (MHD and SHD), negate the signs of the parameters "Distance Master Axis" and "Distance Slave Axis". When using SD parameters, the dimensioning type is specified by the SD variable "SD.SysCyc.PDD".



Parameter change. The parameter "TA" for the starting angle was renamed and called "PA" up to the MTX 10VRS. Calls with old parameter names are compatible with the new ones. But if changes were made via the input helps of the NC program editor, the respective parameters have to be renamed manually.

Hole patterns

Diameter dimension programming (DIA instruction) is only possible for the x-axis.

Example 1:

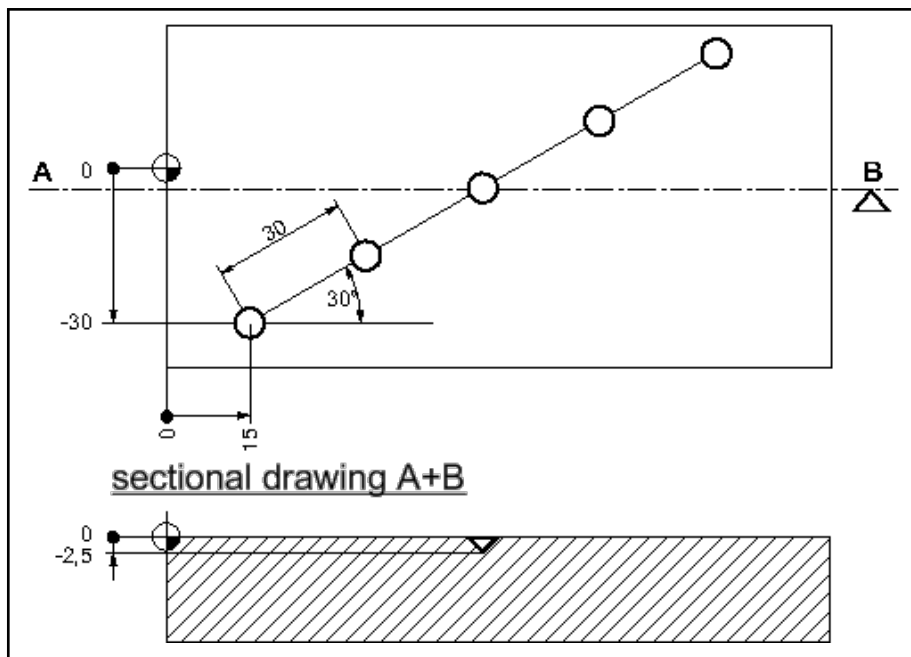


Fig. 6-19: Example of G114 Hole pattern, line (hole distance and angle)
G114 Hole pattern, line (hole distance and angle)

```

N10 T1 M6 ; Tool change
N20 G17 G0 X15 Y-30 Z100 S2500 M3 ; Prepositioning
N30 G81(IX=Z,SL2,DT-2.5,DW1,RL10) F300 ; Cycle definition + feed
N40 G114 (MSP15,SSP-30,TA30,PD30,NH5) ; Cycle definition
N50 G80 Z100 M5 ; Leave contour, delete modal call

```

Example 2:

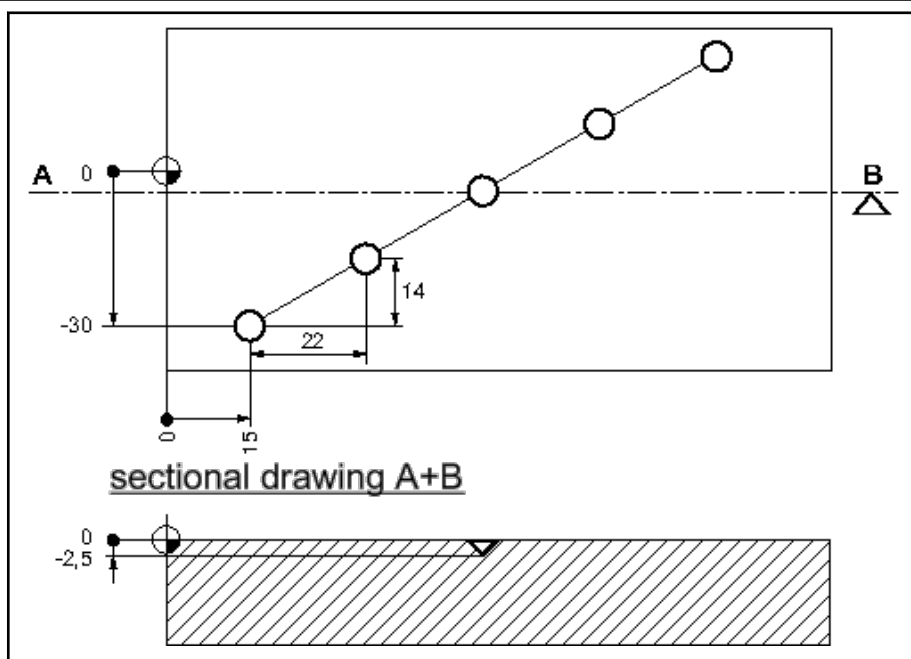


Fig. 6-20: Example of G114 Hole pattern, line (axis-related hole distance)
G114 Hole pattern, line (axis-related hole distance)

```

N10 T1 M6 ; Tool change
N20 G17 G0 X15 Y-30 Z100 S2500 M3 ; Prepositioning

```

```
N30 G81 (IX=Z,SL2,DT-2.5,DW1,RL10) F300 ; Cycle definition + feed
N40 G114 (MSP15,SSP-30,MHD22,SHD14,NH5) ; Cycle definition
N50 G80 Z100 M5 ; Leave contour, delete modal call
```

6.6 G115 Hole pattern, matrix

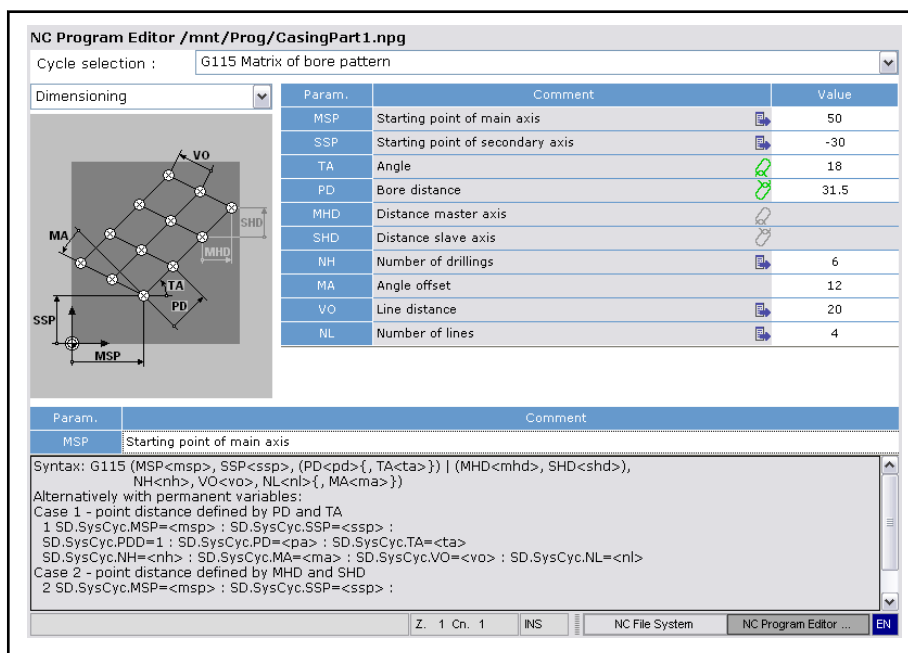


Fig. 6-21: G115 Hole pattern, matrix

G115		Hole pattern, matrix	G115(MSP,SSP,TA,PD,MHD,SHD,NH,MA,VO,NL)		
MSP	SD.SysCyc.MSP	Starting point: master axis	Real	Mandatory	
SSP	SD.SysCyc.SSP	Starting point: slave axis	Real	Mandatory	
TA	SD.SysCyc.TA	Angle	Real	Optional	Degrees 0 is preset
PD	SD.SysCyc.PD	Hole distance	Real	Specify in pairs with TA and alternatively to MHD and SHD	<>0
MHD	SD.SysCyc.MHD	Distance of master axis	Real	Specify in pairs with SHD and alternatively to TA and PD	
SHD	SD.SysCyc.SHD	Distance of slave axis	Integer	Specify in pairs with MHD and alternatively to TA and PD	
NH	SD.SysCyc.NH	Number of drillings	Integer	Mandatory	>=1
MA	SD.SysCyc.MA	Angular offset	Real	Optional	Degrees -90<=MA<=90 0 is preset
VO	SD.SysCyc.VO	Line distance	Real	Mandatory	<>0

G115		Hole pattern, matrix	G115(MSP,SSP,TA,PD,MHD,SHD,NH,MA,VO,NL)		
NL	SD.SysCyc.NL	Number of lines	Integer	Mandatory	≥ 1
ORI	SD.SysCyc.ORI	Dimensioning type	Integer	Only for SD parameters	1 - Dimensioning PA and PD 2 - Dimensioning MHD and SHD

Sequence diagram:

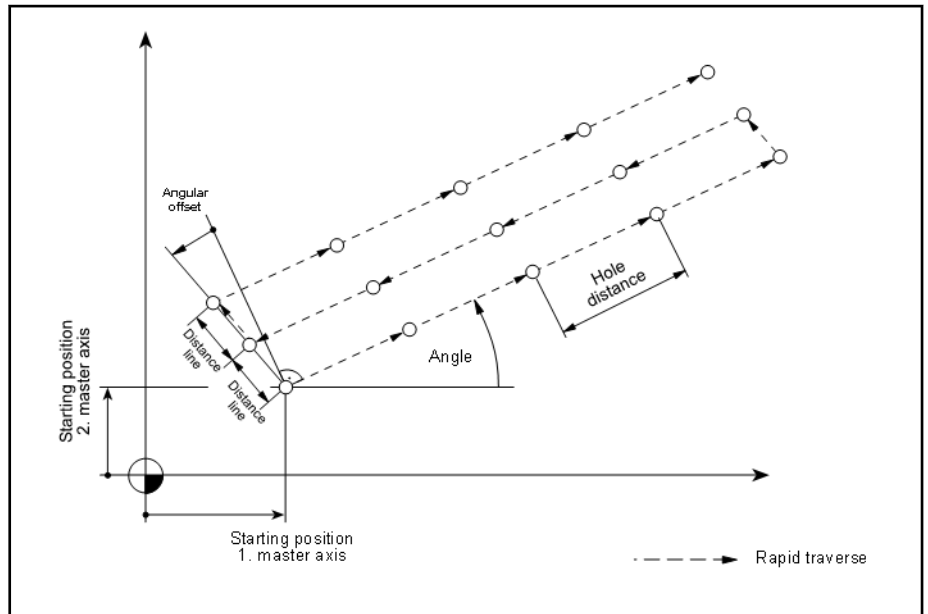


Fig. 6-22: Motion sequence "G115 Hole pattern, matrix (hole distance and angle)

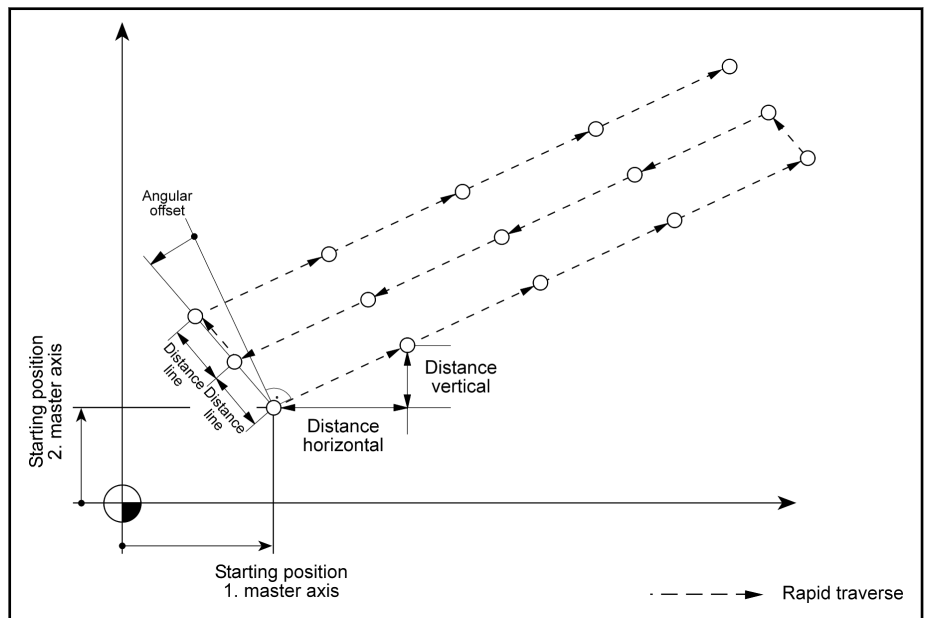


Fig. 6-23: Motion sequence "G115 Hole pattern, matrix (axis-related hole distance)

Sequence: The modal drilling cycle programmed before the cycle call is processed at each position resulted from the input parameters. Define the cycle using the parameters Angle (TA) and Hole Distance (PD) or (probably from MTX 10VRS) using the axis-specific distances (MHD and SHD). To change the machining direction, specify the last drilling position in both cases as starting

Hole patterns

point (MSP and SSP). For the dimensioning type (PA and PD), increase the angle (PA) by 180°. For dimensioning type (MHD and SHD), negate the signs of the parameters "Distance Master Axis" and "Distance Slave Axis". When using SD parameters, the dimensioning type is specified by the SD variable "SD.SysCyc.PDD".

Rotate the matrix additionally by specifying a rotation angle (optionally).



Parameter change. The parameter "BA" for the starting angle was renamed and called "FA" up to the MTX 10VRS. Calls with old parameter names are compatible with the new ones. But if changes were made via the input helps of the NC program editor, the respective parameters have to be renamed manually.

Diameter dimension programming (DIA instruction) is only possible for the x-axis.

Example 1:

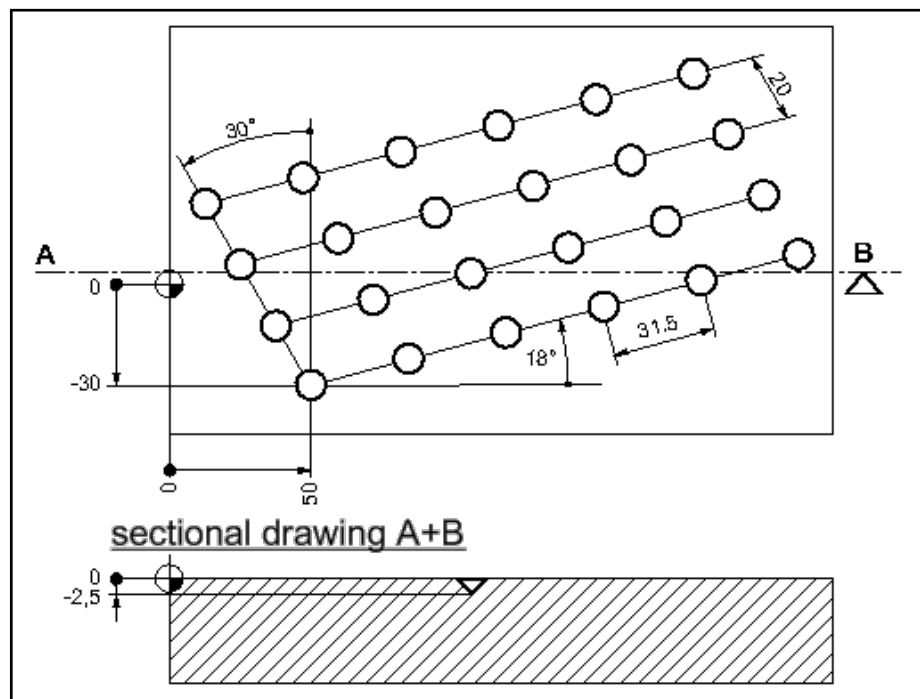


Fig. 6-24: Example of "G115 Hole pattern, matrix (hole distance and angle)"
G115 Hole pattern, matrix (hole distance and angle)

```

N10 T1 M6 ; Tool change
N20 G17 G0 X50 Y-30 Z100 S2500 M3 ; Prepositioning
N30 G81 (IX=Z, SL2, DT-2.5, DW1, RL10) F300 ; Cycle definition + feed
N40 G115 (MSP50, SSP-30, TA18, PD31.5, NH6, MA12, VO20, NL4) ; Cycle definition
N50 G80 Z100 M5 ; Leave contour, delete modal call

```


Example 2:

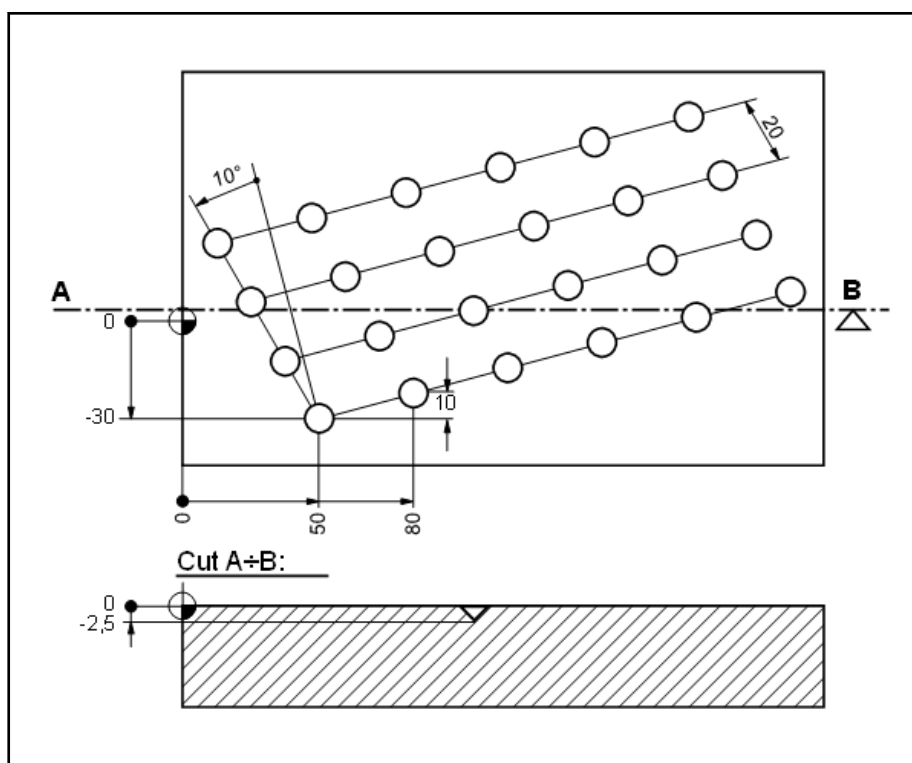


Fig. 6-25: Example of G115 Hole pattern, matrix (axis-related hole distance)
G115 Hole pattern, matrix (axis-related hole distance)

```

N10 T1 M6 ; Tool change
N20 G17 G0 X50 Y-30 Z100 S2500 M3 ; Prepositioning
N30 G81 (IX=Z,SL2,DT-2.5,DW1,RL10) F300 ; Cycle definition + feed
N40 G115 (MSP50,SSP-30,MHD30,SHD10,NH6,TA30,VO20,NL4) ; Cycle definition
N50 G80 Z100 M5 ; Leave contour, delete modal call

```

6.7 G116 Hole pattern, frame

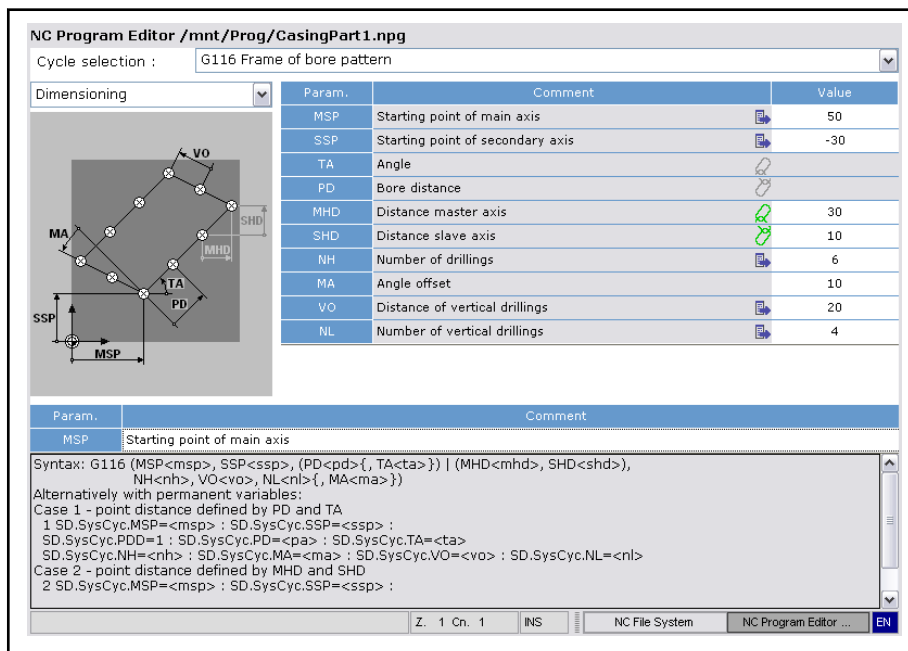


Fig. 6-26: G116 Hole pattern, frame

G116		Hole pattern, frame	G116(MSP,SSP,TA,PD,MHD,SHD,NH,MA,VO,NL)			
MSP	SD.SysCyc.MSP	Starting point: master axis	Real	Mandatory		
SSP	SD.SysCyc.SSP	Starting point: slave axis	Real	Mandatory		
TA	SD.SysCyc.TA	Angle	Real	Optional	Specify in pairs with PD and alternatively to MHD and SHD	0 is preset
PD	SD.SysCyc.PD	Hole distance	Real	Specify in pairs with TA and alternatively to MHD and SHD	<>0 Degrees	
MHD	SD.SysCyc.MHD	Distance of master axis	Real	Specify in pairs with SHD and alternatively to TA and PD		
SHD	SD.SysCyc.SHD	Distance of slave axis	Real	Specify in pairs with MHD and alternatively to TA and PD		
NH	SD.SysCyc.NH	Number of drillings	Integer	Mandatory		>=1
MA	SD.SysCyc.MA	Angular offset	Real	Optional		Degrees -90<=MA<=90 0 is preset
VO	SD.SysCyc.VO	Distance of drillings	Real	Mandatory		<>0

G116		Hole pattern, frame	G116(MSP,SSP,TA,PD,MHD,SHD,NH,MA,VO,NL)		
NL	SD.SysCyc.NL	Number of drillings	Integer	Mandatory	≥ 1
PDD	SD.SysCyc.PDD	Dimensioning type	Integer	Only for SD parameters	1 - Dimensioning PA and PD 2 - Dimensioning MHD and SHD

Sequence diagram:

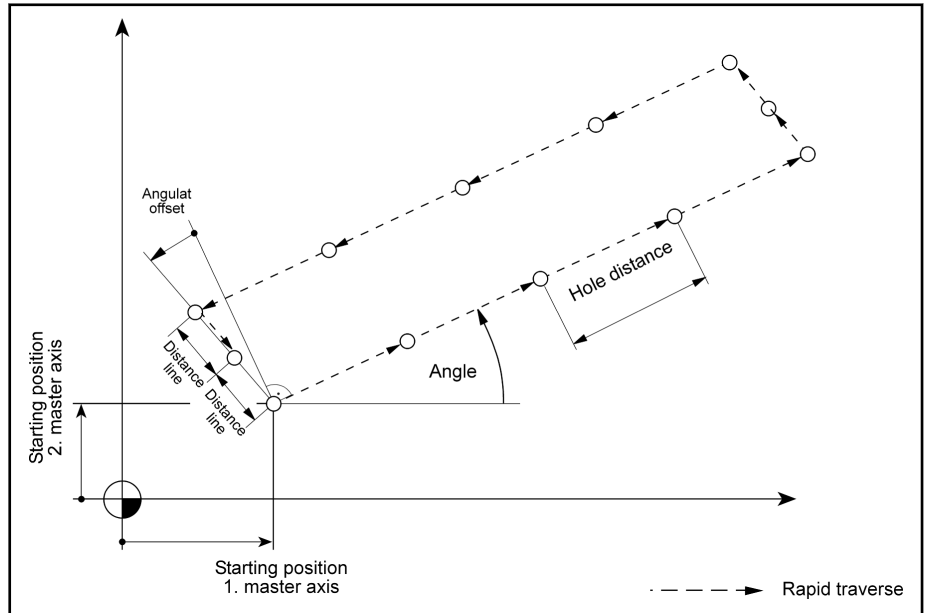


Fig. 6-27: Motion sequence "G116 Hole pattern, frame" (hole distance and angle)

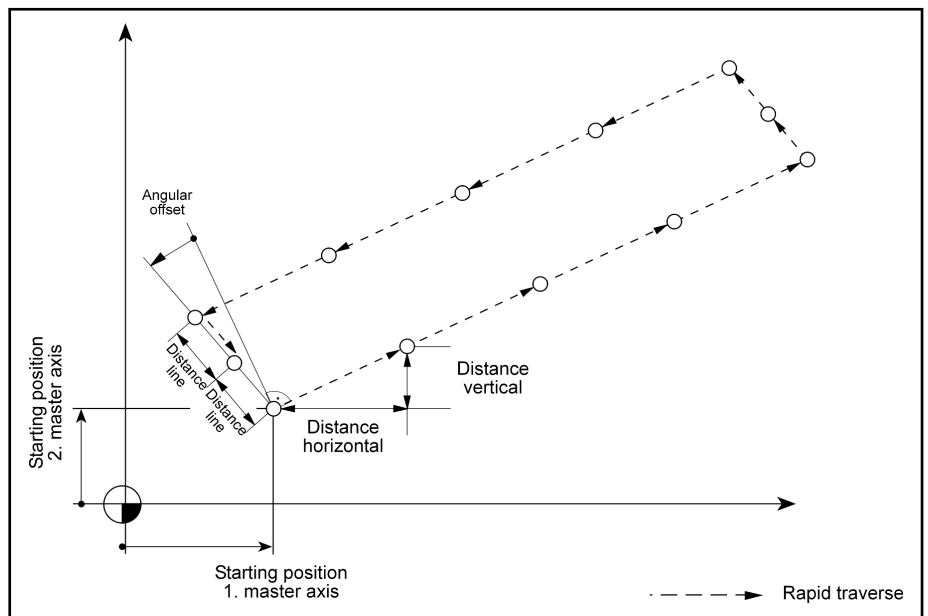


Fig. 6-28: Motion sequence "G116 Hole pattern, frame" ("DIA")

Sequence: The modal drilling cycle programmed before the cycle call is processed at each position resulted from the input parameters. Define the cycle using the parameters Angle (TA) and Hole Distance (PD) or the axis-related distances (MHD and SHD). To change the machining direction, specify the last drilling position in both cases as starting point (MSP and SSP). For the dimensioning

Hole patterns

type (PA and PD), increase the angle (PA) by 180°. For the dimensioning type (MHD and SHD), negate the signs of the parameters Distance Master Axis (MHD) and Distance Slave Axis (SHD). When using SD parameters, the dimensioning type is specified by the SD variable "SD.SysCyc.PDD".

Specify MA (optionally) to rotate the frame additionally.



Parameter change. The parameter "TA" for the starting angle was renamed and called "PA" up to the MTX 10VRS. Calls with old parameter names are compatible with the new ones. But if changes were made via the input helps of the NC program editor, the respective parameters have to be renamed manually.

Diameter dimension programming (DIA instruction) is only possible for the x-axis.

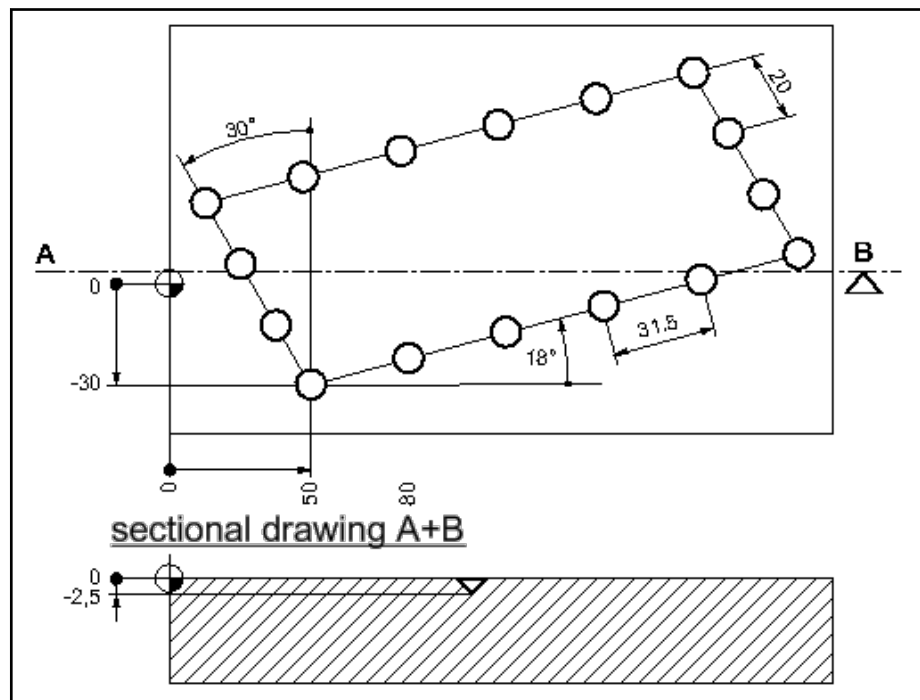
Example 1:

Fig. 6-29: Example of G116 Hole pattern, matrix (hole distance and angle)

Example of G116 Hole pattern, matrix (hole distance and angle)

```

N10 T1 M6 ; Tool change
N20 G17 G0 X50 Y-30 Z100 S2500 M3 ; Prepositioning
N30 G81 (IX=Z,SL2,DT-2.5,DW1,RL10) F300 ; Cycle definition + feed
N40 G116 (MSP50,SSP-30,TA18,PD31.5,NH6,MA12,VO20,NL4) ; Cycle definition
N50 G80 Z100 M5 ; Leave contour, delete modal call

```

Example 2:

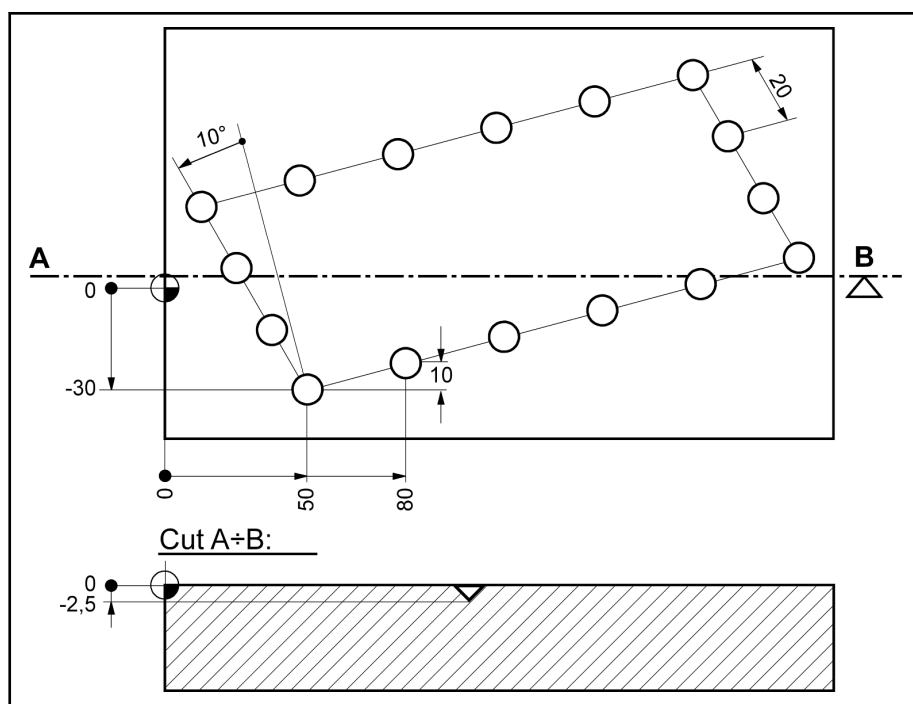


Fig. 6-30: Example of G116 Hole pattern, matrix (axis-related hole distance)

Example of G116 Hole pattern, matrix (axis-related hole distance)

```

N10 T1 M6 ; Tool change
N20 G17 G0 X50 Y-30 Z100 S2500 M3 ; Prepositioing
N30 G81 (IX=Z,SL2,DT-2.5,DW1,RL10) F300 ; Cycle definition + feed
N40 G116 (MSP50,SSP-30,MHD30,SHD10,NH6,MA10,VO20,NL4) ; Cycle definition
N50 G80 Z100 M5 ; Leave contour, delete modal call

```


7 Milling cycles

7.1 General information

The milling cycles described in the following are modal and thus also suitable for processing on point patterns. If the cycles are given without parameter specification, the parameters of the assigned system variables are used.

7.2 G261 Thread milling

Syntax G261(TD,TP,TYP,DK,RLH,DT,UL,RL,SDI,MFI,FF,PC,FD,RC,MFP)

Brief description With the modal cycle G261, internal threads (in case of an existing tapping hole) and external threads can be milled. Optionally, a chamfer can be chamfered using a countersink integrated in the tool face or the tool shank.

Overview

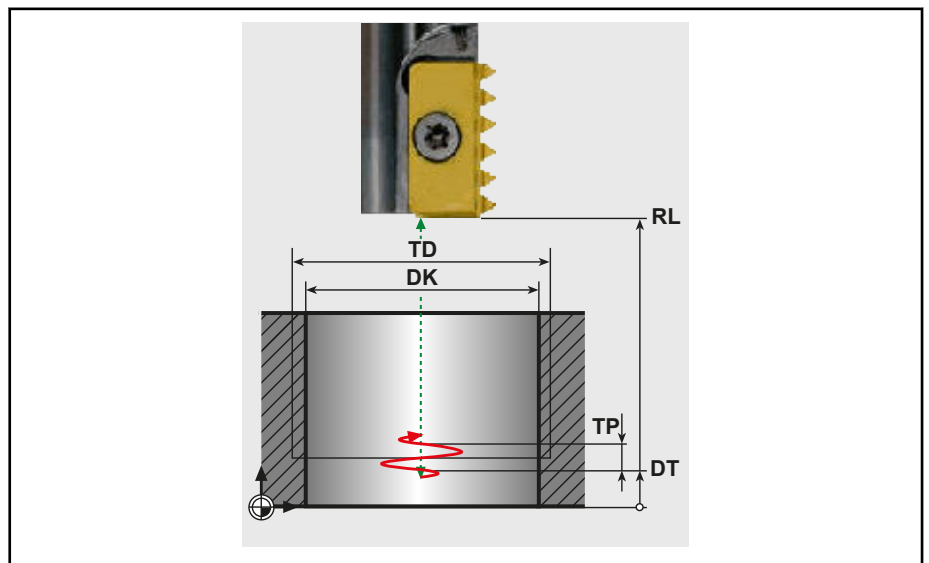


Fig. 7-1: Cycle G261 Thread milling

Parameter

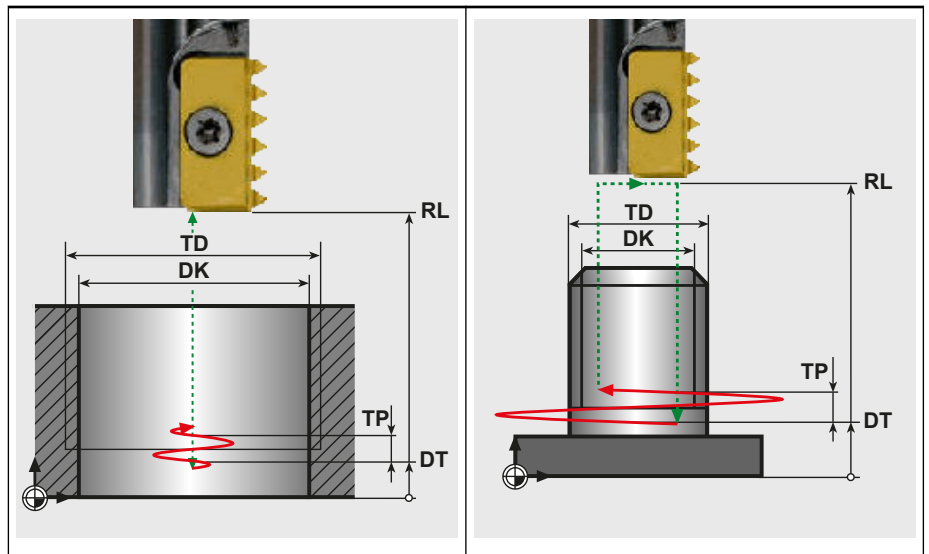
Parameter	Designation	Properties	Values	Pre-assignment
TD	Thread diameter	Mandatory	>0	
TP	Thread pitch	Mandatory	Incremental >0	
TYPE	Type	Optional	Internal thread (0)/ external thread (1)	Internal thread (0)
DK	Kernel diameter	Optional	>0	Internal thread (TYPE0): $TD - 1.08254 \cdot TP$ External thread (TYPE1): $TD - 1.22687 \cdot TP$

Milling cycles

Parameter	Designation	Properties	Values	Pre-assignment
RLH	Right-hand thread/left-hand thread	Optional	Right-hand thread (R)/left-hand thread (L)	Right-hand thread (R)
DT	Infeed coordinate end of thread	Mandatory	Absolute	
UL	Infeed coordinate start of thread	Optional	Absolute	DT +/- TP
RL	Retraction height	Optional	Absolute	UL +/- SDI
SDI	Safety distance	Optional	Incremental ≥0	2 mm
MFI	Feed circular milling	Optional	>0	Current feed
FF	Feed inlet/runout	Optional	>0	MFI
PC	Running direction	Optional	Climb milling (PF)/up-cut milling (CD)	Climb milling (PF)
FD	Infeed coordinate chamfering	Optional specify together with RC	Absolute	No chamfering/countersinking
RC	Radial offset during chamfering	Optional Specify together with FD	>0	No chamfering/countersinking
MFP	Feed chamfering	Optional	>0	MFI

Tab. 7-1: Parameter call

Description



Tab. 7-2: Dimensions of the cylindrical internal and external thread

TD Thread diameter

Corresponds to the nominal diameter of the thread and determines the dimensional accuracy of the internal thread

TP Thread pitch

Pitch of thread

TYPE Type

Switches between internal and external thread

DK Kernel diameter

Determines the milled diameter of an external thread. It is mandatory to program DK if the kernel diameter of an external thread deviates from the default value

In case of an internal thread, the diameter is only taken into consideration in the calculation of the active feed. DK should thus be specified in particular for internal threads with multiple grooves

RLH Right-hand thread/left-hand thread

Determines the direction of rotation of the machined thread

DT Infeed coordinate end of thread

The thread starts or ends at this absolute coordinate

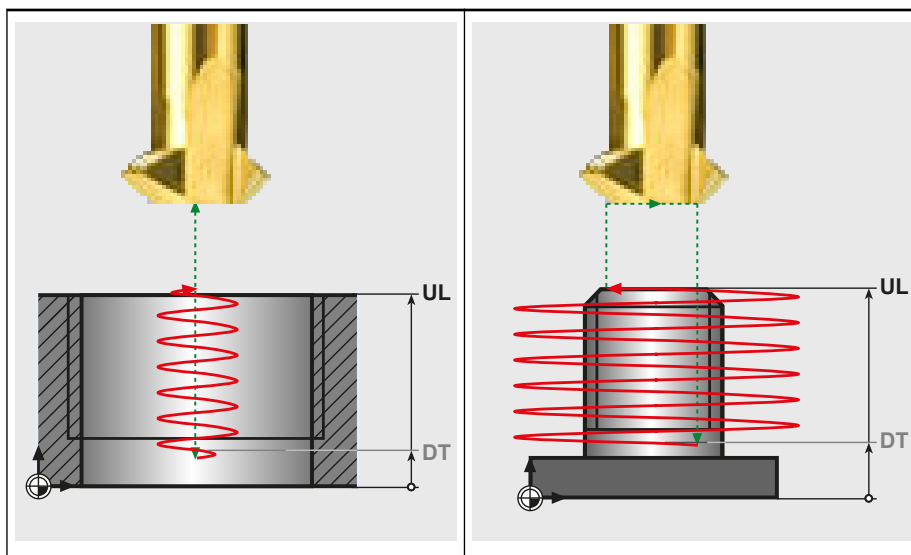
UL Infeed coordinate start of thread

At this absolute coordinate starts or ends the thread. Together with DT and TP, the thread determines the number of milled thread rotations

If UL is not specified, exactly one rotation is milled and the following standard assignment applies (radius programming of infeed axis):

- Retract height RL not programmed:
If the initial position of the infeed axis is greater than DT, $UL = DT + TP$, otherwise $UL = DT - TP$
- RL Retract height programmed:
If RL is greater than DT, $UL = DT + TP$, otherwise $UL = DT - TP$

In case of single tooth milling cutters, several revolutions have to be executed. UL has to be specified



Tab. 7-3: Parameter UL for thread milling

RL Retract height

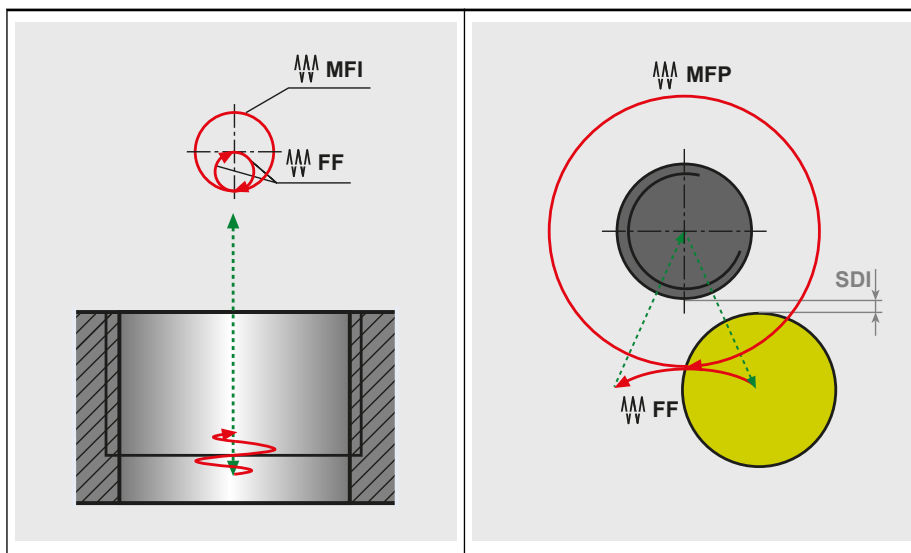
This absolute position is the infeed axis coordinate at the end of the cycle

SDI Safety distance

Is used in radial direction as safety distance for the calculation of the approach circle arc. When chamfering an external thread, this distance is active in case of infeed in axial direction

MFI Feed circular milling

Effective feed when milling the thread. The cycle calculates the feed for the milling center point path from the specified diameters



Tab. 7-4: Feed during circulating

FF Feed inlet/run-out

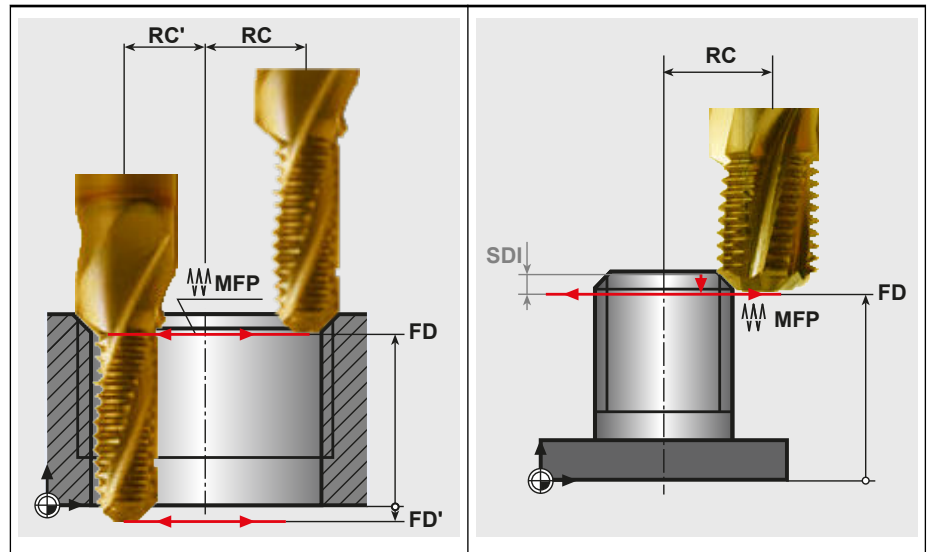
Specifies the effective feed when traveling along the inlet and run-out circle arc

PC Running direction

Switches between parallel feed milling and counter direction milling

FD Infeed coordinate chamfering

At this absolute coordinate, the full circle to chamfer/countersink is executed



Tab. 7-5: Chamfering/countersinking

Radial offset during chamfering

Radius of the full circle for chamfering

Enter the radius of the milling center point path as no milling radius compensation takes place during chamfering/countersinking

MFP Feed chamfering

Effective feed during chamfering/countersinking. The cycle calculates the feed for the milling center point path from the specified diameters

Special features

Note that a modal cycle is not executed with the cycle call during programming, but only after the next traversing command.

The combination of direction of rotation of the spindle, the direction of travel PC and right-hand/left-hand thread RLH determines the circulating direction, i.e. whether machining starts at DT or UL. Thus, UL and DT can be interchanged without influencing the result.

Procedure/processing**Chamfering/countersinking**

IF RC and FD have been defined, it is chamfered first.

- Internal thread:

Positioning to the current position in the interpolation plane with infeed axis to FD takes place in rapid traverse. A starting circle (180°), a full circle for chamfering and a run-out circle (180° ends in the boring center).

- External thread:

The radius of the milling center point path is greater than or equal to the milling diameter. The opening angle of the arc is max. 90°. The arc is calculated, taking the safety distance SDI into consideration. Traversing takes place radially in rapid traverse to the starting point of the entry circle. Subsequently, infeed to FD takes place in the infeed axis, whereas the last distance SDI is covered in feed due to safety reasons. This is followed by a starting circle, a full circle for chamfering and a run-out circle.

Thread milling

- Internal thread:

Positioning to the starting point of the starting arc (180° opening angle) takes place in rapid traverse in the infeed axis. The maximum radius of the infeed arc (center of milling cutter) corresponds to the tool diameter. In case of a tool diameter half the nominal diameter, radial positioning to the starting point of the starting circle takes place. This is followed by a starting circle (180°), a full circle for chamfering and a run-out circle (180°). If the starting circle did not start at the boring center, it is now positioned to the boring center. The tool is moved to RL in rapid traverse.

- External thread:

Traversing takes place radially in rapid traverse to the starting point of the entry circle. Subsequently, the infeed axis is positioned to the starting point of the starting arc. The radius of the milling center point path is greater than or equal to the milling diameter. The opening angle of the arc is max. 90°. The arc is calculated so that at least the safety distance SDI between tool and spigot is complied with. This is followed by a starting circle, circulating of the thread and a run-out circle. The tool is moved to RL in rapid traverse and is positioned to the spigot center.

Example A thread milling cutter whose edge covers the entire total length has to be used to mill the following internal thread.

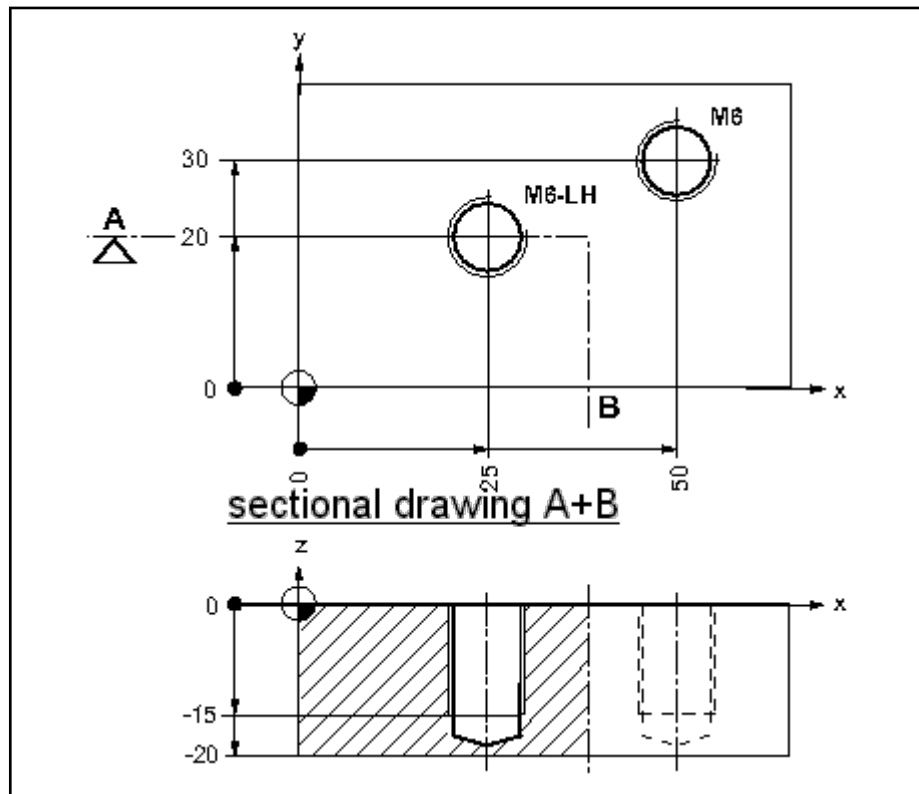


Fig. 7-2: Example of G261 Thread milling
Program for G261 Thread milling

```

N10 T1 M6 ; Tool change
N20 G17 S1500 F800 M3 ; Technology values
N30 G261(TD6, FF1000, RLH=L, DT-15, TP1, RL10) ; Cycle definition
N40 G0 X25 Y20 ; 1. Cycle call
N50 G261(RLH=R) ; Modify parameter for right-hand thread
N60 X50 Y30 ; 2. Cycle call
N70 G80 Z100 M5 ; Leave contour, delete modal call

```

7.3 G262 Thread milling

Syntax G262(HD,TP,RLH,SL,DT,FD,RL,FS,FF,MS,MFI,DW,MFP,IOF,LI,PC)

Brief description Use the modal cycle G262 to bore a tapping hole, countersunk, and to mill an internal thread. A drill and thread milling cutter, a combination tool with cutting bits for the tapping hole and the countersinking and a milling edge for the thread are used.

Overview

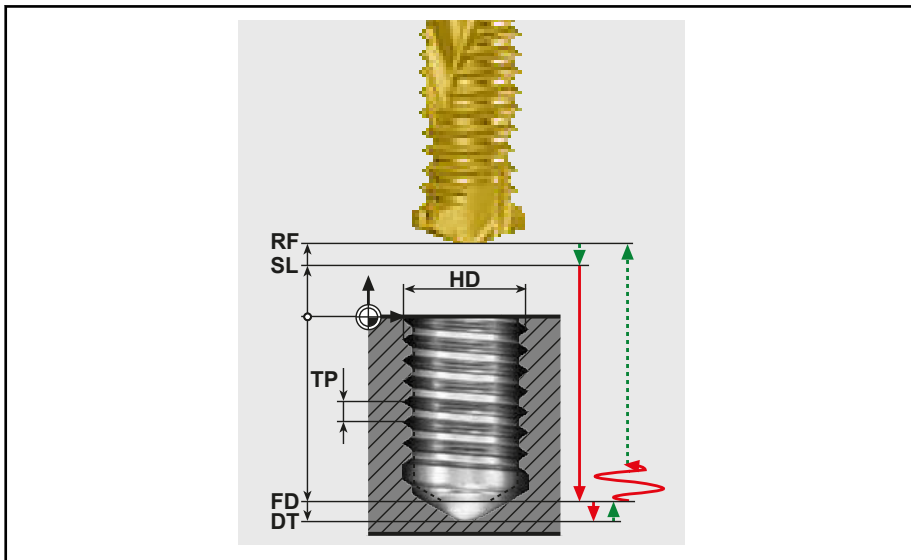


Fig. 7-3: Cycle G262 Drill and thread milling

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
HD	Thread diameter	Mandatory	>0	
TP	Thread pitch	Mandatory	Incremental >0	
RLH	Right-hand thread/left-hand thread	Optional	Right-hand thread (R)/ left-hand thread (L)	Right-hand thread (R)
SL	Safety level	Mandatory	Absolute	
DT	Tap hole depth	Mandatory	Absolute	
FD	Depth of start of countersinking	Optional	Absolute	No countersinking
RL	Retraction height	Optional	Absolute	Initial position of drilling axis in front of the cycle
FS	Drill speed	Optional	>0	Current speed
FF	Drill feed	Optional	>0	Current feed
MS	Countersink speed	Optional	>0	FS

Parameter	Designation	Properties	Values	Pre-assignment
MFI	Countersink feed	Optional	>0	FF
DW	Dwell time	Optional	>= 0 Seconds	0
MFP	Feed circular milling	Optional	>0	Current feed

Tab. 7-6: Parameter call

Description**HD Thread diameter**

Corresponds to the nominal diameter of the thread

TP Thread pitch

Pitch of thread

RLH Right-hand thread/left-hand thread

Determines the direction of rotation of the machined thread

SL Safety level

Machining in drilling feed starts at this absolute position

DT Depth of tap hole

Tap hole drilling or countersinking starts or ends at this absolute coordinate

FD Depth of start of countersinking

Countersinking with countersinking feed MFI and countersinking speed MS starts at this absolute coordinate

RL Retract height

This absolute position is the coordinate of the boring axis at the end of the cycle

FS Drill speed

Speed at tap hole drilling

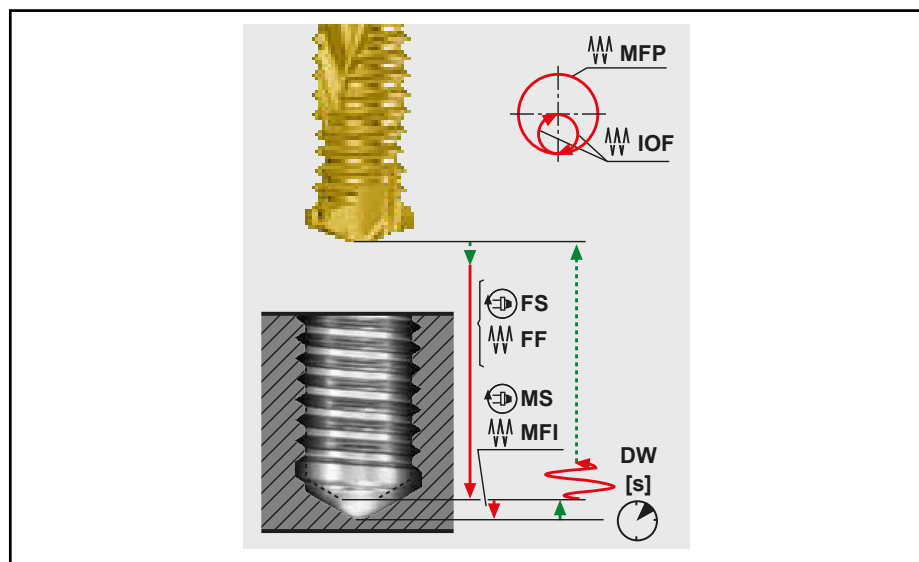


Fig. 7-4: Feeds and speeds for drill and thread milling

FF Drill feed

Feed during tap hole drilling

MS speed

Speed during countersinking

MFI Plunging feed

Feed during countersinking

DW Dwell time

Duration the tool "waits" at the lowest point

MFP Feed circular milling

Effective feed when milling the thread. The cycle calculates the feed for the milling center point path from the specified diameters

IOF Feed inlet/runout

Specifies the effective feed when traveling along the inlet and run-out circle arc. The cycle calculates the feed for the milling center point path from the specified diameters

LI Machining travel (unchuck stroke) before milling

In case of an active machining travel (unchuck stroke), it is positioned to SL in rapid traverse after boring

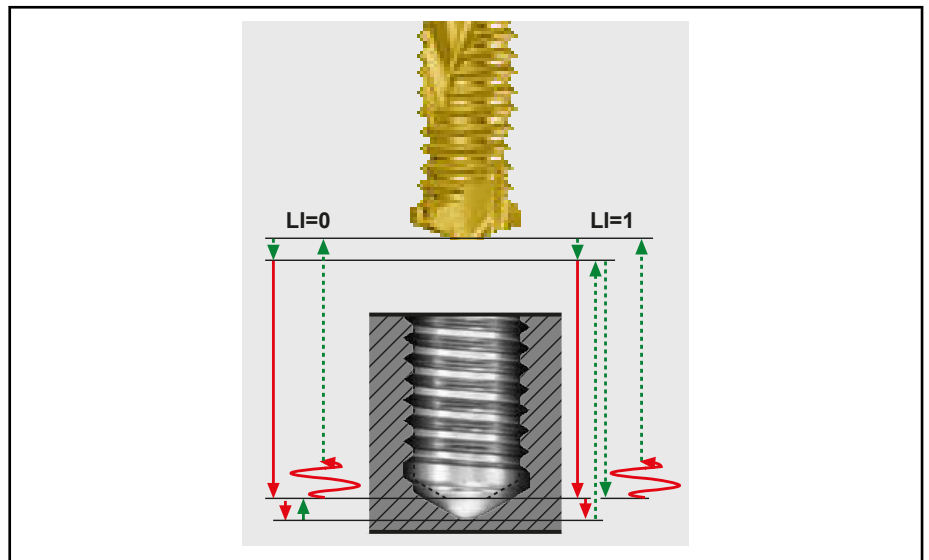


Fig. 7-5: Metal cutting in case of thread Milling

PC Running direction

Switches between parallel feed milling and counter direction milling

Special features Note that a modal cycle is not executed with the cycle call during programming, but only after the next traversing command.

The combination of direction of rotation of the spindle, the direction of travel PC and right-hand/left-hand thread RLH determines the circulating direction.

The thread approach starts at FD or the thread run-out ends at FD (depending on the direction of machining). IF FD was not programmed or if FD is not deeper than half the tap hole depth, the thread approach starts or the thread run-out ends at DT.

Procedure/processing Positioning to the safety plane SL takes place in rapid traverse. Subsequently, the tap hole is drilled with boring feed FF and boring speed FS until the depth for starting the countersink operation (FD) is reached. Countersinking is executed with the countersink speed MS and the countersink feed MFI and ends at the tap hole depth DT.

Without specifying FD, countersinking with special operate values is omitted and drilling to DT is executed in one step.

If a dwell time has been specified, it is waited for DW seconds at the reached depth DT.

If the machining travel LI is active, positioning takes place in rapid traverse to safety level SL.

The thread is milled, consisting of a starting arc with a 180° opening angle (helix arc with an axial traverse path of $0.25 \cdot TP$), a full circle for the thread and a run-out arc. The approach starts or the run-out ends at FD (depending on the direction of machining). The speed valid before the cycle is active during milling. The feed IOF becomes active at the approach/run-out. At the thread, the feed for circular milling MFP is active.

The run-out arc ends in the boring center and is positioned to the retract plane RL in rapid traverse.

Example Drill the following tap holes with a drill and thread milling cutter (without countersinking).

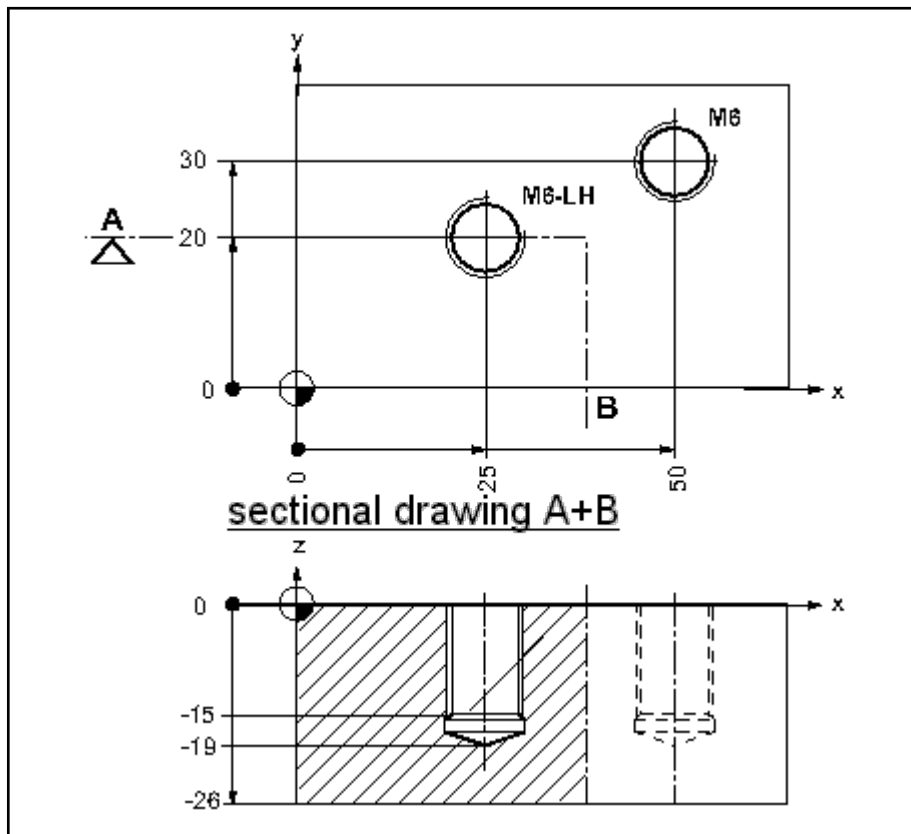


Fig. 7-6: Example of G262 Drill and thread milling
Program for G261 Thread milling

```

N10 T1 M6                ; Tool change
N20 G17 S2000 F800 M3    ; Technology values
N30 G262(SL2, DT-19, FS1000, FF250, RL10, HD6, IOF1000, RLH=L, TP1)
                          ; Cycle definition
N40 G0 X25 Y20           ; 1. Cycle call
N50 G262(RLH=R)          ; Modify parameter for right-hand thread
N60 X50 Y30              ; 2. Cycle call
N70 G80 Z100 M5          ; Leave contour, delete modal call

```

Program:

7.4 G263 Groove circular milling

Syntax G263(HD,GD,CDI,SL,DT,CD,RL,IFP,MFP,OFF,IFI,OFI)

Brief description The modal cycle G263 mills a radial groove in a drill hole or at a circular spigot. If the groove width exceeds the cutting width of the tool, several, axially offset cuts can be executed.

Overview

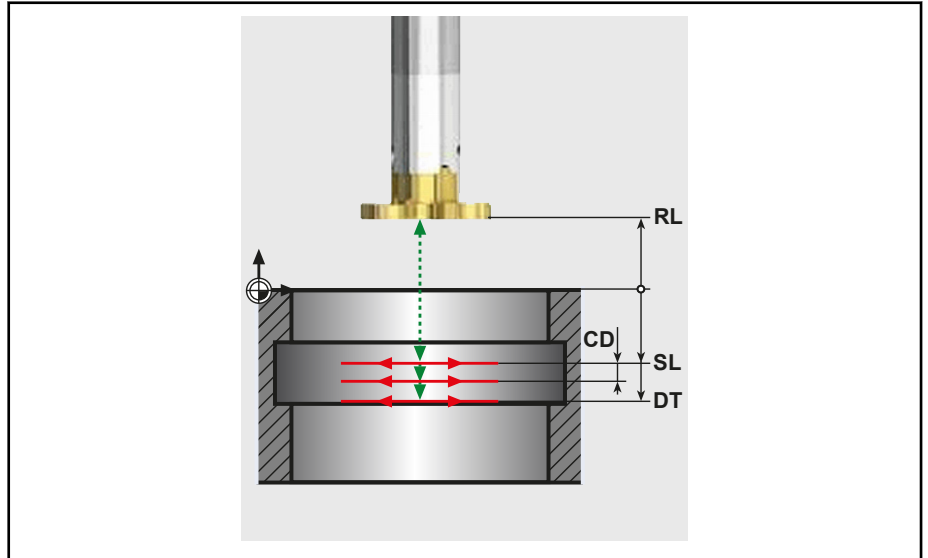


Fig. 7-7: Cycle G263 Groove circular milling

Parameter

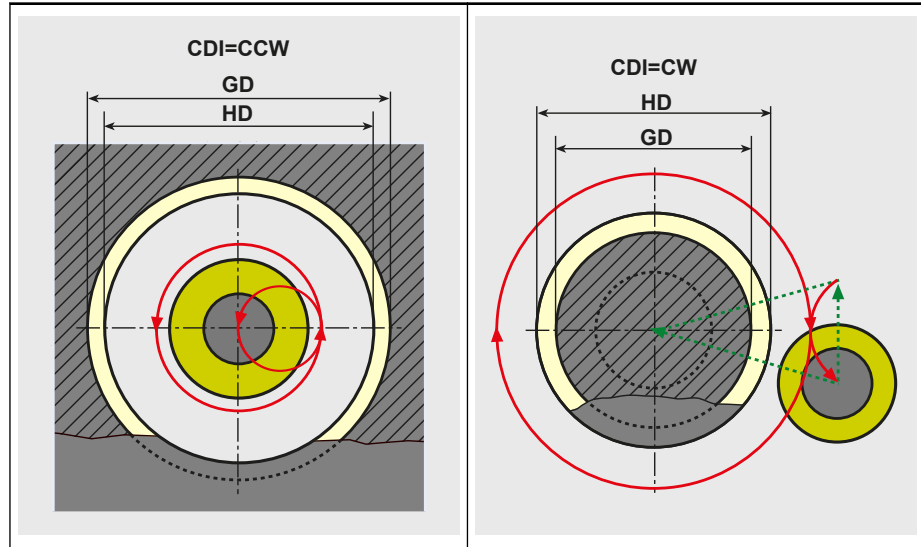
Parameter	Designation	Properties	Values	Pre-assignment
HD	Diameter of boring or spigot	Mandatory	>0	
GD	Diameter of the groove	Mandatory	>0	
CDI	Direction of machining	Optional	Clockwise (CW)/ counter-clockwise (CCW)	Direction of rotation matches the climb milling
SL	First infeed coordinate	Mandatory	Absolute	
DT	Last infeed coordinate	Mandatory	Absolute	
CD	Maximum in-feed depth	Optional	Incremental	No cut segmentation
RL	Retraction height	Optional	Absolute	Initial position of in-feed axis in front of the cycle
IFP	Inlet feed	Optional	>0	MFP
MFP	Feed circular milling	Optional	>0	Current feed

OFP	Run-out feed	Optional	>0	MFP
IFI	Entry feed of infeed axis	Optional	>0	Rapid traverse

Tab. 7-7: Parameter call

Description HD Diameter of boring or spigot

Available diameter of the boring or the spigot



Tab. 7-8: Diameter for circular milling

GD Diameter of the groove

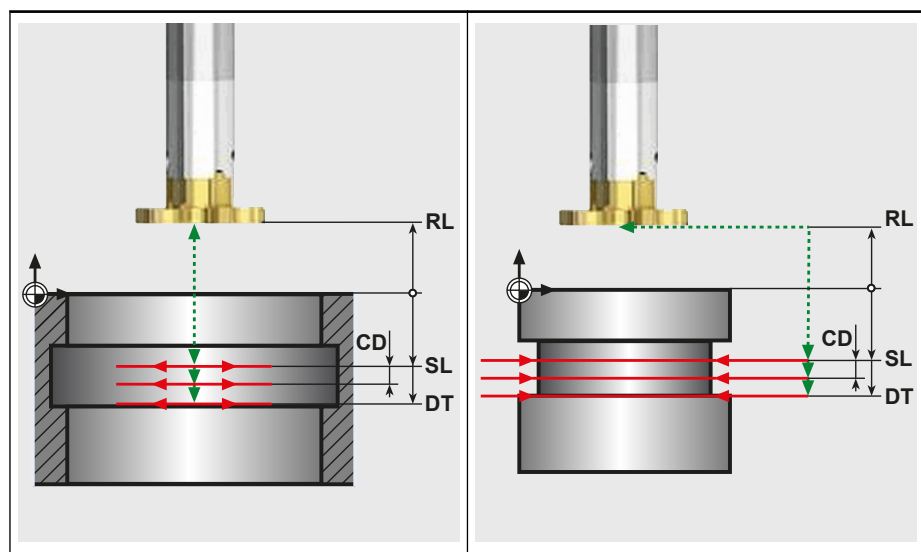
Diameter at groove base

CDI Machining direction

Defines the machining direction during circulation

SL First infeed coordinate

The first step is executed at this absolute position



Tab. 7-9: Cycle parameter in infeed direction

DT Last infeed coordinate

If DT deviates from SL, the last cut is executed at this absolute position

CD Maximum infeed depth

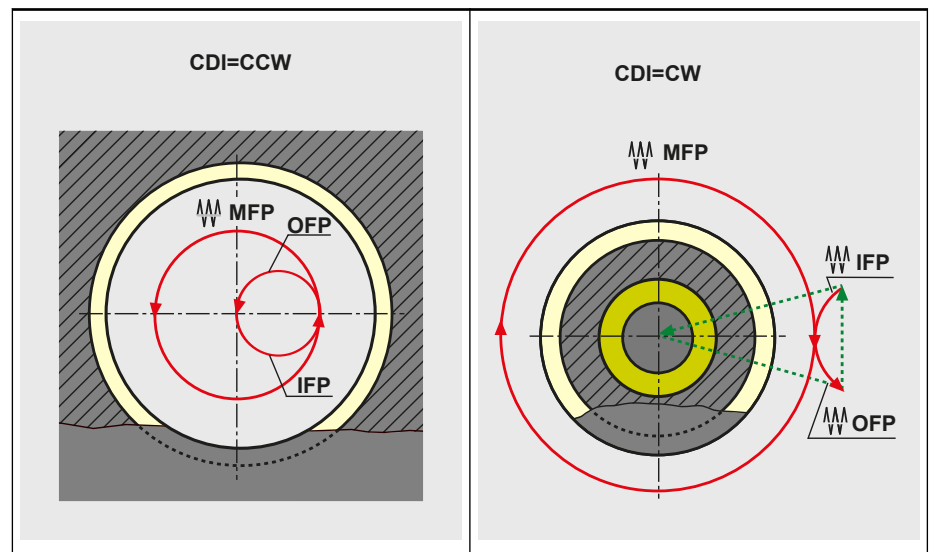
Specifies the maximum distance between two cuts. CD has to be smaller than the tool width to ensure overlapping of the cut

RL Retract height

This absolute position is the infeed axis coordinate at the end of the cycle

IFP Inlet feed

Effective feed on the infeed arc. The cycle calculates the feed for the milling center point path from the specified diameters



Tab. 7-10: Radial feeds for circular milling

MFP Feed circular milling

Effective feed during circulating. The cycle calculates the feed for the milling center point path from the specified diameters

OFP Run-out feed

Effective feed on the run-out arc. The cycle calculates the feed for the milling center point path from the specified diameters

IFI Entry feed of infeed axis

Feed for infeed before a cut

This feed becomes active in a boring also when approaching the starting point of the starting circle, if the circle is not in the boring center point

OFI Leaving feed of infeed axis

Feed for retracting to the retract plane RL. This feed becomes active in a boring also when approaching the starting point after a cut, if the run-out arc does not end in the boring center point

Special features

Note that a modal cycle is not executed with the cycle call during programming, but only after the next traversing command.

The relation between the diameters HD and GD determine if plunging takes place at a groove or in a boring.

Procedure/processing

To machine this cycle, approach the center of the circular groove to be milled.

Boring:

In the cycle, it is moved to the depth SL in rapid traverse or with the entry feed IFI. Subsequently, the approaching circle (semi-circle) whose maximum radius (milling center point path) corresponds to the tool diameter follows. If the starting point of the entry circle is outside the boring center due to this limitation, the starting position of the entry circle is approached using the entry feed IFP or, if not specified, in rapid traverse. The approach circle in the feed IFP or the circular feed MFP and traveling a full circle with the feed MFP follow. A run-out arc with the feed OFP and a return to the center of the drilling follow without specifying OFP in rapid traverse.

If SL and DT deviate from each other, it is infed again and a circle is milled again. If CD (the maximum cutting depth) is specified, more than two steps to reach the final depth are possible. It is infed in the infeed feed IFP or in rapid traverse.

The cycle is completed with positioning to the retract height RL either in rapid traverse or with the leaving feed OFP.

Spigot:

The radius of the milling center point path is greater than or equal to the milling diameter. The opening angle of the arc is max. 90°. The approaching arc is calculated taking the safety distance into consideration. Positioning takes place radially in rapid traverse to the starting point of the entry circle and then it is travelled in infeed direction to the depth SL in rapid traverse or with the entry feed IFI. The approach circle in the feed IFP or the circular feed MFP and traveling a full circle with the feed MFP follow. A run-out arc with the feed OFP follows.

If SL and DT deviate from each other, it is infed again and a circle is milled again. If CD (the maximum cutting depth) is specified, more than two steps to reach the final depth are possible. It is infed in the infeed feed IFP or in rapid traverse.

At the end of the cycle, it is positioned to the retract height RL in rapid traverse or with the leaving feed OFP. Subsequently, the spigot center is traversed.

Example Machine the ring groove with a disk milling cutter using the parallel feed method.

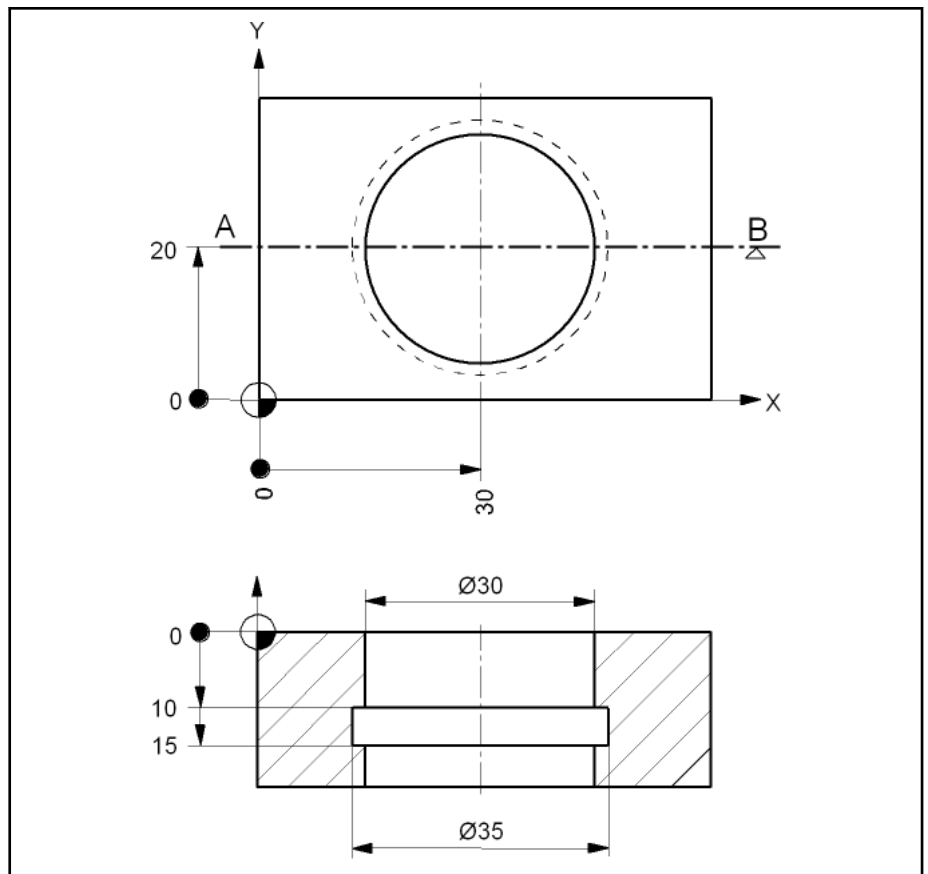


Fig. 7-8: Example G263 Groove circular milling
Program for G263 Groove circular milling

```

N10 T1 M6 ; Tool change (side milling cutter)
N20 G17 S2000 F1000 M3 ; Technology values
N30 G263(GD35, HD30, IFP800, OFP2000, SL-13, DT-15, RL10)
; Cycle definition
N40 G0 X30 Y20 ; Cycle call
N50 G80 Z100 M5 ; Leave contour, delete modal call

```

7.5 G266 Face milling, rectangle

The cycle "G266 Face milling, rectangle" machines arbitrarily oriented rectangular areas. It is distinguished between roughing (multiple milling in infeed direction up to the finishing allowance) and finishing (single milling). The depth infeed is executed outside. Therefore, a material-free area with a minimum width resulting from the sum of milling cutter diameter and safety distance (parameter RD) around the rectangular area has to be ensured.

The cycle works without milling cutter radius correction. A radius value greater than 0 in the tool correction register is a prerequisite. In case of a face milling head (conically positioned edges), the radius set is to correspond to the circular motion of the lower edge corners.

Milling cycles

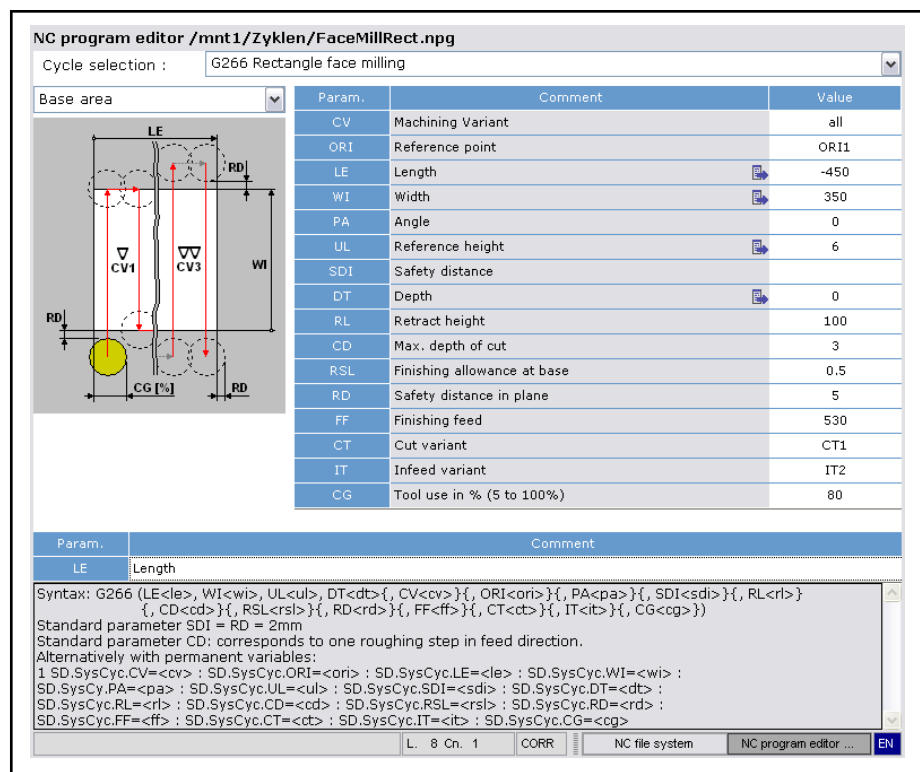


Fig. 7-9: G266 Face milling, rectangle

G266		Face milling, rectangle	G266(CV,ORI,LE,WI,PA,UL,SDI,DT,RL,CD,RSL,RD,FF,C T,IT,IM,CG)			
CV	SSysCyc.CV	Machining (cutting) variant	Integer	Optional	1 Roughing 2 Complete 3 Finishing 2 is the default setting	
ORI	SD.SysCyc.ORI	Reference point	Integer	Optional	1 Corner 2 Center 1 is preset	
LE	SD.SysCyc.LE	Length	Real	Mandatory	Absolute	
WI	SD.SysCyc.WI	Width	Real	Mandatory	Absolute	
PA	SD.SysCyc.PA	Angle	Real	Optional	Absolute 0 is preset	
UL	SD.SysCyc.UL	Reference height	Real	Mandatory	Absolute	
SDI	SD.SysCyc.SDI	Safety distance	Real	Optional	Incremental >0 Default: 2 mm	
DT	SD.SysCyc.DT	Depth	Real	Mandatory	Absolute	
RL	SD.SysCyc.RL	Retraction height	Real	Optional	Absolute	

CD	SD.SysCyc.CD	Maximum infeed depth	Real	Optional	Incremental >0 One roughing cut in infeed direction by default
RSL	SD.SysCyc.RSL	Finishing allowance at the base	Real	Optional	Incremental >0 0 is preset
RD	SD.SysCyc.RD	Safety distance in plane	Real	Optional	Incremental >0 Default: 2 mm
FF	SD.SysCyc.FF	Finishing feed	Real	Optional	>0 Latest programmed F-value by default
CT	SD.SysCyc.CT	Cutting variant	Integer	Optional	1 Milling direction along width WI 2 Milling direction along length LE 1 is preset
IT	SD.SysCyc.IT	Infeed variant	Integer	Optional	1 Alternating direction (meander) 2 In one direction 1 is preset
IM	SD.SysCyc.IM	Arrangement of following cuts	Integer	Optional	1 Optimum in terms of time, with reversal of direction 2 Following cuts congruent with first cut 1 is preset
CG	SD.SysCyc.CG	Tool use	Real	Optional	5 to 100 % 80 % is preset

Sequence: The cycle supports all interpolation planes and the diameter programming of the X-axis. All coordinates (in this case only coordinates in the infeed axis) are absolute coordinates and are to be specified with the respective axis scaling (DIA/RAD). All remaining parameters are to be specified with radius dimension.

The cycle runs modally and has to be deselected with G80.

The position of the area to be machined is determined by the approached coordinates in the interpolation plane to execute the cycle. The position of the rectangle, relative to the initial position, is influenced by the parameter **ORI** (**ORI1** initial position is identical to the corner of the rectangle, **ORI2** initial position corresponds to the center of the triangle), turning angle **PA** as well as the signs of length **LE** and width **WI**. If **PA** is zero, the reference point is the lower left corner in case of a positive sign. A negative sign for **LE** or **WI** offsets the reference point to the opposite corner. Machining starts close to the corner determined by the signs. That means that the signs of **LE** and **WI** determine the machining direction of the first cut.

The cutting variant **IT** determines the rectangular extension of the milling direction. The infeed direction per line is along the other rectangular extension. **IT1** determines that the cuts are made along the width **WI** and the infeed is

executed for each milled line along the length LE. IT2 causes the milling direction to extend in parallel to length LE.

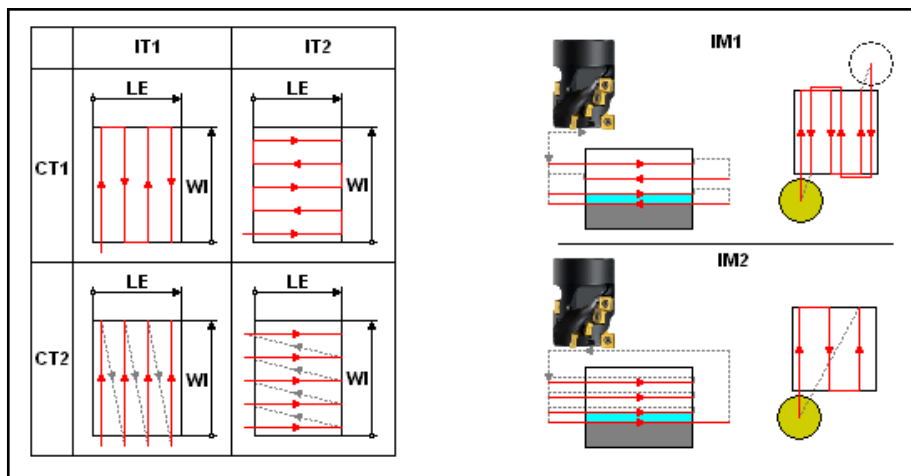


Fig. 7-10: Mode of action of machining strategy parameters CT, IT and IM for G266

The milling cutter center point is positioned around the tool radius plus the radial safety distance **RD** in front of the rectangle when approaching. The distance of the first line from the rectangle limitation and the distance between the lines is influenced by the tool use **CG**. The actual contact width to reach an equal line distance can be reduced.

In addition to the contour extension start, the radial safety distance **RD** also defines the distance radially traveled by the tool during finishing and the distance traveled by the tool outside the edge in the last line to ensure that the complete area is fully cut off.

During roughing, the height to be machined – that is the difference from the depth **DT** and the reference height **UL** minus the finishing allowance **RSL** – is divided into equal infeed amounts. The amounts are not higher than the maximum depth of the cut **CD**. Without **CD** specification, a single milling is executed while roughing. That means that the tool is used for direct machining of the finishing allowance. Otherwise, the infeed amount is reduced by the reference height **UL**. Before finishing, the tool is set to the depth **DT**.

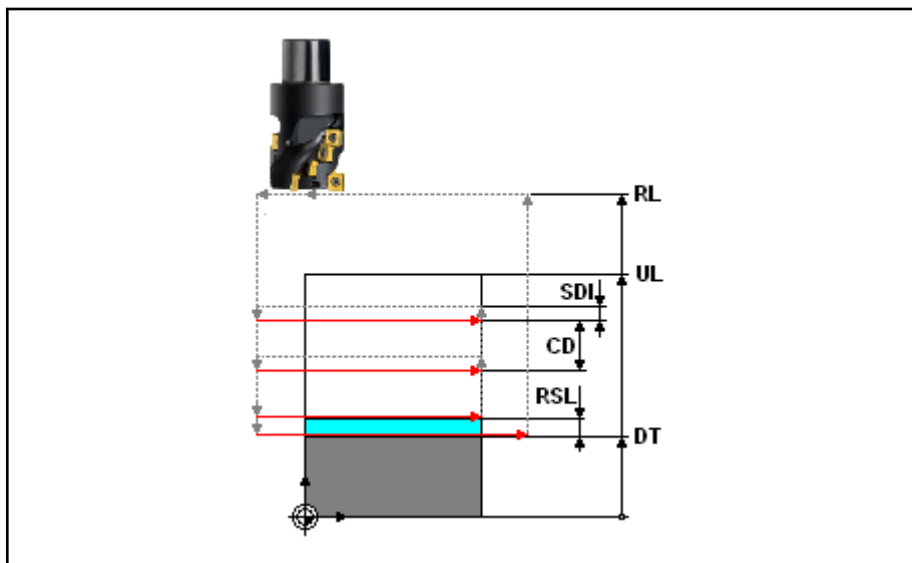


Fig. 7-11: Depth parameters and allowances for G266

The direction of the following lines is determined using the infeed variant **CT**. CT1 means that the direction alternating in each line. The infeed from line to line while roughing in the material is executed in the working feed. CT2 causes each line to be milled in the same direction, with the tool being retracted after each cut.

While roughing, (to be activated with **CV** 1 or 2) the complete area is milled off line by line with the work feed **F** programmed prior to the cycle. If retraction is required before the next depth infeed, the retract distance can be influenced via the safety distance **SDI**. Milling is repeated until the finishing allowance is reached. When finishing (CV2 or 3), the area is milled once. For this procedure, a special finishing feed **FF** can be activated. If combined roughing and finishing (CV2) is selected, finishing is carried out only if a finishing allowance of $RSL > 0$ has been specified.

By default, the following cuts in infeed direction are arranged optimally in terms of path and with reversal of direction. The reversal of direction can be suppressed with the parameter **IM**. Program IM2 in radial cut segmentation direction, e.g., if the space available is confined, as then it is only necessary to keep a free space of at least **RD** on the side specified.

After machining, the tool is positioned on the retrace plane **RL** - or if not specified - on the reference plane **UL** plus the safety distance **SDI**. Finally, the plane of the initial position before the cycle is approached.

G40 (no edge radius correction) is to be active prior to the cycle. The cycle deactivates global transition elements (**RND**, **RNE**, **CHF**, and **CHS**).

Example: A rectangular area is to be removed in several cuts (depth of cut 3 mm max) using a face milling head. A finishing cut with an allowance of 0.5 mm is to be provided. Due to the inclined position of the edges, a radial safety distance of 5 mm is required. The cuts are to be executed in a meander shape parallel to the longer side of the rectangle.

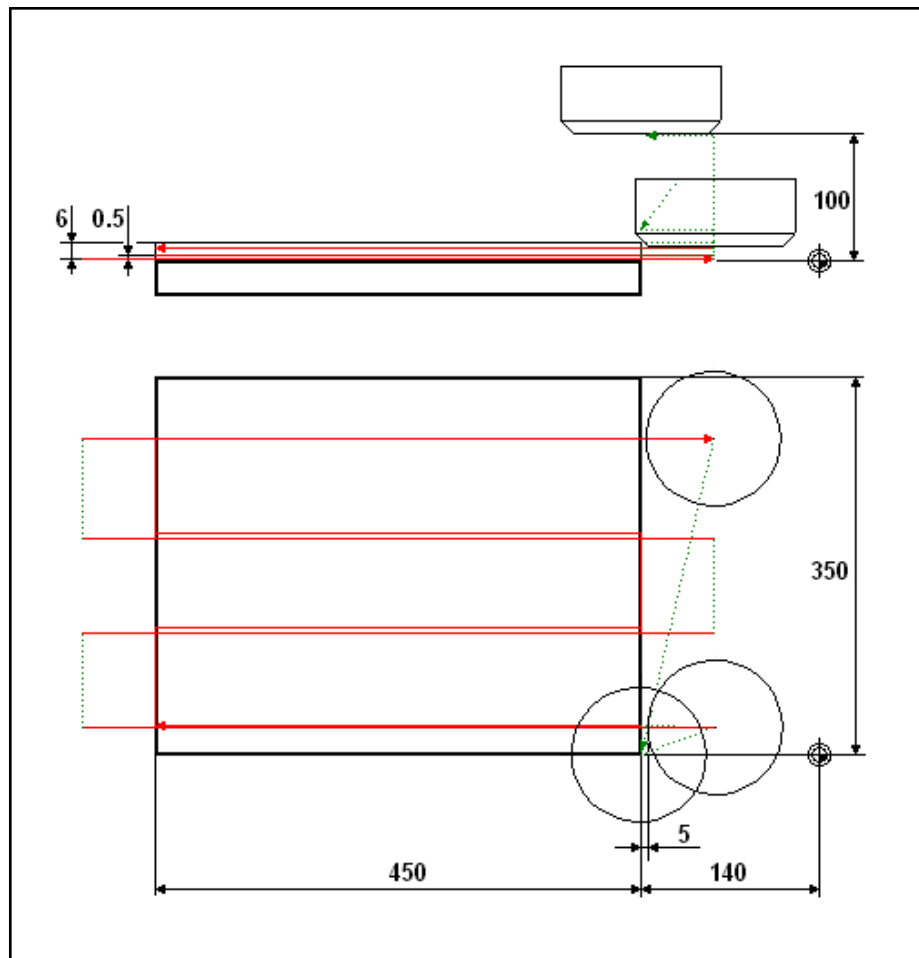


Fig. 7-12: Example of G266 Face milling, rectangle
G266 Face milling, rectangle

```

N10 T1 M6 ; Face milling head Dmr. 125
N20 G17 RAD G54 G94 G97 S640 M13 F760 ; Coordinate system, cutting data
; Cycle definition
N30 G266 (LE-450, WI350, UL6, DT0, RL100, CD3, RSL0.5, RD5, FF530, IT2)
N40 G0 X-140 Y0 Z10 ; Approach reference point at lower right corner; process cycle
N50 G80 M5 ; Deselect modal cycle

```

7.6 G267 Face milling, circle

Circular areas can be machined with the cycle "G267 Face milling, circle". It is distinguished between roughing (multiple milling in infeed direction up to the finishing allowance) and finishing (single milling). The depth infeed is executed outside. Therefore, a material-free area with a minimum width of the sum of milling cutter diameter and safety distance (parameter RD) around the complete circle has to be ensured.

The cycle works without milling cutter radius correction. A radius value greater than 0 in the tool correction register is a prerequisite. In case of a face milling head (conically positioned edges), the radius set is to correspond to the circular motion of the lower edge corners.

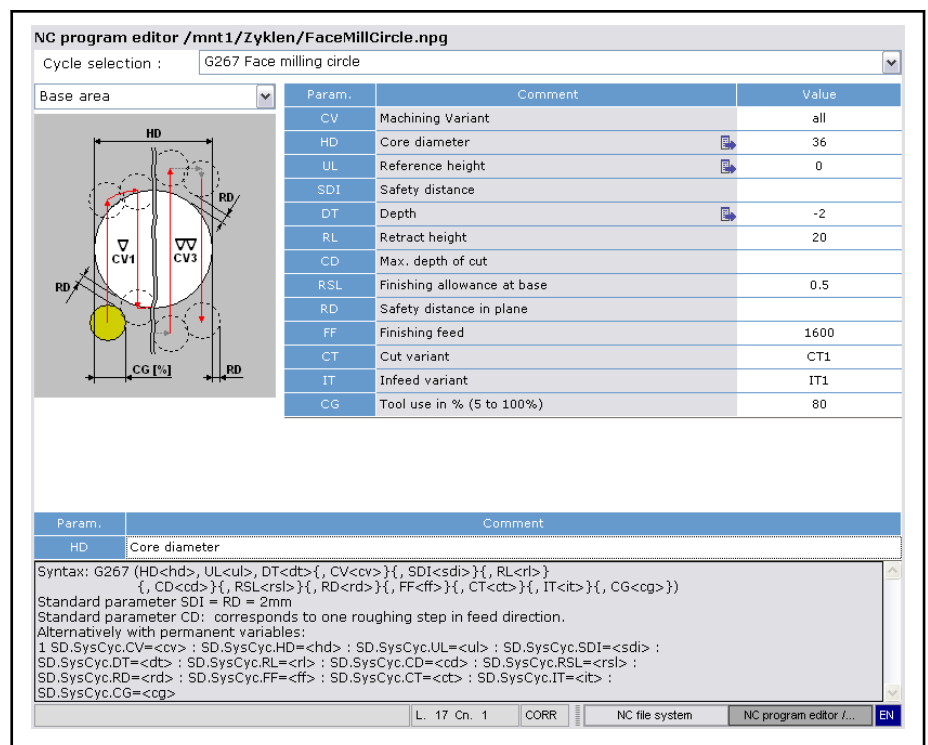


Fig. 7-13: G267 Face milling, circle

G267		Face milling, circle	G267(CV,HD,UL,SDI,DT,RL,CD,RSL,RD,FF,CT,IT,IM,CG)		
CV	SSysCyc.CV	Machining (cutting) variant	Integer	Optional	1 Roughing 2 Complete 3 Finishing 2 is the default setting
HD	SD.SysCyc.HD	Circle diameter	Real	Mandatory	Absolute
UL	SD.SysCyc.UL	Reference height	Real	Mandatory	Absolute
SDI	SD.SysCyc.SDI	Safety distance	Real	Optional	Incremental >0 Default: 2 mm
DT	SD.SysCyc.DT	Depth	Real	Mandatory	Absolute
RL	SD.SysCyc.RL	Retraction height	Real	Optional	Absolute
CD	SD.SysCyc.CD	Maximum infeed depth	Real	Optional	Incremental >0 One roughing cut in infeed direction by default
RSL	SD.SysCyc.RSL	Finishing allowance at the base	Real	Optional	Incremental >0 0 is preset

Milling cycles

G267		Face milling, circle	G267(CV,HD,UL,SDI,DT,RL,CD,RSL,RD,FF,CT,IT,IM,CG)		
RD	SD.SysCyc.RD	Safety distance in plane	Real	Optional	Incremental >0 Default: 2 mm
FF	SD.SysCyc.FF	Finishing feed	Real	Optional	>0 Latest programmed F-value by default
CT	SD.SysCyc.CT	Cutting variant	Integer	Optional	1 Milling direction along the secondary axis 2 Milling direction along the main axis 1 is preset
IT	SD.SysCyc.IT	Infeed variant	Integer	Optional	1 Alternating direction (meander) 2 In one direction 1 is preset
IM	SD.SysCyc.IM	Arrangement of following cuts	Integer	Optional	1 Optimum in terms of time, with reversal of direction 2 Following cuts congruent with first cut 1 is preset
CG	SD.SysCyc.CG	Tool use	Real	Optional	5 to 100 % 80 % is preset

Sequence: The cycle supports all interpolation planes and the diameter programming of the X-axis. All coordinates (in this case only coordinates in the infeed axis) are absolute coordinates and are to be specified with the respective axis scaling (DIA/RAD). All remaining parameters are to be specified with radius dimension.

The cycle runs modally and has to be deselected with G80.

The position of the center of the circle is determined by the approached coordinates in the interpolation plane to execute the cycle.

The cutting variant **IT** defines the axis for the milling direction. IT1 determines that the cuts are along the secondary axis, IT2 along the main axis. The infeed is always executed from line to line in the respective other axis of the plane. The first cut is in positive direction of the main or secondary axis. The milling direction can be reversed by specifying a negative diameter **HD**.

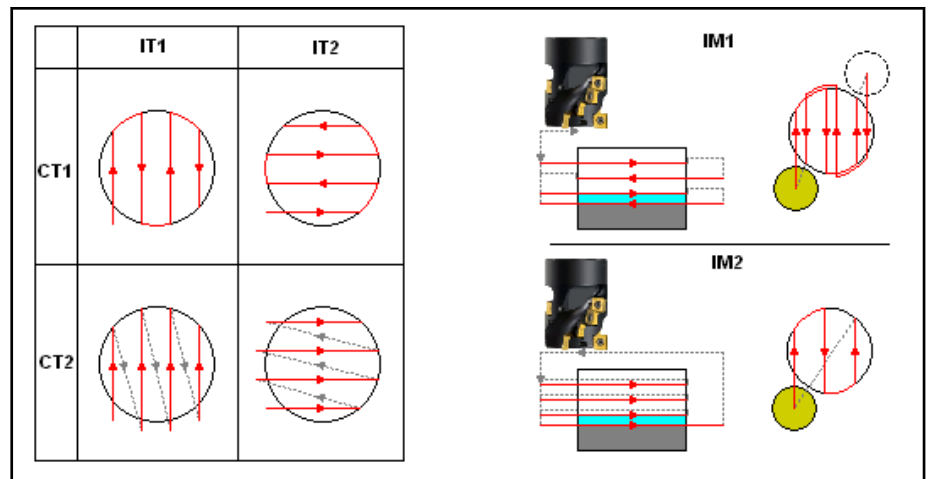


Fig. 7-14: Mode of action of machining strategy parameters CT, IT and IM for G267

The milling cutter center point is positioned around the tool radius plus the radial safety distance **RD** in front of the circle when approaching. The distance of the first line from the circle periphery and the distance between the lines are influenced by the tool use **CG**, the contact width in percent related to the tool diameter. The actual contact width to reach an equal line distance can be reduced.

In addition to the approach, the radial safety distance **RD** also defines the distance radially traveled by the tool during finishing and how far the tool outside transfers the circular periphery in the last line to ensure that the complete area is fully cut off.

During roughing, the height to be machined – that is the difference from the depth **DT** and the reference height **UL** minus the finishing allowance **RSL** – is divided into equal infeed amounts. The amounts are not higher than the maximum depth of the cut **CD**. Without **CD** specification, a single milling is executed while roughing. That means that the tool is used for direct machining of the finishing allowance. Otherwise, the infeed amount is reduced by the reference height **UL**. Before finishing, the tool is set to the depth **DT**.

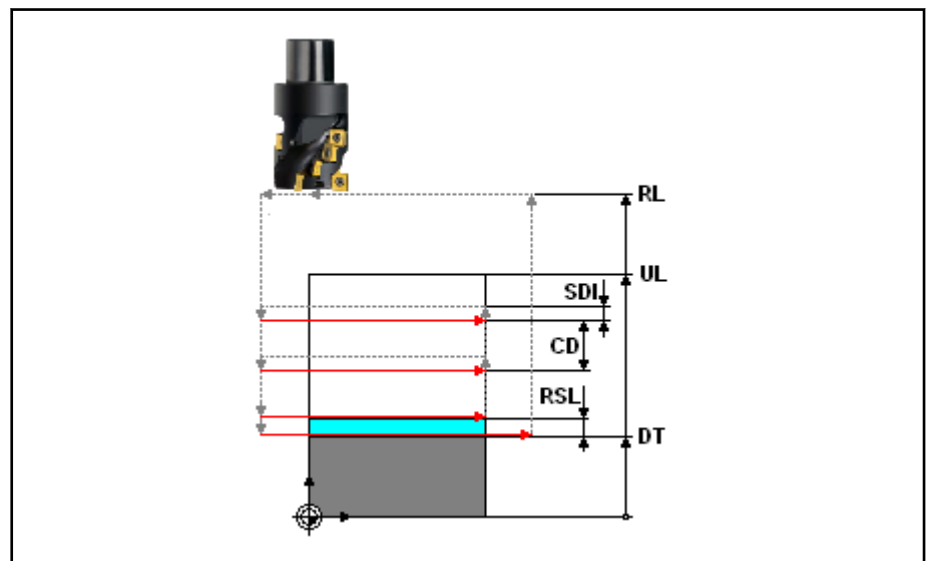


Fig. 7-15: Depth parameters and allowances for G267

Milling cycles

The milling direction of the following lines is determined using the infeed variant **CT**. CT1 means that the direction alternating in each line. The infeed from line to line while roughing in the material is executed in the working feed. CT2 causes each line to be milled in the same direction, with the tool being retracted after each cut.

While roughing, (to be activated with **CV** 1 or 2) the complete area is milled off line by line with the work feed **F** programmed prior to the cycle. If retraction is required before the next depth infeed, the retract distance can be influenced via the safety distance **SDI**. Milling is repeated until the finishing allowance is reached. When finishing (CV2 or 3), the area is milled once. For this procedure, a special finishing feed **FF** can be activated. If combined roughing and finishing (CV2) is selected, finishing is carried out only if a finishing allowance of $RSL > 0$ has been specified.

By default, the following cuts in infeed direction are arranged optimally in terms of path and with reversal of direction. The reversal of direction can be suppressed with the parameter **IM**. Program IM2 in radial cut segmentation direction, e.g., if the space available is confined, as then it is only necessary to keep a clearance of at least **RD** on the side specified.

After machining, the tool is positioned on the retrace plane **RL** - or if not specified - on the reference plane **UL** plus the safety distance **SDI**. Finally, the plane of the initial position before the cycle is approached.

G40 (no edge radius correction) is to be active prior to the cycle. The cycle deactivates global transition elements (**RND**, **RNE**, **CHF**, and **CHS**).

Example: The three circular areas located on a pitch circle have to be face-milled. The milling depth of 2 mm in total has to be removed by a roughing cut and a finishing cut (finishing allowance of 0.5 mm). 20 mm have to be removed before the infeed to the next circular area. The cut is to be in meander shape.

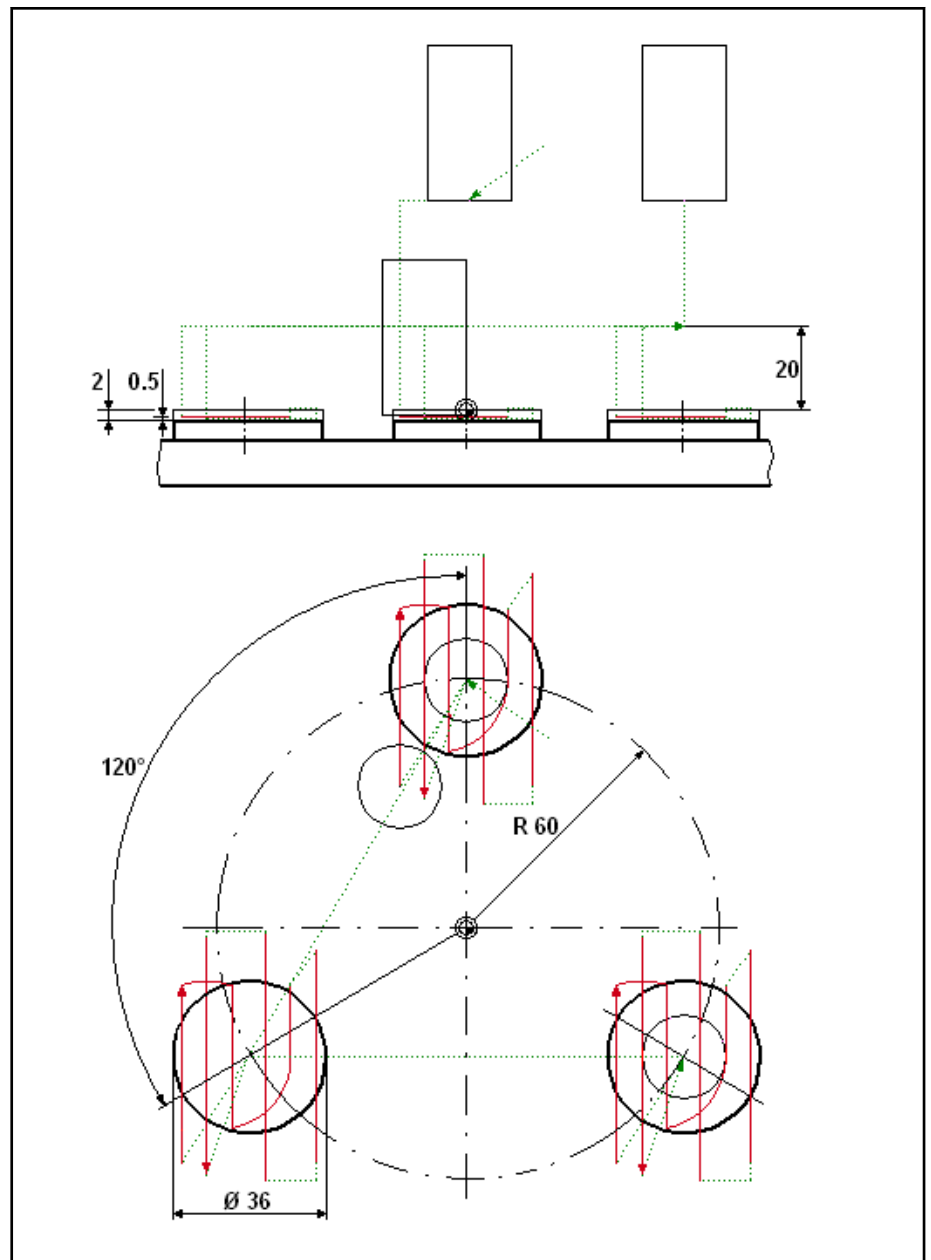


Fig. 7-16: Example of G267 Face milling, circle

Program for G267 Face milling, circle

```

N10 T1 M6 ; End milling cutter Dmr. 20
N20 G17 RAD G97 G54 S4000 M13 F2400 ; Coordinate system, cutting data
N30 G267 (HD36, UL0, DT-2, RL20, RSL0.5, FF1600) ; Cycle definition
N40 G113 (MCP0, SCP0, PR60, NH3, BA90) ; Teilkreis R60 ; Apply cycle to partial circle
N50 G0 G80 Z50 M5 ; Leave contour, delete modal call

```

7.7 G271 Rectangular pocket milling

The cycle "Rectangle pocket milling" (G271) mills rectangular-shaped pockets. Depending on the direction of rotation of the tool, milling can be performed in clockwise or counterclockwise direction. The infeed is executed either in direction of the tool axis or oscillating.

Milling cycles

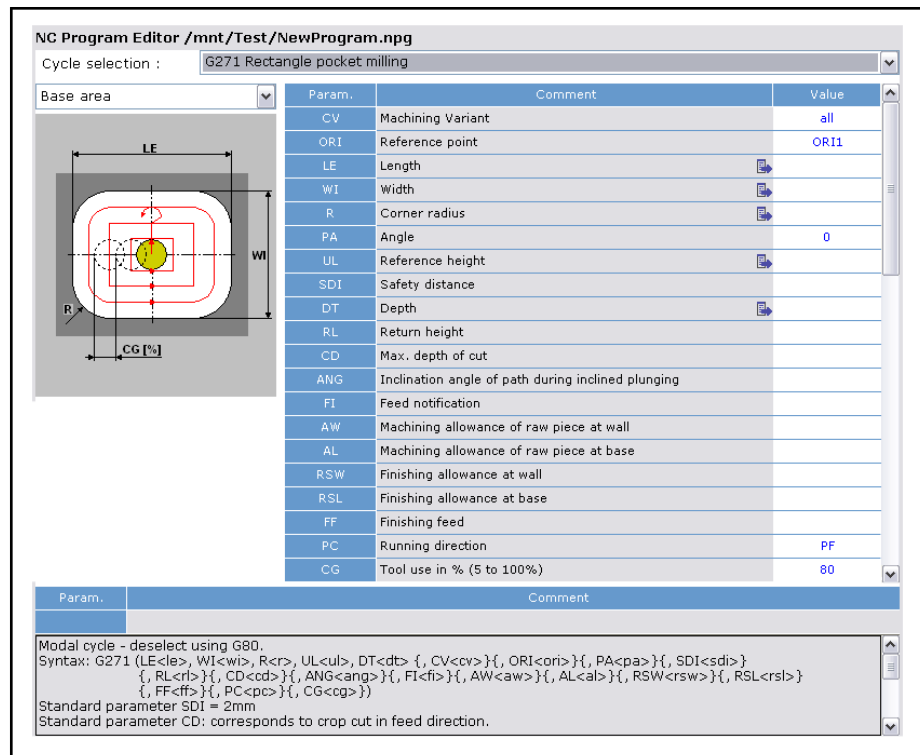


Fig. 7-17: G271 Rectangle pocket milling

G271		Rectangle pocket milling	G271(CV,ORI,LE,WI,R,PA,UL,SDI,DT,RL,CD,ANG,FI,AW,AL,RSW,RSL,FF,PC,CG)		
CV	SSysCyc.CV	Machining (cutting) variant	Integer	Optional	1 Roughing 2 Complete 3 Finishing 2 is the default setting
ORI	SD.SysCyc.ORI	Reference point	Integer	Optional	1 Corner 2 Center 2 is the default setting
LE	SD.SysCyc.LE	Length	Real	Mandatory	Absolute
WI	SD.SysCyc.WI	Width	Real	Mandatory	Absolute
R	SD.SysCyc.R	Corner radius	Real	Mandatory	Absolute >0
PA	SD.SysCyc.PA	Angle	Real	Optional	Absolute 0 is preset
UL	SD.SysCyc.UL	Reference height	Real	Mandatory	Absolute
SDI	SD.SysCyc.SDI	Safety distance	Real	Optional	Incremental 2 mm is the default setting
DT	SD.SysCyc.DT	Depth	Real	Mandatory	Absolute
RL	SD.SysCyc.RL	Retraction height	Real	Optional	Absolute
CD	SD.SysCyc.CD	Max. infeed depth	Real	Optional	Incremental >0

G271		Rectangle pocket milling	G271(CV,ORI,LE,WI,R,PA,UL,SDI,DT,RL,CD,ANG,FI,AW,AL,RSW,RSL,FF,PC,CG)		
ANG	SD.SysCyc.ANG	Inclination angle of path during inclined plunging	Real	Optional	1° to 45°
FI	SD.SysCyc.FI	Infeed feed	Real	Optional	>0 50 % of the latest programmed F-value by default
AW	SD.SysCyc.AW	Machining allowance of the un-machined part at the wall	Real	Optional	>0
AL	SD.SysCyc.AL	Machining allowance of the un-machined part at the base	Real	Optional	>0
RSW	SD.SysCyc.RSW	Finishing allowance at the wall	Real	Optional	>0
RSL	SD.SysCyc.RSL	Finishing allowance at the base	Real	Optional	>0
FF	SD.SysCyc.FF	Finishing feed	Real	Optional	>0 Latest programmed F-value by default
PC	SD.SysCyc.PC	Running direction	String	Optional	PF clockwise direction CD counterclockwise direction PF is preset
CG	SD.SysCyc.CG	Tool use	Real	Optional	5 to 100 % 80 % is preset

Sequence: The cycle supports planes G17, G18, G19 and radius allowance. The cycle runs modally and has to be deselected with G80. The tool is set in the plane on the starting position. Depending on **ORI**, it is the corner (ORI 1) or the center of the rectangle (ORI 2). From this position, the height of the safety distance **SDI** and the center of the rectangle is approached in rapid traverse. Machining starts in infeed direction at the reference height **UL** and ends at the depth **DT**. A machining allowance at the base "AL" moves the machining start to this height. The finishing allowance **RSL** remains while roughing. The machining allowance **AW** at the wall moves the machining start out of the center if the free space in the center exceeds the milling cutter diameter.

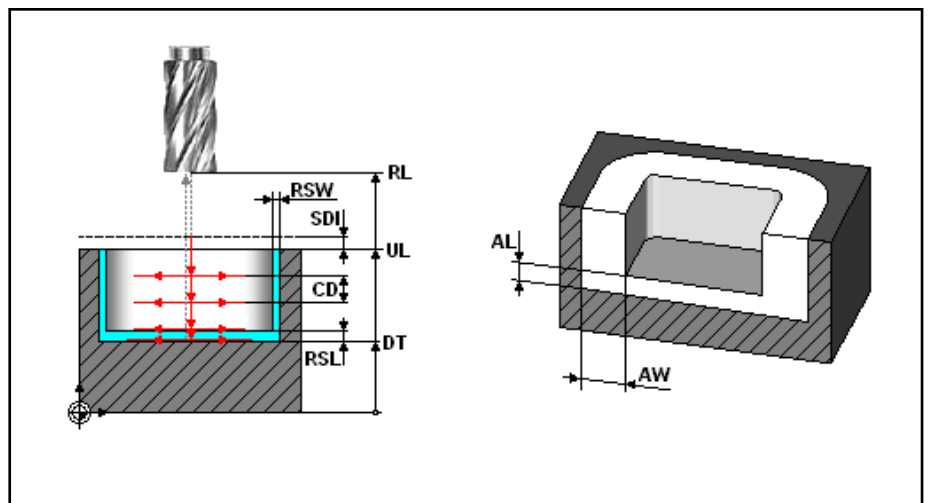


Fig. 7-18: Depth parameters, machining allowances and allowances for G271

It can be infed in three different variants. If there is enough free space available for the tool (ratio of length and width of the rectangle, tool diameter and AW considered), the infeed depth is approached in rapid traverse. If there is not enough free space available, it is traveled with the infeed feed to the infeed depth. If an inclination angle **ANG** is specified and there is not enough free space available, the infeed depth is reached by oscillating in longitudinal direction with the specified inclination. The oscillating distance required is determined internally.

If a maximum infeed depth **CD** is specified, the total infeed depth is segmented into equal infeeds (at the most maximum infeed depth). At the end of machining, the tool is traveled in rapid traverse to the reference height or, if this height is not specified, to the safety distance.

The machining variants are roughing, roughing and finishing or finishing (completely). Finishing requires at least one finishing allowance (RSW or RSL). Roughing is executed with a rectangular infeed in the plane and without edge radius correction. To finish the contour, an approach and run-out radius is added and an edge position correction is used.

If a machining allowance is specified (**AW** or **AL**), machining is restricted to these values.

Depending on the milling direction and the existing direction of the spindle rotation, machining takes place in a clockwise and counterclockwise direction.

The tool radius is determined from the NC (active tool required).

G40 (no edge radius correction) is to be active prior to the cycle. The cycle deactivates global transition elements (RND, RNE, CHF, and CHS).

Example:

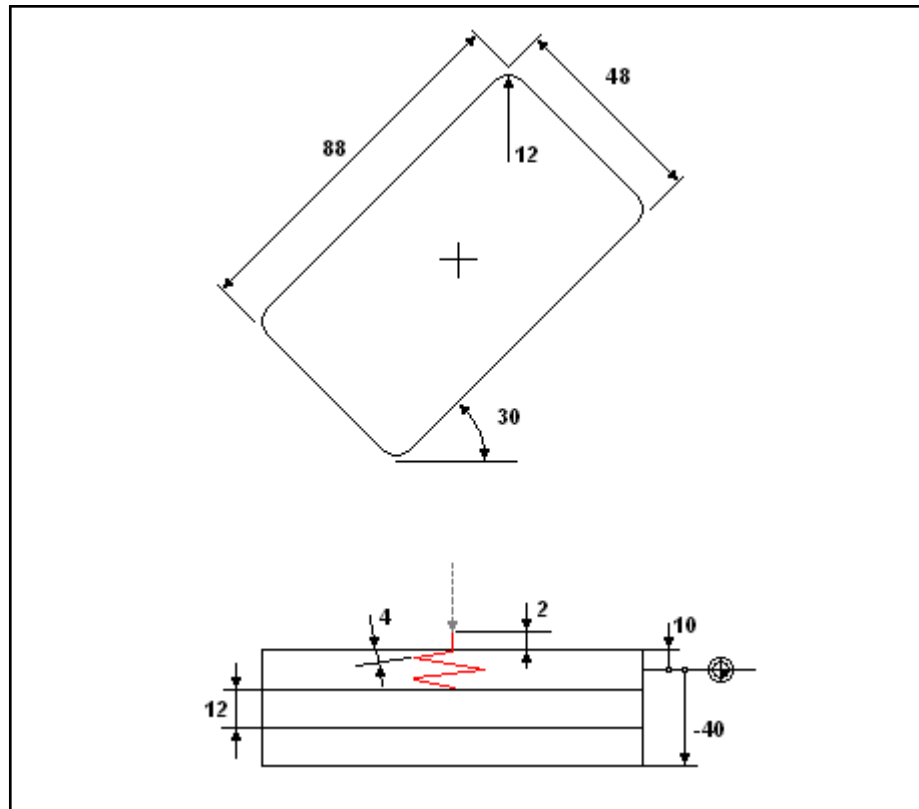


Fig. 7-19: Example of G271 Rectangle pocket milling

Program for G271 Rectangle pocket milling

```

N10 T1 M6 ; Tool change (milling cutter)
N20 RAD G90 G17 ; Plane
N30 G95 G97 S102 M4 ; Speed and feed settings
N40 G0 X110 Y110 Z2 F222 ; Prepositioning
//STOCK/BLOCK,300,300,300,0,0,300\\ [" "+NCF("G90")]
; Stock for simulation
//TOOL/MILL,18,0,80,0\\[" "+NCF("G90")] ; Tool for simulation
N50 G271 (CV2, ORI2, LE88, WI48, R12, RSW1, RSL1, PA30, ANG4, UL10, SDI2, DT-40, CD12, PC CD, CG80)
; Cycle definition
N60 X30 Y20 ; Cycle call
N70 G80 Z100 M5 ; Leave contour, delete modal call

```

7.8 G272 Circular pocket milling

The "Circular pocket milling" circle (G272) mills circular pockets. Depending on the direction of rotation of the tool, milling can be performed in clockwise or counterclockwise direction. The infeed is executed either in direction of the tool axis or as helical path.

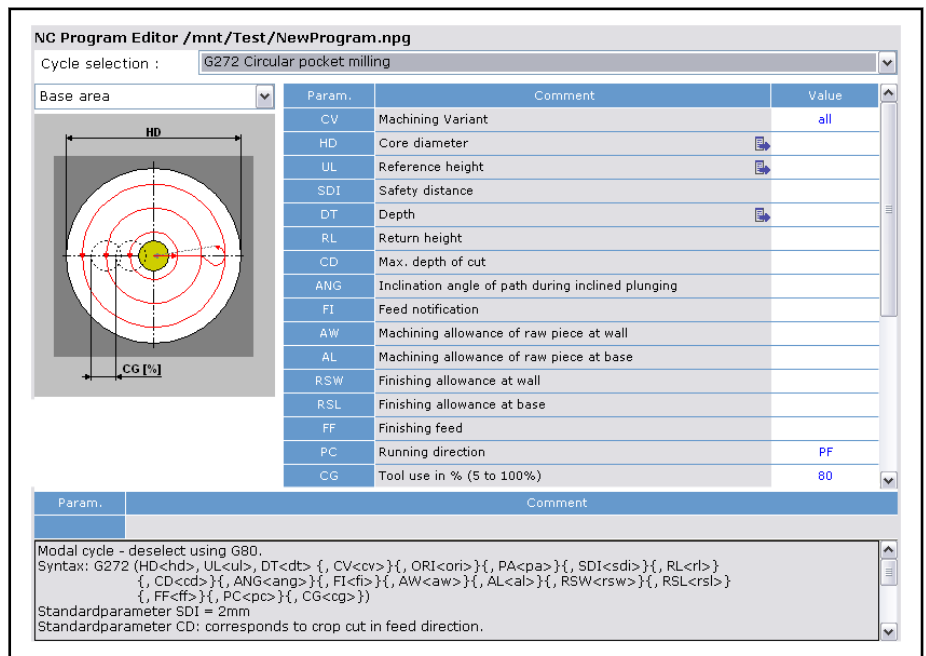


Fig. 7-20: G272 Circular pocket milling

G272		Circular pocket milling	G272(CV,HD,UL,SDI,DT,RL,CD,ANG,FI,AW,AL,RSW,RS L,FF,PC,CG)		
CV	SSysCyc.CV	Machining (cutting) variant	Integer	Optional	1 Roughing 2 Complete 3 Finishing 2 is the default setting
HD	SD.SysCyc.HD	Diameter	Real	Mandatory	Absolute
UL	SD.SysCyc.UL	Reference height	Real	Mandatory	Absolute
SDI	SD.SysCyc.SDI	Safety distance	Real	Optional	Incremental 2 mm is the default setting
DT	SD.SysCyc.DT	Depth	Real	Mandatory	Absolute
RL	SD.SysCyc.RL	Retraction height	Real	Optional	Absolute

Milling cycles

G272		Circular pocket milling	G272(CV,HD,UL,SDI,DT,RL,CD,ANG,FI,AW,AL,RSW,RS L,FF,PC,CG)		
CD	SD.SysCyc.CD	Max. infeed depth	Real	Optional	Incremental
ANG	SD.SysCyc.ANG	Inclination angle of path for helix	Real	Optional	1° to 45°
FI	SD.SysCyc.FI	Infeed feed	Real	Optional	>0 50 % of the latest programmed F-value by default
AW	SD.SysCyc.AW	Machining allowance of the unmachined part at the wall	Real	Optional	>0
AL	SD.SysCyc.AL	Machining allowance of the unmachined part at the base	Real	Optional	>0
RSW	SD.SysCyc.RSW	Finishing allowance at the wall	Real	Optional	>0
RSL	SD.SysCyc.RSL	Finishing allowance at the base	Real	Optional	>0
FF	SD.SysCyc.FF	Finishing feed	Real	Optional	>0 Latest programmed F-value by default
PC	SD.SysCyc.PC	Running direction	String	Optional	PF clockwise direction CD counterclockwise direction PF is preset
CG	SD.SysCyc.CG	Tool use	Real	Optional	5 to 100 % 80 % is preset

Sequence: The cycle supports planes G17, G18, G19 and radius allowance. The cycle runs modally and has to be deselected with G80. The tool is set in the plane on the starting position (center of the circle). From this position, the height of the safety distance **SDI** is approached in rapid traverse. Machining starts in infeed direction at the reference height **UL** and ends at the depth **DT**. A machining allowance at the base **AL** moves the machining start to this height. The finishing allowance **RSL** remains while roughing. The machining allowance **AW** at the wall moves the machining start out of the center if the free space in the center exceeds the milling cutter diameter.

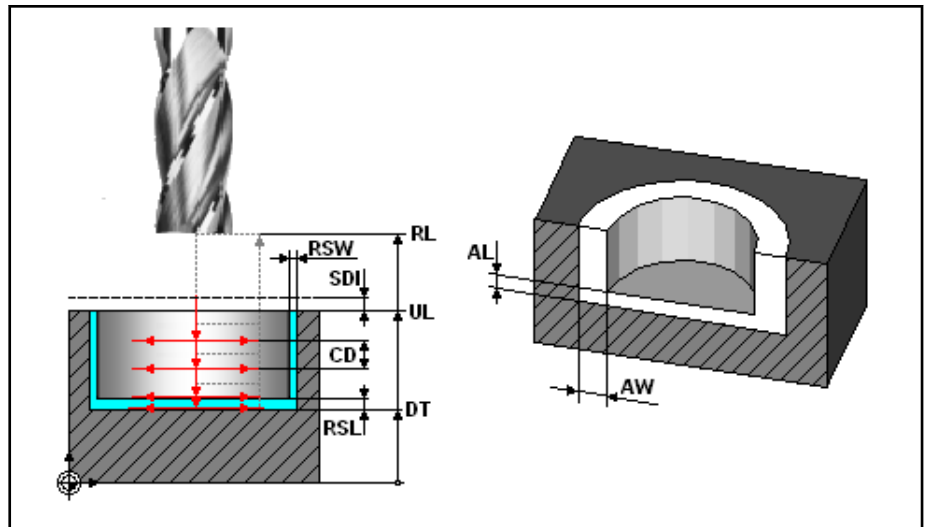


Fig. 7-21: Depth parameters, machining allowances and allowances for G272

It can be infeed in three different variants. If there is enough free space available for the tool (ratio of circle diameter, tool diameter and AW is considered), the infeed depth is approached in rapid traverse. If there is not enough free space available, it is traveled with the infeed feed to the infeed depth. If an inclination angle **ANG** is specified and there is not enough free space available, the infeed depth is reached by a helix-shaped infeed with the specified inclination. The diameter for the helix path is determined by the tool diameter.

If a maximum infeed depth **CD** is specified, the total infeed depth is segmented into equal infeeds (at the most maximum infeed depth). At the end of machining, the tool is traveled in rapid traverse to the reference height or, if this height is not specified, to the safety distance.

The machining variants are roughing, roughing and finishing or finishing (completely). Finishing requires at least one finishing allowance (RSW or RSL). Roughing is executed with a rectangular infeed in the plane and without edge radius correction. To finish the contour, an approach and run-out radius is added and an edge position correction is used.

If a machining allowance is specified (AW or AL), machining is restricted to these values.

Depending on the milling direction and the existing direction of the spindle rotation, machining takes place in a clockwise and counterclockwise direction.

The tool radius is determined from the NC (active tool required).

G40 (no edge radius correction) is to be active prior to the cycle. The cycle deactivates global transition elements (RND, RNE, CHF, and CHS).

Example:

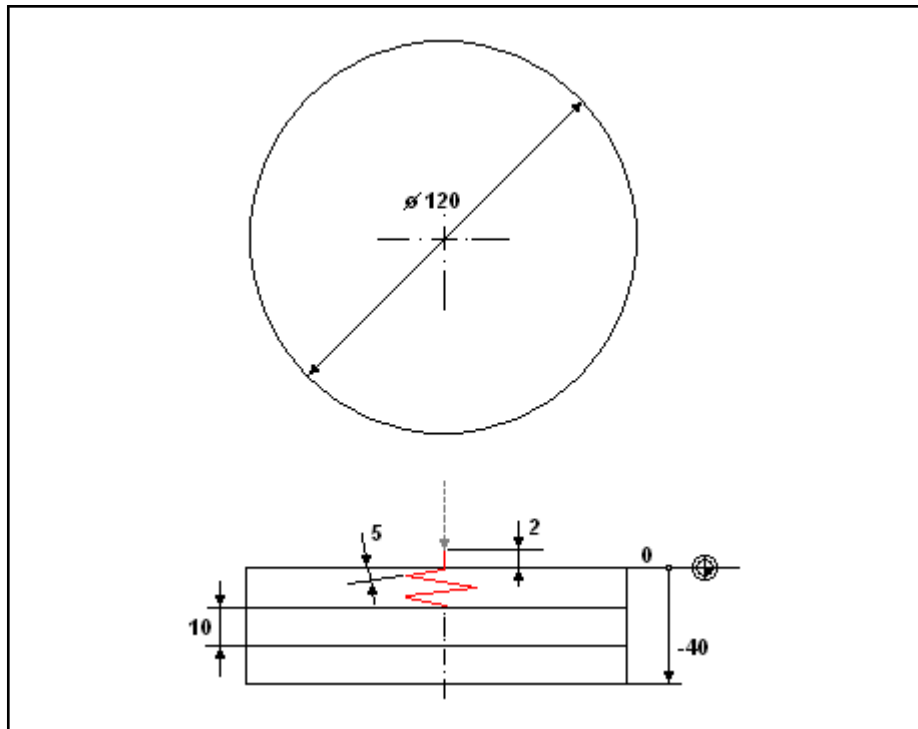


Fig. 7-22: Example of G272 Circular pocket milling
Program for G272 Circular pocket milling

```

N10 T1 M6 ; Tool change (milling cutter)
N20 RAD G90 G17 ; Plane
N30 G95 G97 S102 M4 ; Speed and feed settings
N40 G0 X110 Y110 Z2 F222 ; Prepositioning
//STOCK/BLOCK,300,300,300,0,0,300\\ [""+NCF("G90")]
; Stock for simulation
//TOOL/MILL,18,0,80,0\\[""+NCF("G90")] ; Tool for simulation
N50 G272(HD120, DT-40, UL0, FI100,ANG5,FF88,RSLO.5,RSW1,SDI2,PC CD,CD 10 )
; Cycle definition
N60 X30 Y20 ; Cycle call
N70 G80 Z100 M5 ; Leave contour, delete modal call
N80 G0 X100 Y100 Z10

```

7.9 G274 Straight slot milling

Cycle "G274 Straight groove milling" can be used to machine straight elongated holes and straight grooves. When elongated holes are machined, the tool diameter defines the internal width of the shape. When grooves are machined, several radial cuts can be executed (similar to pocket milling) and the groove widths can be made user-specifically (in excess of the tool diameter).

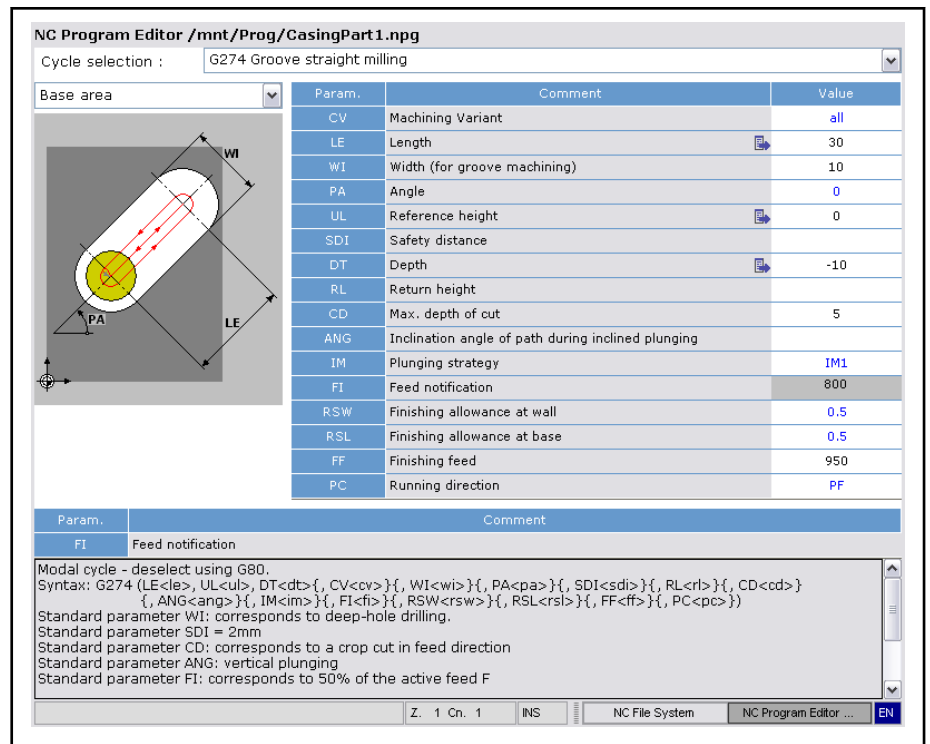


Fig. 7-23: G274 Straight slot milling

G274		Straight slot milling	G274(CV,LE,WI,PA,UL,SDI,DT,RL,CD,IM,ANG,FI,RSW,RSL,FF,PC)		
CV	SSysCyc.CV	Machining (cutting) variant	Integer	Optional	1 Roughing 2 Complete 3 Finishing 2 is the default setting
LE	SD.SysCyc.LE	Length	Real	Mandatory	Absolute >0
WI	SD.SysCyc.WI	Width	Real	Optional Specify for groove milling	Absolute ≥0
PA	SD.SysCyc.PA	Angle	Real	Optional	Absolute 0 is preset
UL	SD.SysCyc.UL	Reference height	Real	Mandatory	Absolute
SDI	SD.SysCyc.SDI	Safety distance	Real	Optional	Incremental 2 mm is the default setting
DT	SD.SysCyc.DT	Depth	Real	Mandatory	Absolute
RL	SD.SysCyc.RL	Retraction height	Real	Optional	Absolute
CD	SD.SysCyc.CD	Max. infeed depth	Real	Optional	Incremental >0

Milling cycles

G274		Straight slot milling	G274(CV,LE,WI,PA,UL,SDI,DT,RL,CD,IM,ANG,FI,RSW,RSL,FF,PC)		
IM	SD.SysCyc.IM	Infeed variant	Integer	Optional	1 Time-optimized 2 Infeed at first rounding 1 is preset
ANG	SD.SysCyc.ANG	Inclination angle of path during inclined plunging	Real	Optional	1° to 45°
FI	SD.SysCyc.FI	Infeed feed	Real	Optional	> 0 50 % of the latest programmed F-value by default
RSW	SD.SysCyc.RSW	Finishing allowance at the wall	Real	Optional	>0
RSL	SD.SysCyc.RSL	Finishing allowance at the base	Real	Optional	>0
FF	SD.SysCyc.FF	Finishing feed	Real	Optional	>0 Latest programmed F-value by default
PC	SD.SysCyc.PC	Running direction	String	Optional	PF clockwise direction CD counterclockwise direction PF is preset

Sequence: The cycle supports all interpolation planes and diameter programming of the x-axis. All coordinates (in this case only coordinates in the infeed axis) are absolute coordinates and are to be specified with the respective axis scaling (DIA/RAD). All remaining parameters are to be specified with radius dimension.

The cycle runs modally and has to be deselected with G80.

The center of the two rounding is the reference point and has to be approached to execute the cycle. The position of the center of the second rounding is defined by the rotation angle **PA** and the distance between the two rounding centers, i.e., the length **LE**.

In addition to the radial dimensions, the machining task is described by two coordinates in infeed direction, which define the position of the upper side (**UL**) and the base (**DT**). During roughing, the cuts are segmented between these two coordinates with a uniform depth of cut (**CV** 1 or 2) which can be limited with **CD**. If **CD** is not used, only one roughing cut is executed in infeed direction.

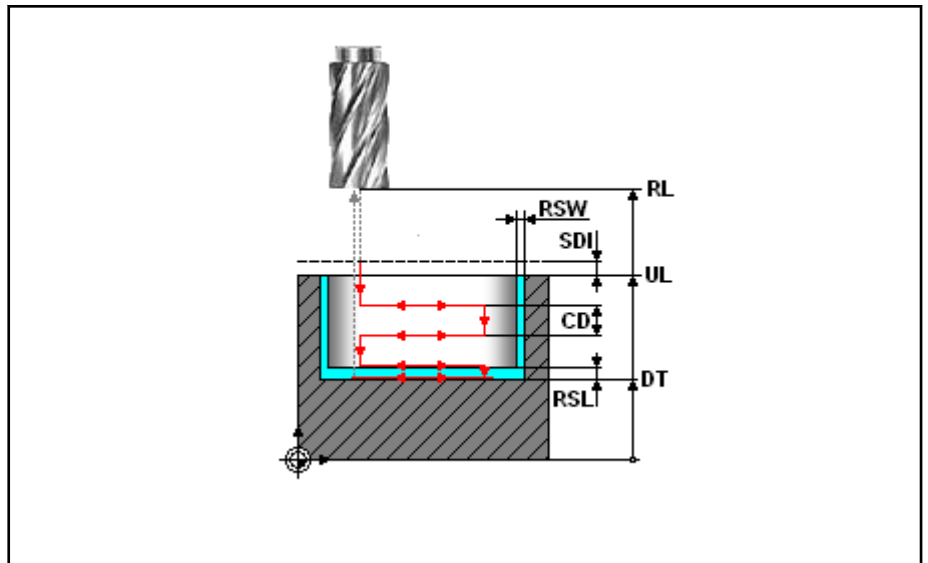


Fig. 7-24: Depth parameters and finishing allowances for G274

Infeed is vertical in plunge feed **FI**, either alternating at the centers of both roundings or always in the center of the first rounding. Infeed variant **IM** can be used to allow alternating infeed. If necessary, inclined infeed along the path between the two rounding centers can be activated with the plunging angle **ANG**. The parameters **IM** and **FI** do not have any effect on inclined plunging.

Each cut in the segmentation in infeed direction consists of at least one milling step along the path connecting the rounding centers (machining of elongated holes). If the width **WI** is greater than the tool diameter, this is followed by radial cuts until the width is reached (machining of grooves).

The direction of tracing out the radial cuts is defined with regard to the direction of spindle rotation and can be influenced by **PF**.

After a chain of radial cuts has been completed, the tool is lifted by the safety distance **SDI** in infeed direction and, if necessary, infeed again for the next cut. After cutting has been completed, the tool moves to the return position **RL** or across the reference plane **UL** by **SDI** and then again reaches the initial coordinates in the interpolation plane which were approached before the cycle was called.

Finishing requires a wall-sided finishing allowance **RSW** (wall finishing) and/or a finishing allowance on the bottom **RSL** (bottom finishing). Finishing takes always place in infeed direction in a single cut. In radial direction (groove machining), finishing can also comprise several cuts. A special feed **FF** can be activated for finishing. If the only machining step is finishing (**CV3**), **RSW** and/or **RSL** have also to be specified, as these parameters define the initial machining dimensions.

G40 (no edge radius correction) is to be active prior to the cycle. The cycle deactivates global transition elements (**RND**, **RNE**, **CHF**, and **CHS**).

Example 1: Two continuous elongated holes are to be produced. Their width is determined by the diameter of the milling cutter. It is infeed on a path inclined by no more than 16°.

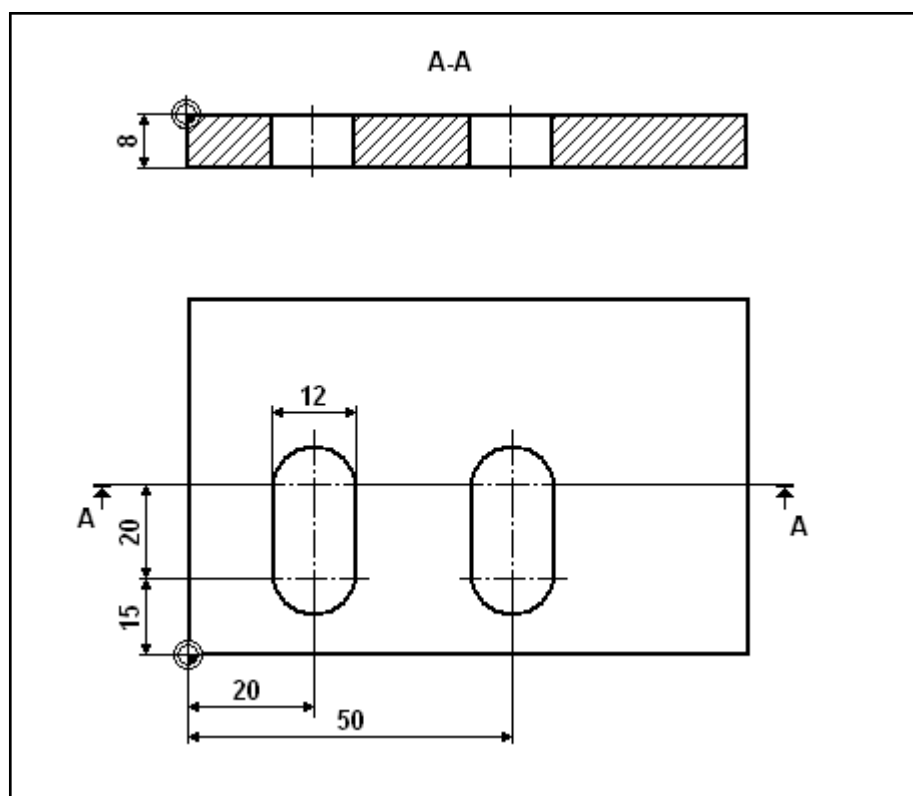


Fig. 7-25: Example 1: G274 Machining elongated holes

Program for G274 Machining elongated holes

```

N10 T1 M6; End milling cutter D 12
N20 RAD G17 G90 G95 G97 S4500 M3
N30 G274 (LE20, PA90, UL0, DT-9, ANG16) ; Cycle "Curved elongated hole"
N40 G0 X20 Y15 Z10 F1200 ; Approach 1st reference position and execute cycle
N50 X50 ; Approach 2nd reference position and execute cycle
N60 G80 X200 Y200 Z200 M5 ; Leave contour, delete modal call

```

Example 2: Four radially arranged keyways are to be machined. A diameter-8 groove milling cutter is to be used for machining which also includes finishing on all sides (finishing allowance 0.5 mm). The depth of cut is limited to 5 mm.

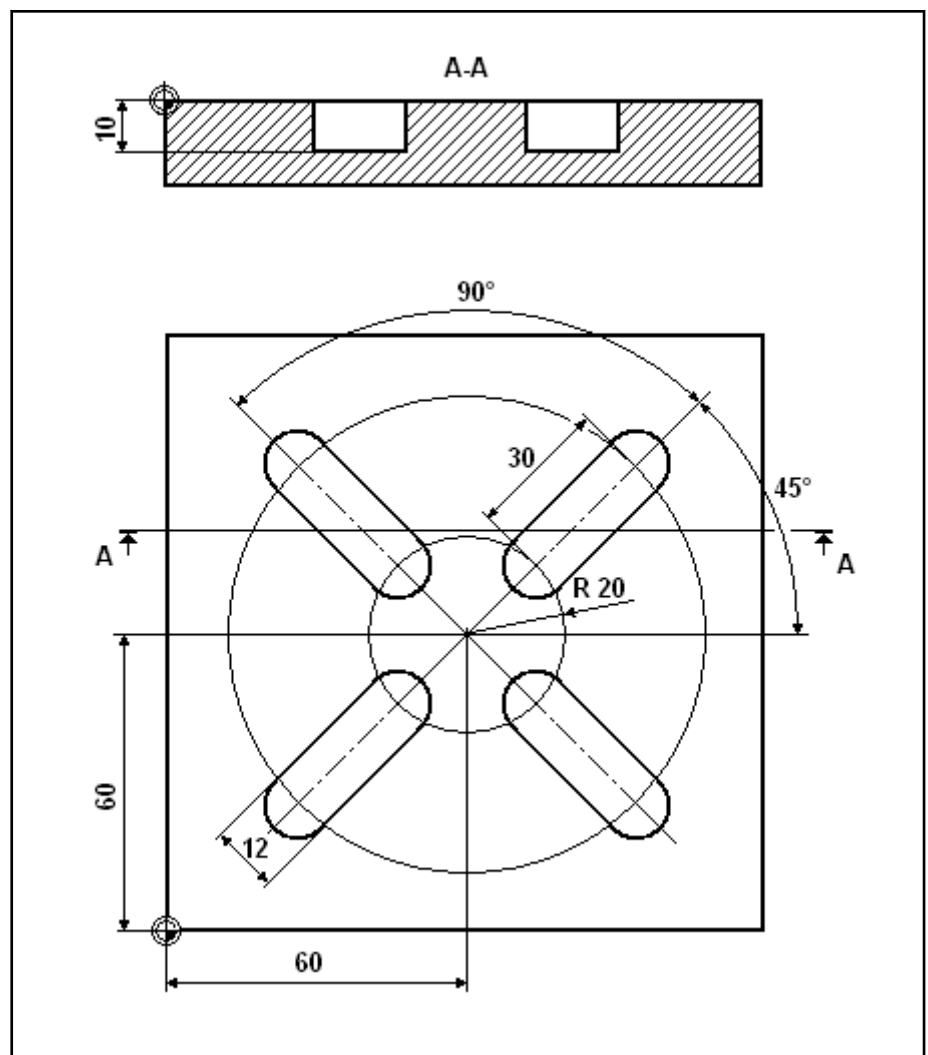


Fig. 7-26: Example 2: G274 Machining grooves
Program for G274 Machining grooves

```

N10 T1 M6; Groove milling cutter D 8
N20 RAD G17 G90 G95 G97 F1200 S5600 M3
N30 G274 (LE30, WI10, UL0, DT-10, CD5, RSW0.5, RSL0.5, FI800, FF950); Cycle "Groove"
N40 G113 (MCP60, SCP60, PR20, NH4, BA45, ROT2) ; Execute cycle on partial circle
N50 G80 X200 Y200 Z200 M5 ; Leave contour, delete modal call

```

7.10 G275 Curved slot milling

- Syntax** G275(CV,ORI,WI,R,PA,OA,UL,SDI,DT,RL,CD,ANG,IM,FI,RSW,RSL,FF,PC)
- Brief description** The cycle "G275 Curved groove milling" (G275) can machine elongated holes and grooves which follow the curvature of an arc.
- **Elongated hole machining:**
The tool only moves along the curved center line. The groove width corresponds to the tool diameter
 - **Groove machining:**
Arbitrary groove widths greater than the tool diameter are created due to radial cuts with milling cutter radius compensation

Overview

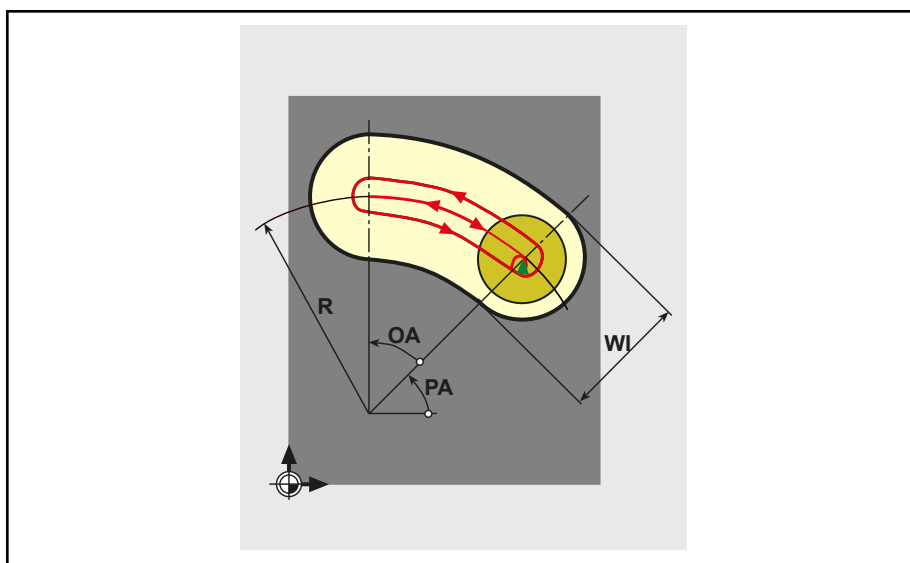


Fig. 7-27: G275 Curved slot milling

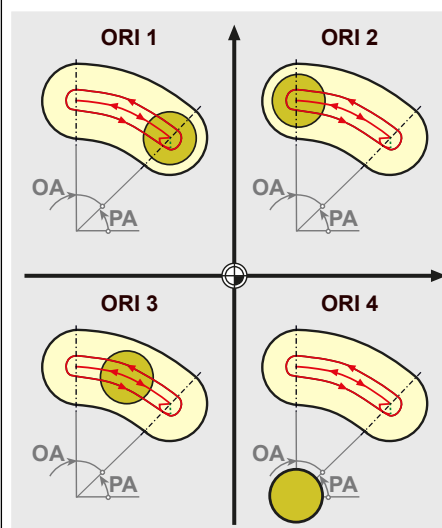
Parameter

Parameter	Designation	Properties	Values	Pre-assignment
CV	Machining (cutting) variant	Optional	Roughing (1) / Complete (2) / Finishing (3)	2
ORI	Reference point	Optional		1
WI	Width	Optional Specify for groove milling	>0	Current tool diameter
R	Radius	Mandatory	>0	
PA	Starting angle	Optional	Degree	0
OA	Opening angle	Mandatory	Incremental Degree <>0	
UL	Reference height	Mandatory	Absolute	
SDI	Safety distance	Optional	Incremental ≥0	2 mm
DT	Depth	Mandatory	Absolute	
RL	Retraction height	Optional	Absolute	Initial position of infeed axis in front of the cycle
CD	Maximum in-feed depth	Optional	Incremental >0	One-time infeed until finishing allowance RSL

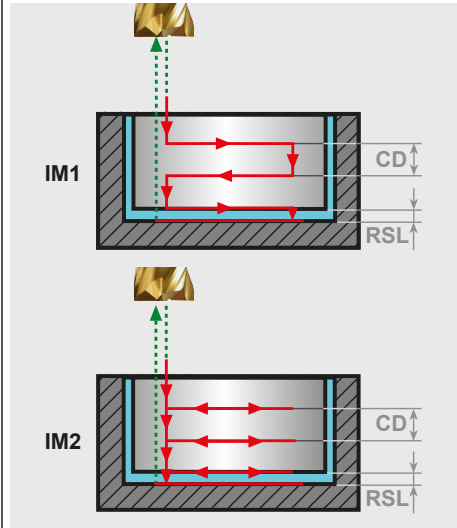
ANG	Inclination angle of path during inclined plunging	Optional	$0 < \text{ANG} \leq 45$	Vertical plunging
IM	Plunging strategy	Optional		1
FI	Feed infeed	Optional	> 0	50% of the current feed
RSW	Finishing allowance at the wall	Optional	≥ 0	0
RSL	Finishing allowance at the base	Optional	≥ 0	0
FF	Finishing feed	Optional	> 0	Current feed
PC	Running direction	Optional	Climb milling (PF)/up-cut milling (CD)	Climb milling (PF)

Tab. 7-11: Parameter call

ORI	Effect
1	Reference point is the rounding center point, referring to PA
2	Reference point is the rounding center point, referring to OA
3	Reference point to bisecting line of OA
4	Reference point is the center of curvature (radius R)



Tab. 7-12: Description of the parameter ORI - Reference point

IM	Effect
1	Infeed at both rounding center points for time-optimized machining
2	Infeed at a rounding center point • ORI<>2: Infeed at rounding center point, referring to PA • ORI=2: Infeed at rounding center point, referring to OA
	

Tab. 7-13: Description of the parameter IM - Plunging strategy

Description

CV Machining (cutting) variant

Selecting pre-machining and fine machining

ORI Reference point

Selecting a point at the contour element to which the machining geometry refers to and which has to be approached for calling the modal cycle

WI Width

Width of the slot

R Radius

Center of curvature of the center line of the groove

PA Starting angle

Absolute angle from the center point of the curvature to the center point of a groove rounding

OA Opening angle

Relative angle between both center points of roundings, enclosing the groove

UL Reference height

The topside is located at this absolute position in infeed direction

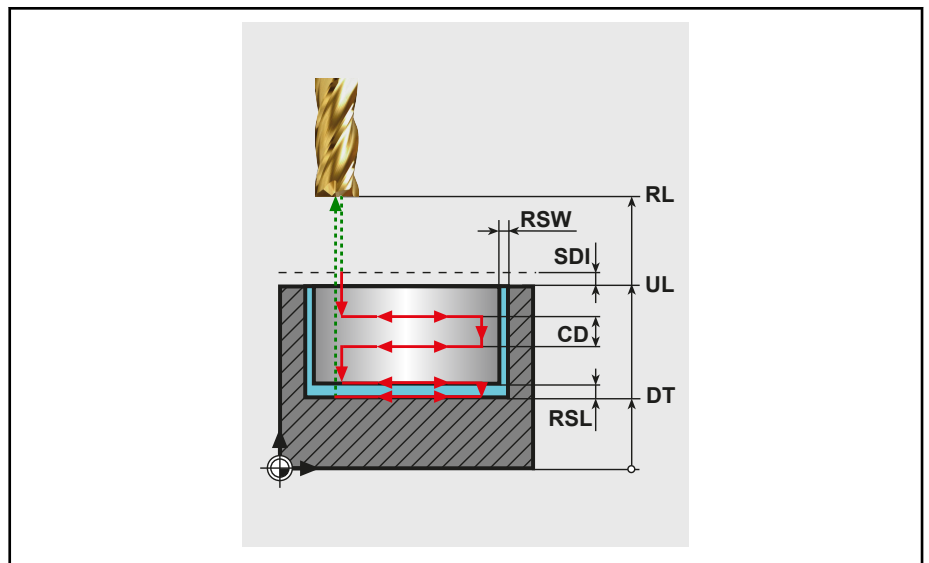


Fig. 7-28: Cycle parameter in infeed direction and finishing allowances

SDI Safety distance

Distance between tool and workpiece in radial direction or direction of the in-feed. If they are fallen below, it is always traveled in feed

DT Depth

At this absolute position in infeed direction, the base of the groove is located or the tool during the last cut in case of a (continuous) elongated hole

RL Retract height

This absolute position is the infeed axis coordinate at the end of the cycle

CD Maximum infeed depth

Maximum distance between two roughing cuts in infeed direction

ANG Inclination angle of path during inclined plunging

Maximum angle between tool path and interface for reciprocating plunging (only roughing)

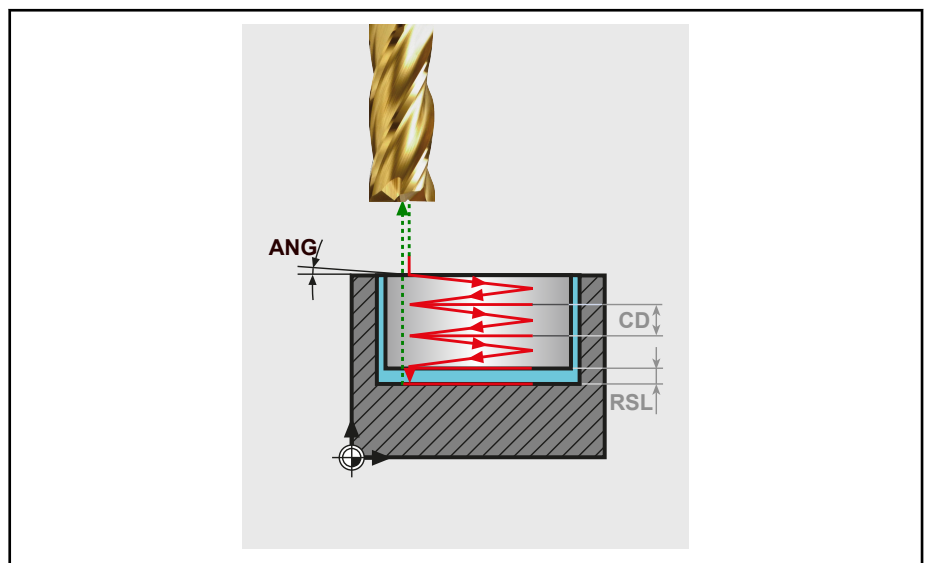


Fig. 7-29: Plunging angle ANG

IM Plunging strategy

	Only specifies if infeed can take place at a certain end of the groove This is important, if rough-drilling is only to be used at one end of the groove Does not have any effect in case of inclined plunging with ANG FI Feed infeed Special feed for vertical infeed/plunging RSW Finishing allowance at the wall Radial allowance during roughing, remaining as machining allowance for finishing Only active during groove milling ($WI > \text{tool diameter}$) Finishing allowance at the base Allowance at groove base during roughing which remains as machining allowance for finishing Do not specify if it is an elongated hole without base FF Finishing feed Special feed for finishing PC Running direction Switches between parallel feed milling and counter direction milling and influences the contour direction of radial cuts depending on the direction of rotation of the spindle
Special features	Note that a modal cycle is not executed with the cycle call during programming, but only after the next traversing command. G40 (no edge radius correction) is to be active prior to the cycle. The cycle deactivates global transition elements (RND, RNE, CHF, and CHS). In case of only finishing finish cut(CV3), at least one of the finishing allowances RSW and/or RSL has to be greater than zero, as these parameters define the initial machining dimensions.
Procedure/processing	Starting from the reference point (parameter ORI) which is to be approached to execute the cycle, it is positioned to the starting point of machining (results from parameters IM and ORI). The groove depth between UL and DT minus RSL is distributed into uniform cuts with a maximum depth of CD during roughing (CV1 and CV2). If CD is not specified, only one roughing cut is executed in infeed direction. Infeed is possible vertically in plunging feed FI or via an inclined position along a helical path between the two curve roundings which can be activated with the plunging angle ANG. Each cut of the segmentation in infeed direction consists of at least one milling step along the center line of the curvature (machining of elongated holes). If the width WI is greater than the tool diameter, this is followed by radial cuts until the width is reached (machining of grooves). After a chain of radial cuts has been completed, the tool is lifted by the safety distance SDI in infeed direction and, if necessary, infeed again for the next cut which can take place at the other side of the groove for IM2. Finishing requires a wall-sided finishing allowance RSW (wall finishing only in case of groove machining) and/or a finishing allowance on the bottom RSL (bottom finishing). Finishing while feed FF is active always takes place in infeed direction in a single cut with exclusively vertical plunging. In radial direction, finishing can also comprise several cuts. At the end of the cycle, it is retracted to RL and the initial position before the cycle is traversed.

- Example 1** A continuous curved elongated hole is to be milled. Its width is determined by the diameter of the milling cutter. It is infed on a path inclined by no more than 12° . Machining has to take place path-optimized without ORI4 which, however, would be appropriate taking the dimensions into consideration.

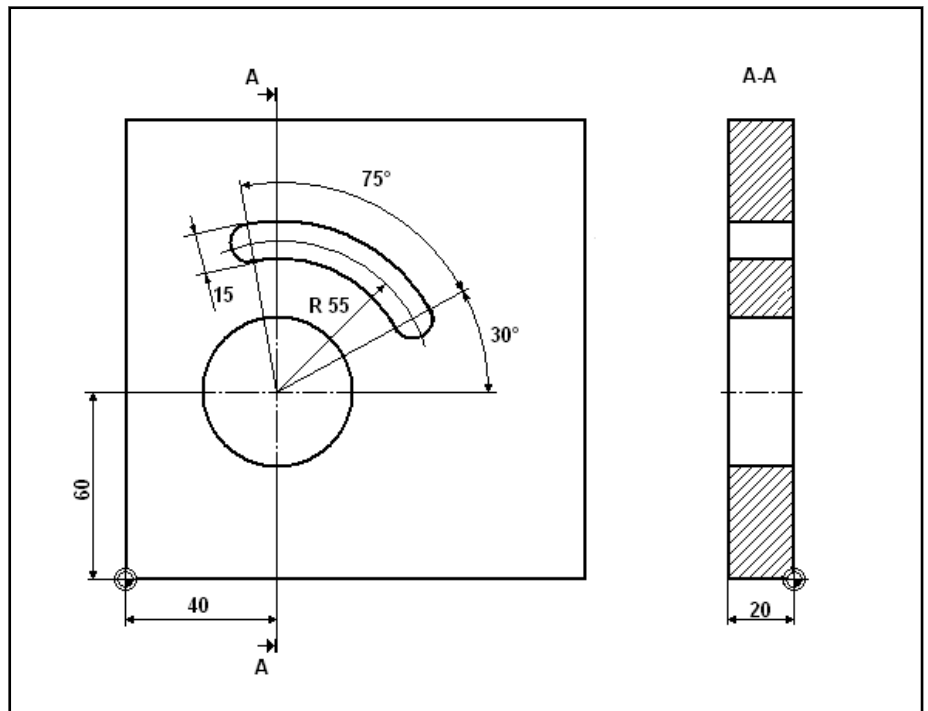


Fig. 7-30: Example 1: G275 Machining elongated holes

Program for G275 Machining elongated holes

N10 T1 M6	; End mill D 15
N20 RAD G17 G90 G95 G97 S5200 M3	; Technology values
N30 G275 (R55, PA30, OA75, UL0, DT-21, ANG12)	; Cycle Milling curved elongated hole cycle
N40 G0 X[40+55*COS(30)] Y[60+55*SIN(30)] Z10 F1100	; Move to reference position and execute cycle
	; calculation with CPL for path optimization
G0 G80 X200 Y200 Z200 M5N50 G0 G80 X200 Y200 Z200 M5	; Leave contour, delete modal call

- Example 2** Three concentrically arranged round slots are to be milled. A groove milling cutter with a diameter of 10 is to be used for machining which also includes finishing on all sides (finishing allowance 0.5 mm). The depth of cut is limited to 5 mm.

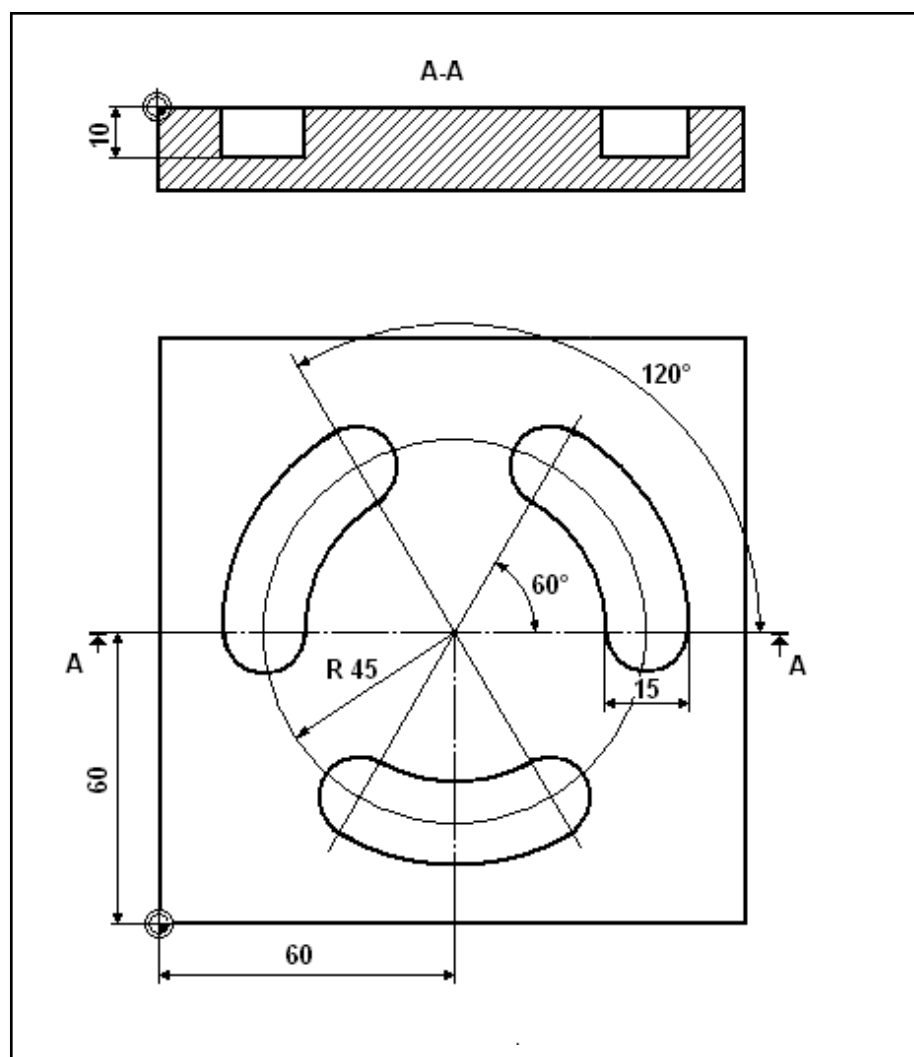


Fig. 7-31: Example 2: G275 Machining grooves

Program for G275 Machining grooves

```

N10 T1 M6 ; Slot milling cutter D 10
N20 RAD G17 G90 G95 G97 F1200 S4800 M3 ; Technology values
N30 G275 (WI15, R45, OA60, UL0, DT-10, CD5, FI800, RSW0.5, RSL0.5, FF900) ; Cycle Milling curved slot
N40 G113 (MCP60, SCP60, PR45, NH3, ROT2, ROT2) ; Execute cycle on pitch circle
N50 G0 G80 X200 Y200 Z200 M5 ; Delete modal call and leave contour

```

7.11 G276 Rectangular pin milling

Syntax G276(CV,ORI,LE,WI,R,PA,UL,SDI,DT,RL,CD,RD,AW,AL,RSW,RSL,FE,FF,P C,CG)

Brief description Using the cycle "G276 Rectangular pin milling", rectangular spigots including the plane surface can be machined.

Overview

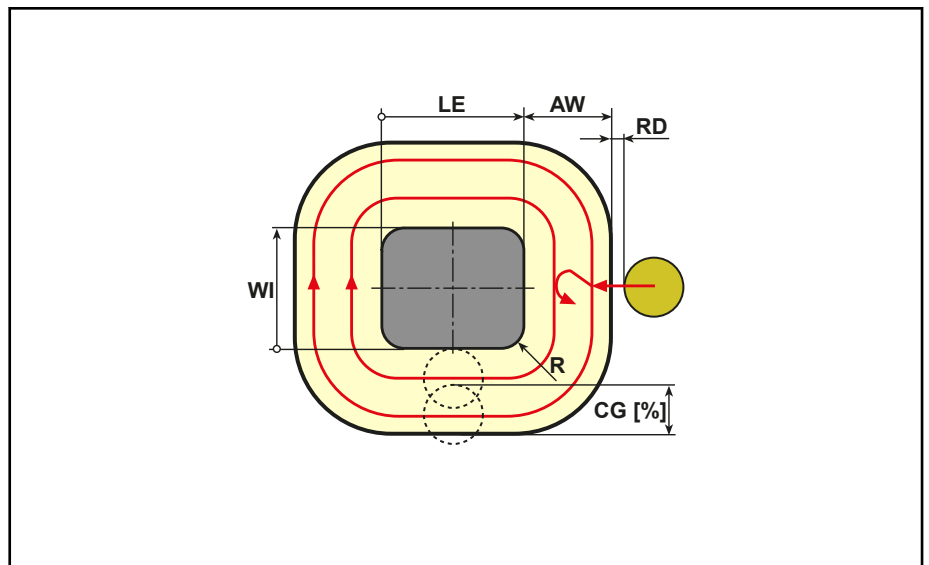


Fig. 7-32: G276 Rectangular pin milling

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
CV	Machining (cutting) variant	Optional	Roughing (1) / Complete (2) / Finishing (3)	2
ORI	Reference point	Optional		1
LE	Length	Mandatory	Incremental <>0	
WI	Width	Mandatory	Incremental <>0	
R	Corner radius	Optional	>=0	0
PA	Angle	Optional	Degree	0
UL	Reference height	Mandatory	Absolute	
SDI	Safety distance	Optional	Incremental >=0	2 mm
DT	Depth	Mandatory	Absolute	
RL	Retraction height	Optional	Absolute	Initial position of infeed axis in front of the cycle
CD	Maximum in-feed depth	Optional	Incremental >0	One-time in-feed until finishing allowance RSL
RD	Safety distance in plane	Optional	>=0	2 mm

Milling cycles

AW	Machining allowance of the unmachined part at the wall	Optional	>0	CG% of the tool diameter
AL	Machining allowance of the unmachined part at the base	Optional	>0	Uniform machining allowance between UL and DT
RSW	Finishing allowance at the wall	Optional	>=0	0
RSL	Finishing allowance at the base	Optional	>=0	0
FE	First element	Optional	1..4	1
FF	Finishing feed	Optional	>0	Current feed
PC	Running direction	Optional	Climb milling (PF)/up-cut milling (CD)	Climb milling (PF)
CG	Tool use	Optional	0<CG<=100, Percent of the tool diameter	80

Tab. 7-14: Parameter call

ORI	Effect
1	Reference point is a corner
2	Reference point is the center

The diagram shows two coordinate systems with X and Y axes. In the top system, labeled 'ORI 1', a rectangular workpiece is shown with a reference point (yellow dot) at one of its corners. Dimensions LE (Lead edge) and WI (Workpiece inner corner) are indicated. In the bottom system, labeled 'ORI 2', the same workpiece is shown with the reference point at its center. The labels LE, WI, and PA (Part axis) are also present in this diagram.

Tab. 7-15: Description of the parameter ORI - Reference point

FE	Effect
1	Start at element, a positive LE is referencing to
2	Start at element, a positive WI is referencing to
3	Start at the element opposite to FE1
4	Start at the element opposite to FE2

Tab. 7-16: Description of the parameter FE - First element

Description

CV Machining (cutting) variant

Selecting pre-machining and fine machining

ORI Reference point

Selecting a point at the contour element to which the machining geometry refers to and which has to be approached for calling the modal cycle

LE Length

Length of the spigot

The reference point can be positioned at a different corner as described using the sign.

WI Width

Width of the spigot

The reference point can be positioned at a different corner as described using the sign.

R Corner radius

Rounding radius

PA Angle

Orientation angle in positive x-direction

UL Reference height

The topside is located at this absolute position in infeed direction

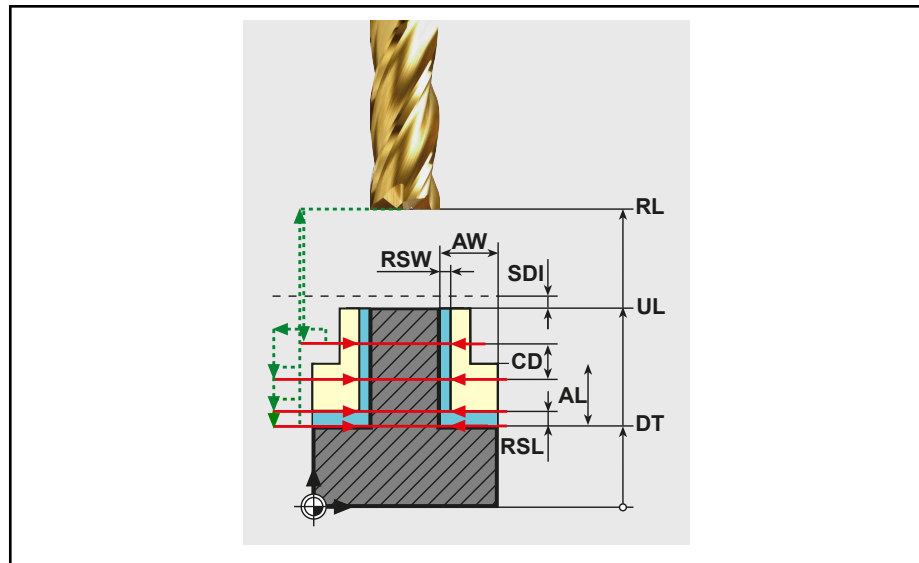


Fig. 7-33: Cycle parameter in infeed direction, dimensions and machining allowances

SDI Safety distance

Distance between tool and workpiece in direction of the infeed. If the distance is fallen below, it is always traveled in feed

DT Depth

The plane surface where the spigot starts is located at this absolute position in infeed direction

RL Retract height

This absolute position is the infeed axis coordinate at the end of the cycle

CD Maximum infeed depth

Maximum distance between two roughing cuts in infeed direction

RD Safety distance in plane

Minimum distance between tool and workpiece that has to be complied with during positioning in the interpolation level

Machining allowance of the unmachined part at the wall

Radial machining allowance on the lateral surface of the spigot

AW applies to roughing. AW specifies the width of the plane level (base) to be machined during finishing

AL Machining allowance of the unmachined part at the base

Machining allowance on the plane surface where the spigot starts

AL applies to roughing. If AL is specified, there is a non-uniform machining allowance AW along the height. The standard machining allowance that can be cut with a radial cut applies above AL

RSW Finishing allowance at the wall

Radial allowance during roughing, remaining as machining allowance for finishing

Finishing allowance at the base

Allowance on the adjacent plane surfaces during roughing, remaining as machining allowance for finishing

FE First element

Index of the contour element at whose center each radial cut starts and ends

FF Finishing feed

Special feed for finishing

PC Running direction

Switches between parallel feed milling and counter direction milling and influences the contour direction depending on the direction of rotation of the spindle

CG Tool use

Maximum contact width of the tool in percent of the diameter

Special features

Note that a modal cycle is not executed with the cycle call during programming, but only after the next traversing command.

G40 (no edge radius correction) is to be active prior to the cycle. The cycle deactivates global transition elements (RND, RNE, CHF, and CHS).

The depth infeed is executed outside in rapid traverse. Therefore, a material-free area with a minimum width resulting from the sum of milling cutter diameter, safety distance (parameter RD) and wall-sided machining allowance (parameter AW) around the spigot has to be ensured.

In case of only finishing (CV3), at least one of the finishing allowances RSW and/or RSL has to be greater than zero, as these parameters define the initial machining dimensions.

The feed FF and the current feed are taken into consideration as effective feeds. The cycle automatically adjusts the feed according to the current path curvature and the effective contact width.

Process/processing

The land depth between UL and DT minus RSL is distributed into uniform cuts with a maximum depth of CD during roughing (CV1 and CV2). If CD is not specified, only one roughing cut is executed in infeed direction. Depending on the machining allowance, each cut in infeed direction can be segmented in several cuts in parallel to the contour (radial cut segmentation - external part to internal part). The equidistance between these cuts is limited by the tool use.

While approaching, the milling cutter center is positioned beyond the center of the first element FE by the radial distance resulting from the tool radius, radial safety distance RD and the radial machining allowance and the tool is moved to the infeed depth in rapid traverse. If the safety distance SDI to the workpiece is fallen below, it is traveled in feed. This is followed by a radial radial infeed and an equidistant tracing out of the pin. This, in turn, is followed by further radial infeeds and tracing outs until finishing allowance RSL is reached during roughing or until the final size of the pin is reached during finishing. One approach and run-out arc are added when milling the finished dimension.

After a chain of radial cuts has been completed, the tool is lifted by the amount of safety distance SDI in infeed direction and the next cut of the segmentation can be made in infeed direction.

After cutting has been completed, the tool moves to the return position RL and reaches the initial coordinates in the interpolation plane which were approached before the cycle was called.

Example

A rectangular pin is to be machined by roughing and finishing. During roughing, the maximum depth of cut is 8 mm. A wall-sided allowance of 1 mm and an allowance on the bottom of 0.8 mm is to be kept for finishing.

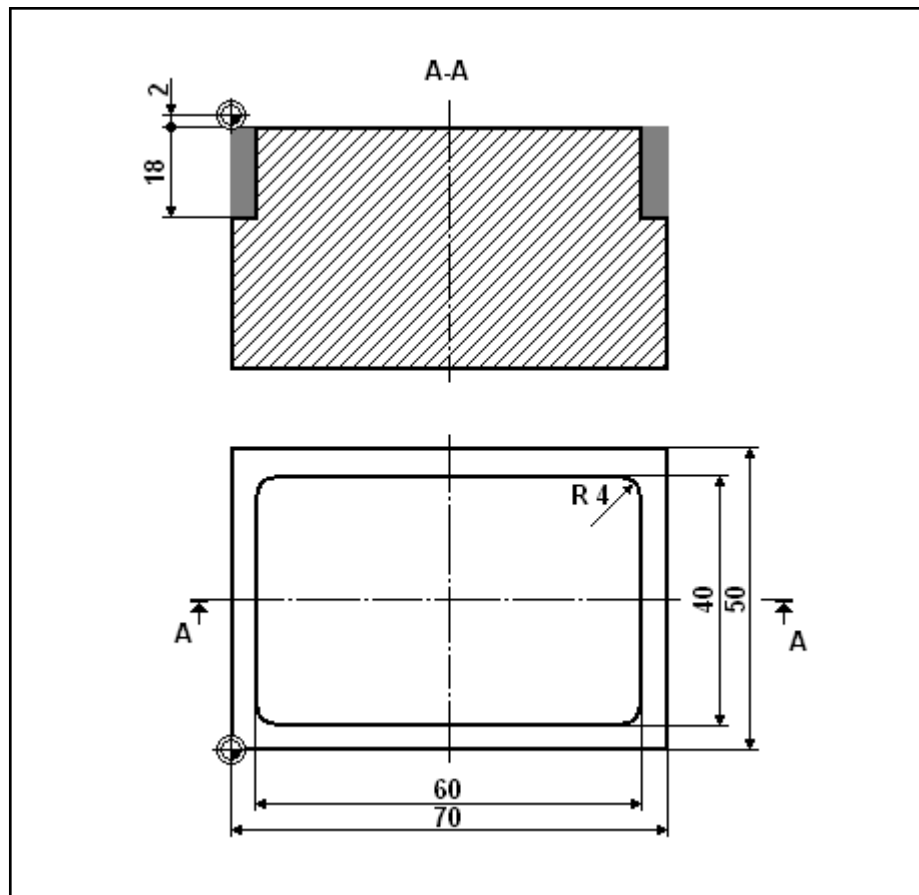


Fig. 7-34: Example of G276 Rectangular pin milling
Program for G276 Rectangular pin milling

```

N10 T1 M6 ; tool change
N20 RAD G90 G17 ; Program coordinate system
N30 G95 G97 F1000 S1000 M3 ; technology values
N40 G0 Z200 ; prepositioning
N50 G276 (CV2, ORI1, LE60, WI40, R4, PA0, UL-2, SDI3, DT-20, RL0, CD8,
        RD2, AW5, AL18, RSW1, RSL0.8, FF790, PC PF, CG80)
        ; rectangular pin cycle
N60 X5 Y5 ; Move to reference position and execute cycle
N70 G80 X200 Y200 Z200 M5 ; Leave contour, delete modal call

```

7.12 G277 Circular pin milling

The cycle "G277 Circular spigot milling" can machine spigots with a circular cross section with the lateral surface and the plane surface being milled at the spigot face (called bottom). It is distinguished between roughing (several cuts both in infeed direction and radial direction) and finishing (single wall milling, if necessary several radial cuts on the bottom). The depth infeed is executed outside in rapid traverse. Therefore, a material-free area with a minimum width resulting from the sum of milling cutter diameter, safety distance (parameter RD) and wall-sided machining allowance (parameter AW) around the circle has to be ensured.

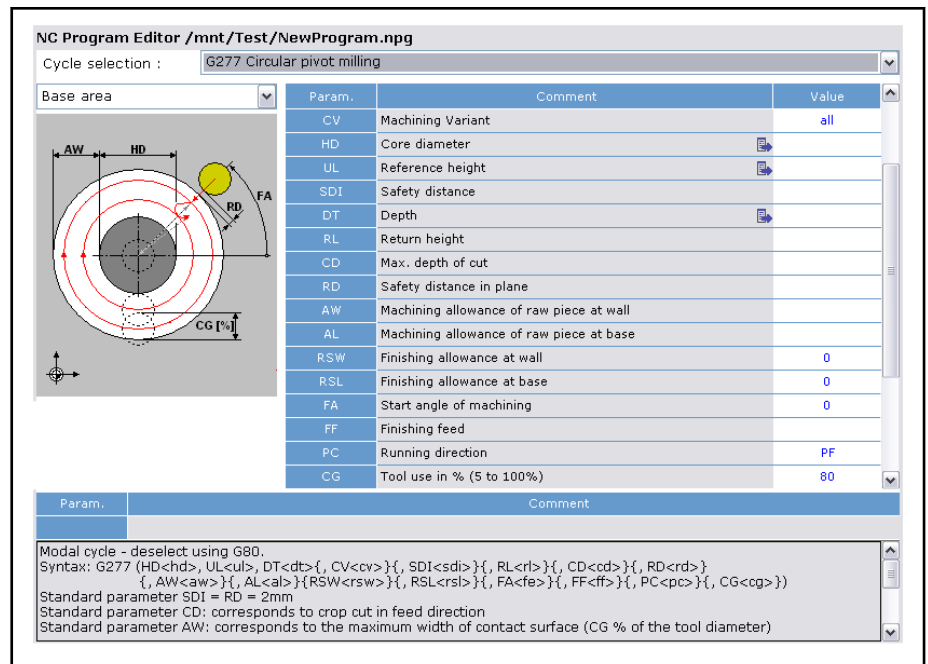


Fig. 7-35: G277 Circular pin milling

G277		Circular pin milling	G277(CV,HD,UL,SDI,DT,RL,CD,RD,AW,AL,RSW,RSL,F A,FF,PC,CG)		
CV	SD.SysCyc.CV	Machining (cutting) variant	Integer	Optional	1 Roughing 2 Complete 3 Finishing 2 is the default setting
HD	SD.SysCyc.HD	Diameter	Real	Mandatory	>0
UL	SD.SysCyc.UL	Reference height	Real	Mandatory	Absolute
SDI	SD.SysCyc.SDI	Safety distance	Real	Optional	>=0 Default: 2 mm
DT	SD.SysCyc.DT	Depth	Real	Mandatory	Absolute
RL	SD.SysCyc.RL	Retraction height	Real	Optional	Absolute
CD	SD.SysCyc.CD	Max. infeed depth	Real	Optional	>0 One roughing cut in infeed direction by default
RD	SD.SysCyc.RD	Safety distance in plane	Real	Optional	>=0 Default: 2 mm
AW	SD.SysCyc.AW	Machining allowance of the un-machined part at the wall	Real	Optional	>0
AL	SD.SysCyc.AL	Machining allowance of the un-machined part at the base	Real	Optional	>0
RSW	SD.SysCyc.RSW	Finishing allowance at the wall	Real	Optional	>=0
RSL	SD.SysCyc.RSL	Finishing allowance at the base	Real	Optional	>=0

Milling cycles

G277		Circular pin milling	G277(CV,HD,UL,SDI,DT,RL,CD,RD,AW,AL,RSW,RSL,F A,FF,PC,CG)		
FA	SD.SysCyc.FA	Start angle of machining	Real	Optional	Absolute 0 is preset
FF	SD.SysCyc.FF	Finishing feed	Real	Optional	>0 Latest programmed F-value by default
PC	SD.SysCyc.PC	Running direction	String	Optional	PF clockwise direction CD counterclockwise direction PF is preset
CG	SD.SysCyc.CG	Tool use	Real	Optional	5 to 100 % 80 % is preset

Sequence: The cycle supports all interpolation planes and the diameter programming of the X-axis. All coordinates (in this case only coordinates in the infeed axis) are absolute coordinates and are to be specified with the respective axis scaling (DIA/RAD). All remaining parameters are to be specified with radius dimension.

The cycle runs modally and has to be deselected with G80.

The position of the center of the pin to be machined with diameter **HD** is determined by the coordinates in the interpolation plane approached to execute the cycle.

In addition to the radial dimensions, the machining task is described by two coordinates in infeed direction, which define the position of the upper side (**UL**) and the base (**DT**). During roughing, the cuts are segmented between these two coordinates with a uniform depth of cut (**CV** 1 or 2) which can be limited with **CD**. If **CD** is not used, only one roughing cut is executed in infeed direction.

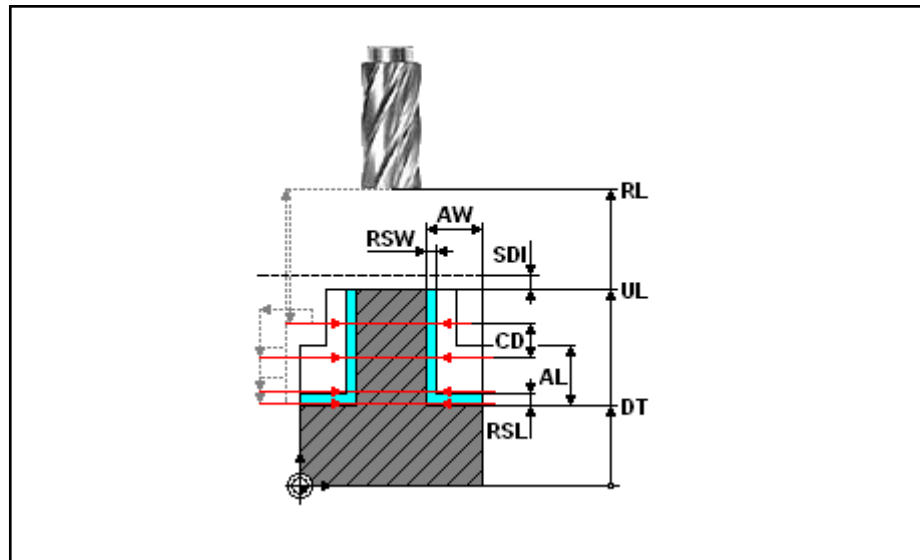


Fig. 7-36: Depth parameters, machining allowances and allowances for G277

Each cut in infeed direction can be segmented in several cuts in parallel to the contour (radial cut segmentation from without inward). The equidistance between these cuts is limited by the tool diameter multiplied with the maximum tool use **CG**. The direction of tracing out is defined with regard to the

direction of spindle rotation and can be influenced by **PF**. Radial infeed is made on the ray that exits the center of the circle with start angle **FA**.

The machining allowance at the wall **AW** describes the initial state of the raw piece. If this parameter is not used, a machining allowance is assumed which can be machined with a single radial cut, i.e., allowance equal to the maximum tool use **CG** multiplied with the tool diameter. Since it defines the radial distance of the tool during infeed, **AW** is always to be specified for safety reasons. This also applies to premachined spigot. **W** describes the width of the bottom area to be machined.

Machining allowance at the base **AL** can be used to define the height above the plane surface up to which the machining allowance **AW** is supposed to extend. Above this height and up to **UL**, the default wall-sided machining allowance of **CG%** of the tool diameter is applicable. Thus, Specify **AL** during roughing (**CV** 1 or 2) to declare a non-uniform machining allowance between **UL** and **DT**, such as it can result from a casting allowance on walls and bottom.

While approaching, the milling cutter center is positioned beyond the first element **FE** by the radial distance resulting from the tool radius, radial safety distance **RD** and the particular radial machining allowance and the tool is moved to the particular infeed depth in rapid traverse. This is followed by a vertical radial infeed and an equidistant tracing out of the pin. This, in turn, is followed by further radial infeeds and tracing outs until finishing allowance **RSL** is reached during roughing or until the final size of the pin is reached during finishing. While the final size is milled, the tool is traveled with edge radius correction. One approach and run-out arc are added each.

After a chain of radial cuts has been completed, the tool is lifted by the amount of safety distance **SDI** in infeed direction and the next cut of the segmentation can be made in infeed direction. After cutting has been completed, the tool moves to the return position **RL** or across the reference plane **UL** by **SDI** and then again reaches the initial coordinates in the interpolation plane which were approached before the cycle was called.

Finishing requires a wall-sided finishing allowance **RSW** (wall finishing) and/or a finishing allowance on the bottom **RSL** (bottom finishing). During bottom finishing, the bottom width is defined by **AW**. Finishing takes always place in infeed direction in a single cut. In radial direction, finishing can also comprise several cuts. A special feed **FF** can be activated for finishing.

If the actual contact width of the tool is less than the diameter of the milling cutter, the effective feed is increased as compared with the programmed feed to ensure the intended feed within certain limits at the tool engaging point.

G40 (no edge radius correction) is to be active prior to the cycle. The cycle deactivates global transition elements (**RND**, **RNE**, **CHF**, and **CHS**).

Example: A circular spigot is to be machined. The machining allowance is uniformly distributed across the depth. It is assumed that the upper part with the radial allowance can be machined in a single cut. The maximum depth of cut is 10 mm.

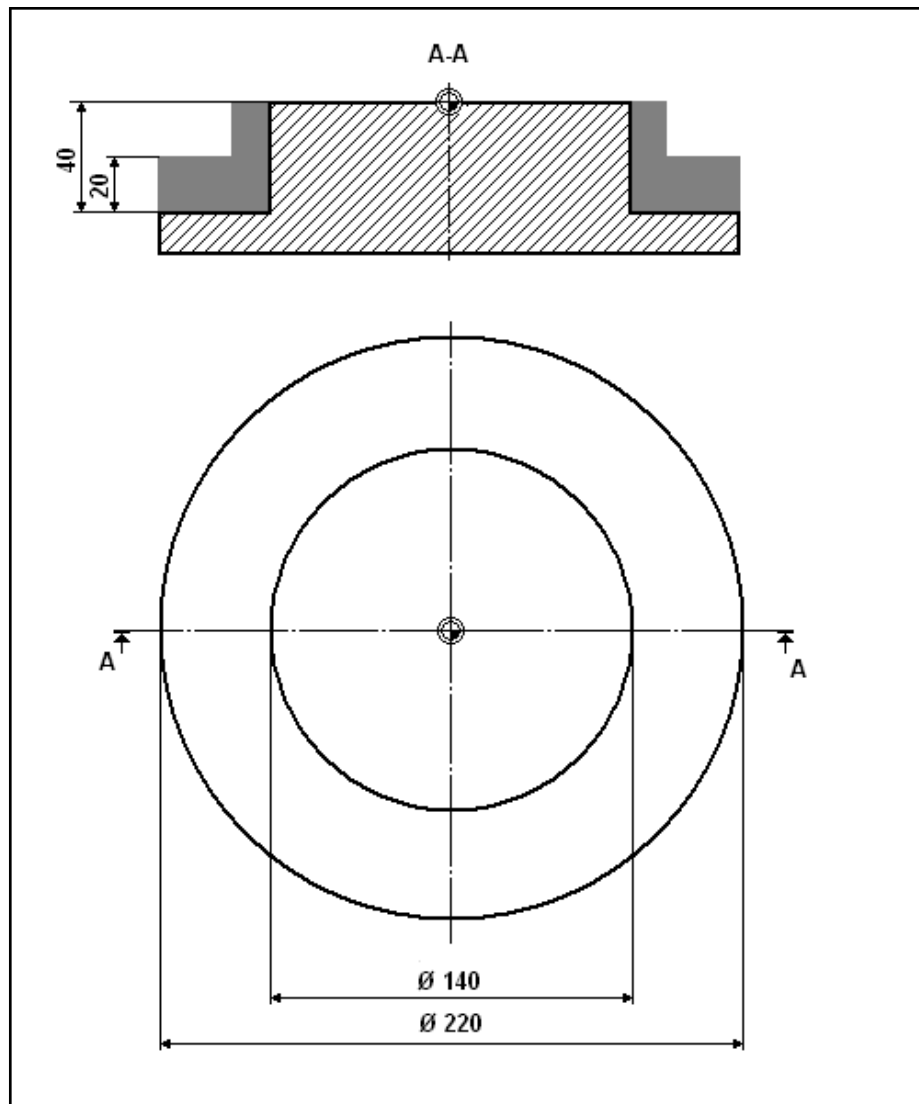


Fig. 7-37: Example of G277 Circular pin milling
Program for G277 Circular pin milling

```

N10 T1 M6 ; Tool change (milling cutter D 32)
N20 RAD G90 G17 ; Plane
N30 G95 G97 S1000 M3 F800 ; Technology values
N40 G0 Z100 ; Prepositioning
N50 G277 (HD140, UL0, SDI0, DT-40, CD10, AL20, AW40, RSL1, RSW.02, FF600, PC CD) ; Cycle definition
N60 G0 XX0 Y0 ; Cycle call
N70 G80 X200 Y200 Z100 M5 ; Leave contour, delete modal call

```

7.13 G279 Land straight milling

The cycle "G279 Land straight milling" can machine an elevated form element from outside. The element consists of two parallel surfaces spaced apart by the width Wl and has two semicircular roundings on both sides. A straight land is the counterpart of a straight groove or a straight elongated hole. The cycle allows milling the lateral surface and the plane surface at the land face (called bottom). It is distinguished between roughing (several cuts both in infeed direction and radial direction) and finishing (single wall milling, if necessary several radial cuts on the bottom). The depth infeed is executed outside in rapid traverse. Therefore, a material-free area with a minimum width resulting from the sum of milling cutter diameter, safety distance (pa-

parameter RD) and wall-sided machining allowance (parameter AW) around the land has to be ensured.

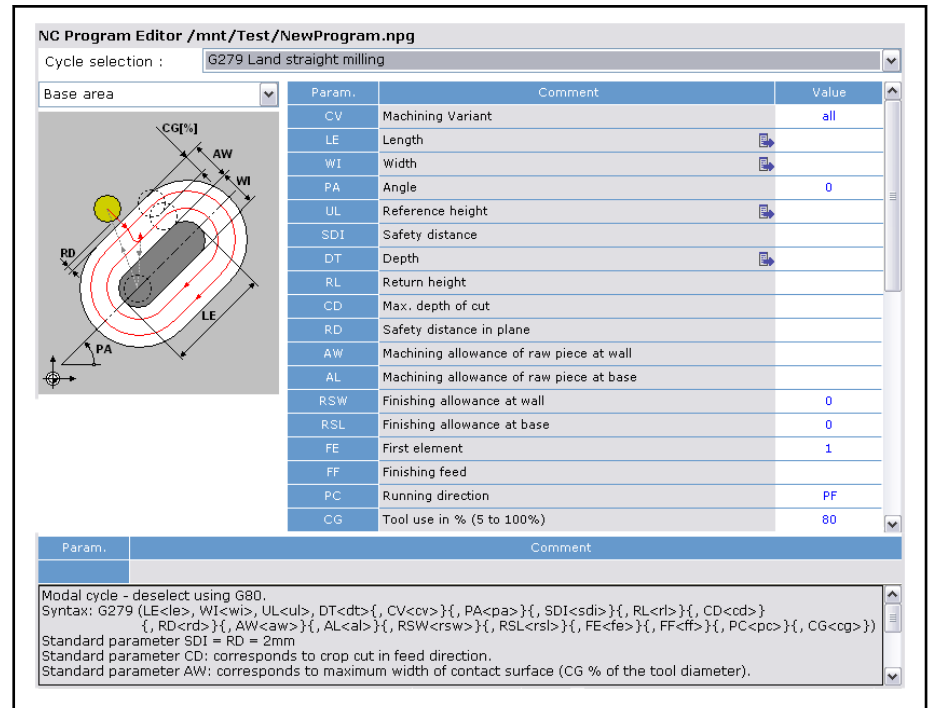


Fig. 7-38: G279 Land straight milling

G279		Land straight milling	G279(CV,LE,WI,PA,UL,SDI,DT,RL,CD,RD,AW,AL,RSW,RSL,FE,FF,PC,CG)		
CV	SD.SysCyc.CV	Machining (cutting) variant	Integer	Optional	1 Roughing 2 Complete 3 Finishing 2 is the default setting
LE	SD.SysCyc.LE	Length (total from rounding to rounding)	Real	Mandatory	>0
WI	SD.SysCy.WI	Width	Real	Mandatory	0<WI<LE
PA	SD.SysCyc.PA	Angle	Real	Optional	Absolute 0 is preset
UL	SD.SysCyc.UL	Reference height	Real	Mandatory	Absolute
SDI	SD.SysCyc.SDI	Safety distance	Real	Optional	>=0 Default: 2 mm
DT	SD.SysCyc.DT	Depth	Real	Mandatory	Absolute
RL	SD.SysCyc.RL	Retraction height	Real	Optional	Absolute
CD	SD.SysCyc.CD	Max. infeed depth	Real	Optional	>0 One roughing cut in infeed direction by default
RD	SD.SysCyc.RD	Safety distance in plane	Real	Optional	>=0 Default: 2 mm

Milling cycles

G279		Land straight milling	G279(CV,LE,WI,PA,UL,SDI,DT,RL,CD,RD,AW,AL,RSW,RS L,FE,FF,PC,CG)		
AW	SD.SysCyc.AW	Machining allowance of the un-machined part at the wall	Real	Optional	>0 CG% of tool diameter by default
AL	SD.SysCyc.AL	Machining allowance of the un-machined part at the base	Real	Optional	>0 Uniform machining allowance between UL and DT by default
RSW	SD.SysCyc.RSW	Finishing allowance at the wall	Real	Optional	>=0
RSL	SD.SysCyc.RSL	Finishing allowance at the base	Real	Optional	>=0
FE	SD.SysCyc.FE	First element. Element at which radial infeed takes place	Integer	Optional	1<=FE<=4 FE1 corresponds to the center of the rounding arranged in PA direction. Counting in counterclockwise direction 1 is preset
FF	SD.SysCyc.FF	Finishing feed	Real	Optional	>0 Latest programmed F-value by default
PC	SD.SysCyc.PC	Running direction	String	Optional	PF clockwise direction CD counterclockwise direction PF is preset
CG	SD.SysCyc.CG	Tool use	Real	Optional	5 to 100 % 80 % is preset

Sequence: The cycle supports all interpolation planes and diameter programming of the x-axis. All coordinates (in this case only coordinates in the infeed axis) are absolute coordinates and are to be specified with the respective axis scaling (DIA/RAD). All remaining parameters are to be specified with radius dimension.

The cycle runs modally and has to be deselected with G80.

The position of the center of a rounding of the land to be machined is determined by the coordinates in the interpolation plane approached to execute the cycle. The connecting line to the center of the second rounding occupies the absolute angle **PA**. The diameter of either curve corresponds to the width **WI**. The dimension from rounding to rounding is defined by the length **LE**.

In addition to the radial dimensions, the machining task is described by two coordinates in infeed direction, which define the position of the upper side (**UL**) and the base (**DT**). During roughing, the cuts are segmented between these two coordinates with a uniform depth of cut (**CV** 1 or 2) which can be limited with **CD**. If CD is not used, only one roughing cut is executed in infeed direction.

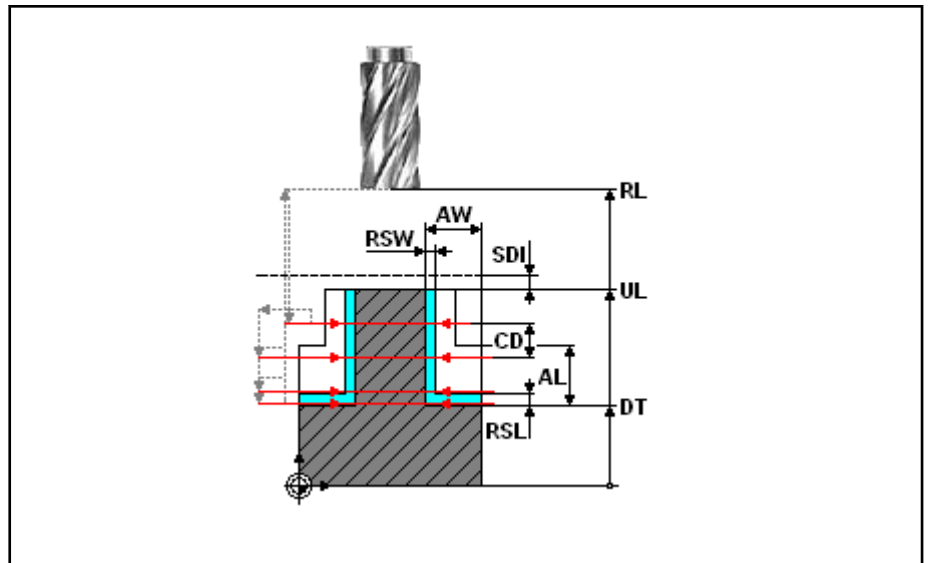


Fig. 7-39: Depth parameters, machining allowances and allowances for G279

Each cut in infeed direction can be segmented in several cuts in parallel to the contour (radial cut segmentation from without inward). The equidistance between these cuts is limited by the tool diameter multiplied with the maximum tool use **CG**. The direction of tracing out is defined with regard to the direction of spindle rotation and can be influenced by **PF**. Radial infeed is always executed to the center of the contour element to be defined by the parameter **FE**.

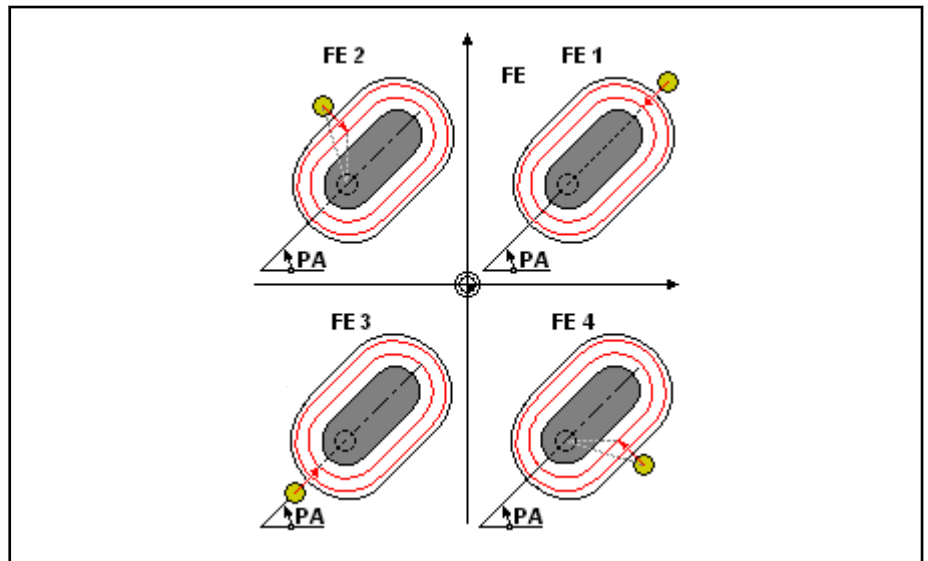


Fig. 7-40: Parameter "First element FE" for G279

The machining allowance at the wall **AW** describes the initial state of the raw piece. If this parameter is not used, a machining allowance is assumed which can be machined with a single radial cut, i.e., allowance equal to the maximum tool use **CG** multiplied with the tool diameter. Since it defines the radial distance of the tool during infeed, **AW** is always to be specified for safety reasons. This also applies to premachined lands. **W** describes the width of the bottom area to be machined.

Machining allowance at the base **AL** can be used to define the height above the plane surface up to which the machining allowance **AW** is supposed to extend. Above this height and up to **UL**, the default wall-sided machining al-

lowance of CG% of the tool diameter is applicable. Thus, Specify AL during roughing (CV 1 or 2) to declare a non-uniform machining allowance between UL and DT, such as it can result from a casting allowance on walls and bottom.

While approaching, the milling cutter center is positioned beyond the first element **FE** by the radial distance resulting from the tool radius, radial safety distance **RD** and the particular radial machining allowance and the tool is moved to the particular infeed depth in rapid traverse. This is followed by a vertical radial infeed and an equidistant tracing out of the spigot. Subsequently, more radial infeeds and tracing outs take place until the finishing allowance **RSL** is reached during roughing or until the final size of the spigot is reached during finishing. While the final size is milled, the tool is traveled with edge radius correction. One approach and run-out arc are added each.

After a chain of radial cuts has been completed, the tool is lifted by the amount of safety distance **SDI** in infeed direction and the next cut of the segmentation can be made in infeed direction. After cutting has been completed, the tool moves to the return position **RL** or across the reference plane UL by SDI and then again reaches the initial coordinates in the interpolation plane which were approached before the cycle was called.

Finishing requires a wall-sided finishing allowance **RSW** (wall finishing) and/or a finishing allowance on the bottom **RSL** (bottom finishing). During bottom finishing, the bottom width is defined by AW. Finishing takes always place in infeed direction in a single cut. In radial direction, finishing can also comprise several cuts. A special feed **FF** can be activated for finishing.

If the actual contact width of the tool is less than the diameter of the milling cutter, the effective feed on curves may be higher than the programmed feed to ensure the intended feed within certain limits at the tool engaging point.

G40 (no edge radius correction) is to be active prior to the cycle. The cycle deactivates global transition elements (RND, RNE, CHF, and CHS).

Example: A land is to be machined on a cast workpiece, first by roughing with a maximum depth of cut of 7 mm and then by finishing in a call with a end mill with a diameter of 16. The maximum machining allowance on the land is 4 mm. Perform the radial infeed at the rounding whose center is approached prior to the cycle. To completely machine the plane surface even at the roundings R12 and to optimize the paths, a wall-sided machining allowance of 9 mm is defined while the radial safety distance is reduced to 1 mm at the same time. The maximum machining allowance on the plane surface does not exceed 6 mm. Tool use is 50% which increases to almost 100% at the R12 roundings.

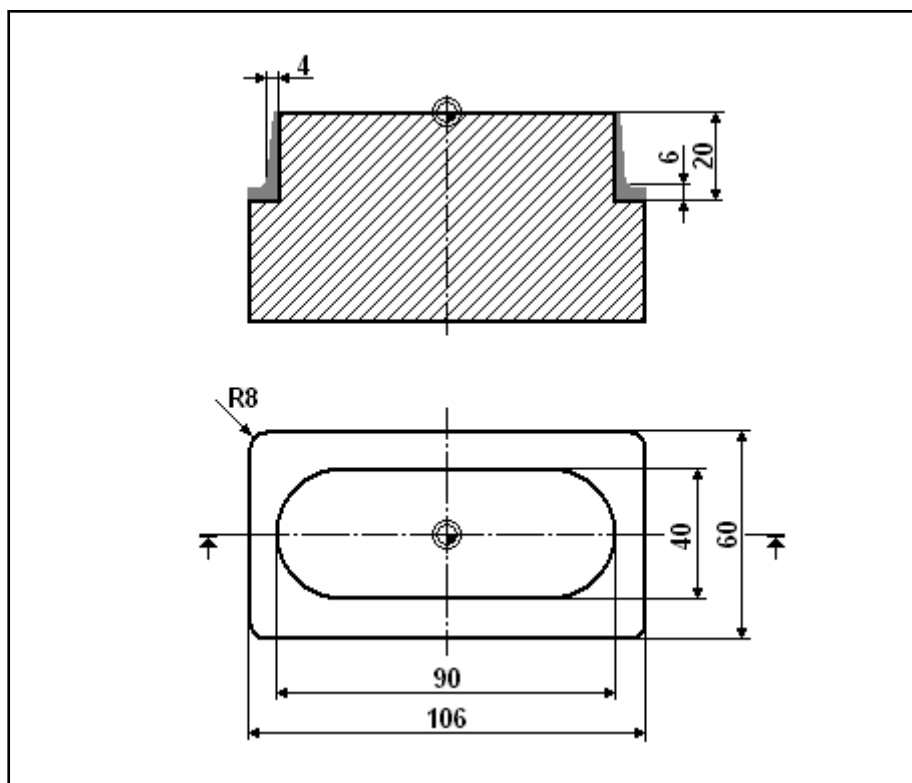


Fig. 7-41: Example of G279 Land straight milling
Program for G279 Land straight milling

```

N10 T1 M6; End milling cutter D 16
N20 RAD G17 G90 G95 G97 S3500 M3
N30 G279 (LE90, WI40, UL0, DT-20, CD7, RD1, AW9, AL6, RSW0.5, RSL0.5, FE3, FF1250, CG50)
N40 G0 X-25 Y0 F1700 ; Approach reference position and execute cycle
N50 G80 X200 Y200 Z200 M5 ; Leave contour, delete modal call

```

7.14 G280 Land curved milling

Syntax G280(CV,ORI,WI,R,PA,OA,UL,SDI,DT,RL,CD,RD,AW,AL,RSW,RSL,FE,FF,PC,CG)

Brief description The cycle "G280 Land curved milling" can machine an elevated form element from outside. The element is the counterpart of a curved groove.

Overview

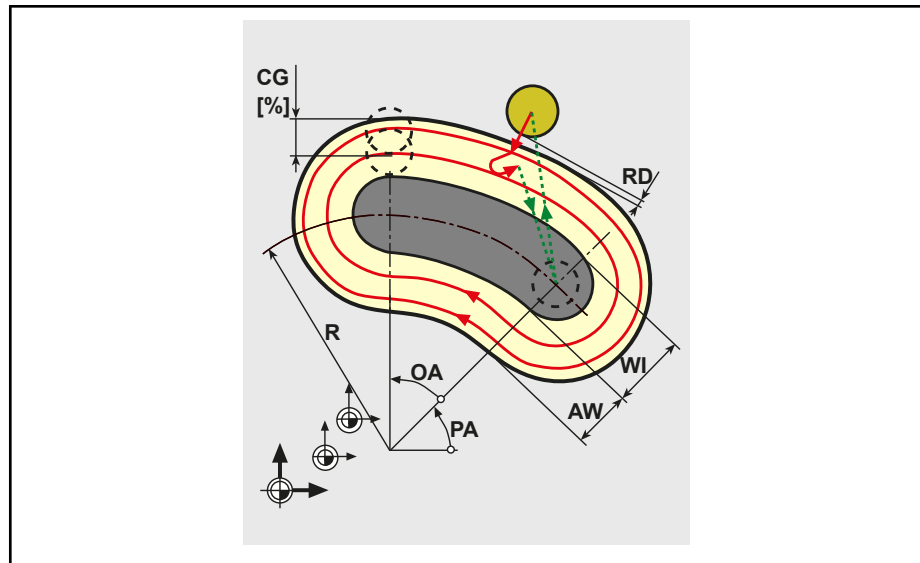


Fig. 7-42: Cycle G280 Land curved milling

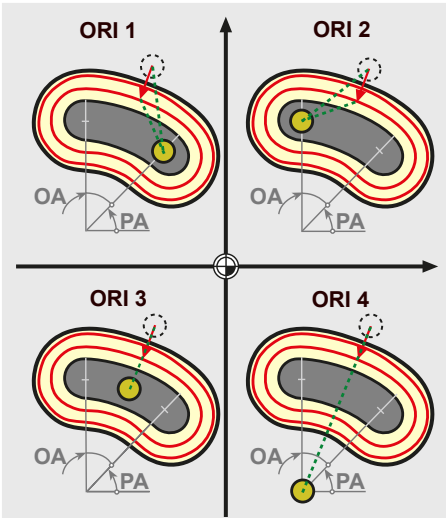
Parameter

Parameter	Designation	Properties	Values	Pre-assignment
CV	Machining (cutting) variant	Optional	Roughing (1) / Complete (2) / Finishing (3)	2
ORI	Reference point	Optional		1
WI	Width	Mandatory	>0	
R	Radius	Mandatory	>0.5*WI	
PA	Starting angle	Optional	Degree	0
OA	Opening angle	Mandatory	Incremental Degree <>0	
UL	Reference height	Mandatory	Absolute	
SDI	Safety distance	Optional	Incremental ≥0	2 mm
DT	Depth	Mandatory	Absolute	
RL	Retraction height	Optional	Absolute	UL +/- SDI
CD	Maximum in-feed depth	Optional	Incremental >0	One-time in-feed until finishing allowance RSL
RD	Safety distance in plane	Optional	Incremental ≥ 0	2 mm

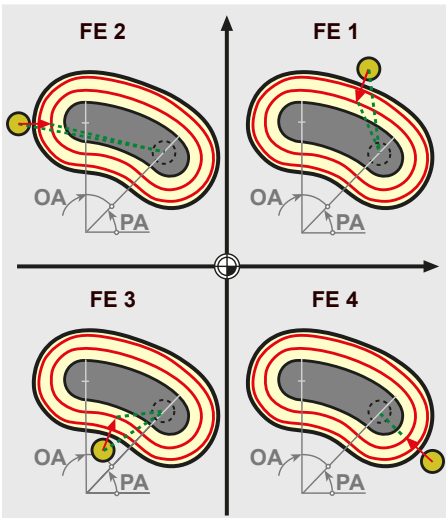
AW	Machining allowance of the unmachined part at the wall	Optional	$0 < AW \leq R - 0.5 \cdot AW$	CG % of the current tool diameter
AL	Machining allowance of the unmachined part at the base	Optional	> 0	Difference between UL and DT
RSW	Finishing allowance at the wall	Optional	≥ 0	0
RSL	Finishing allowance at the base	Optional	≥ 0	0
FE	first element	Optional		1
FF	Finishing feed	Optional	> 0	Current feed
PC	Running direction	Optional	Climb milling (PF)/up-cut milling (CD)	Climb milling (PF)
CG	Tool use	Optional	$0 < CG \leq 100$, Percent of the tool diameter	80

Tab. 7-17: Parameter call

ORI	Effect
1	Reference point is the rounding center point, referring to PA
2	Reference point is the rounding center point, referring to OA
3	Reference point to bisecting line of OA

ORI	Effect
4	Reference point is the center of curvature (radius R)
	

Tab. 7-18: Description of the parameter ORI - Reference point

FE	Effect
1	Start at external rounding with opening angle OA
2	Start at rounding, counterclockwise at the external rounding
3	Start at internal rounding
4	Start at rounding, clockwise at the internal rounding
	

Tab. 7-19: Description of the parameter FE - First element

Description CV Machining (cutting) variant
Selecting pre-machining and fine machining
ORI Reference point

Selecting a point at the contour element to which the machining geometry refers to and which has to be approached for calling the modal cycle

WI Width

Width of the land

R Radius

Center of curvature of the center line of the land

PA Starting angle

Absolute angle from the center point of the curvature to the center point of a land rounding

OA Opening angle

Relative angle between both center points of roundings, limiting the land

UL Reference height

The topside is located at this absolute position in infeed direction

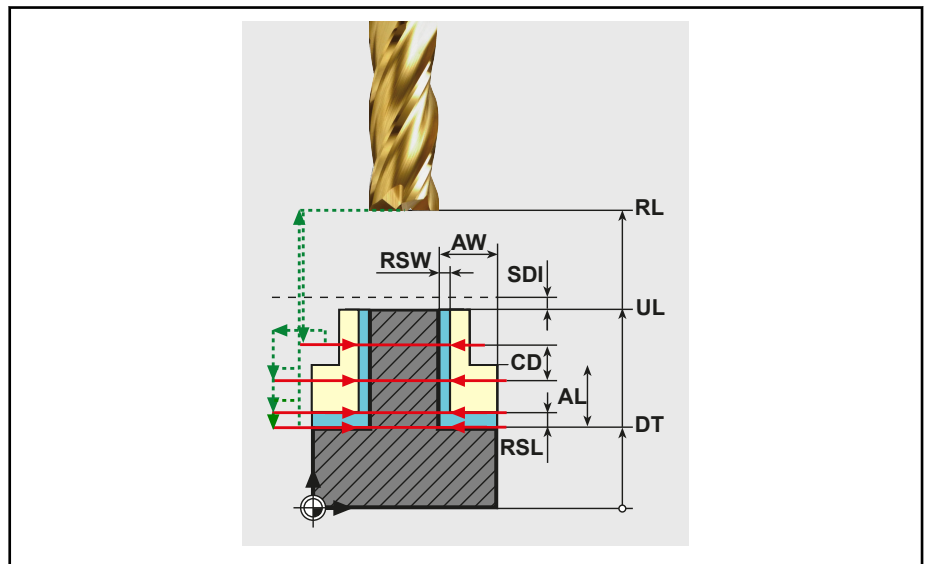


Fig. 7-43: Cycle parameter in infeed direction, dimensions and machining allowances

SDI Safety distance

Distance between tool and workpiece in direction of the infeed. If the distance is fallen below, it is always traveled in feed

DT Depth

The plane surface where the land starts is located at this absolute position in infeed direction

RL Retract height

This absolute position is the infeed axis coordinate at the end of the cycle

CD Maximum infeed depth

Maximum distance between two roughing cuts in infeed direction

RD Safety distance in plane

Minimum distance between tool and workpiece that has to be complied with during positioning in the interpolation level

Machining allowance of the unmachined part at the wall

Radial machining allowance on the lateral surface of the land

	AW applies to roughing. AW specifies the width of the plane level (base) to be machined during finishing AL Machining allowance of the unmachined part at the base Machining allowance on the plane surface where the land starts AL applies to roughing. If AL is specified, there is a non-uniform machining allowance AW along the height. The standard machining allowance that can be cut with a radial cut applies above AL RSW Finishing allowance at the wall Radial allowance during roughing, remaining as machining allowance for finishing Finishing allowance at the base Allowance on the adjacent plane surfaces during roughing, remaining as machining allowance for finishing FE First element Index of the contour element at whose center each radial cut starts and ends FF Finishing feed Special feed for finishing PC Running direction Switches between parallel feed milling and counter direction milling and influences the contour direction depending on the direction of rotation of the spindle CG Tool use Maximum contact width of the tool in percent of the diameter
Special features	Note that a modal cycle is not executed with the cycle call during programming, but only after the next traversing command. G40 (no edge radius correction) is to be active prior to the cycle. The cycle deactivates global transition elements (RND, RNE, CHF, and CHS). The depth infeed is executed outside in rapid traverse. Therefore, a material-free area with a minimum width resulting from the sum of milling cutter diameter, safety distance (parameter RD) and wall-sided machining allowance (parameter AW) around the land has to be ensured. In case of only finishing (CV3), at least one of the finishing allowances RSW and/or RSL has to be greater than zero, as these parameters define the initial machining dimensions. The feed FF and the current feed are taken into consideration as effective feeds. The cycle automatically adjusts the feed according to the current path curvature and the effective contact width.
Procedure/processing	The land depth between UL and DT minus RSL is distributed into uniform cuts with a maximum depth of CD during roughing (CV1 and CV2). If CD is not specified, only one roughing cut is executed in infeed direction. Depending on the machining allowance, each cut in infeed direction can be segmented in several cuts in parallel to the contour (radial cut segmentation - external part to internal part). The equidistance between these cuts is limited by the tool use. While approaching, the milling cutter center is positioned beyond the center of the first element FE by the radial distance resulting from the tool radius, radial safety distance RD and the radial machining allowance and the tool is moved to the infeed depth in rapid traverse. If the safety distance SDI to the workpiece is fallen below, it is traveled in feed. This is followed by a radial in-

feed and an equidistant tracing out of the land. This, in turn, is followed by further radial infeeds and tracing outs until finishing allowance RSL is reached during roughing or until the final size of the land is reached during finishing. One approach and run-out arc are added when milling the finished dimension.

After a chain of radial cuts has been completed, the tool is lifted by the amount of safety distance SDI in infeed direction and the next cut of the segmentation can be made in infeed direction.

After cutting has been completed, the tool moves to the return position RL and reaches the initial coordinates in the interpolation plane which were approached before the cycle was called.

Example

The blades of an impeller are to be machined by finishing. The finishing allowance existing on the blades is 0.8 mm (wall finishing). To minimize the infeed paths, infeed should be radial at each internal curve (with an R23 dimension).

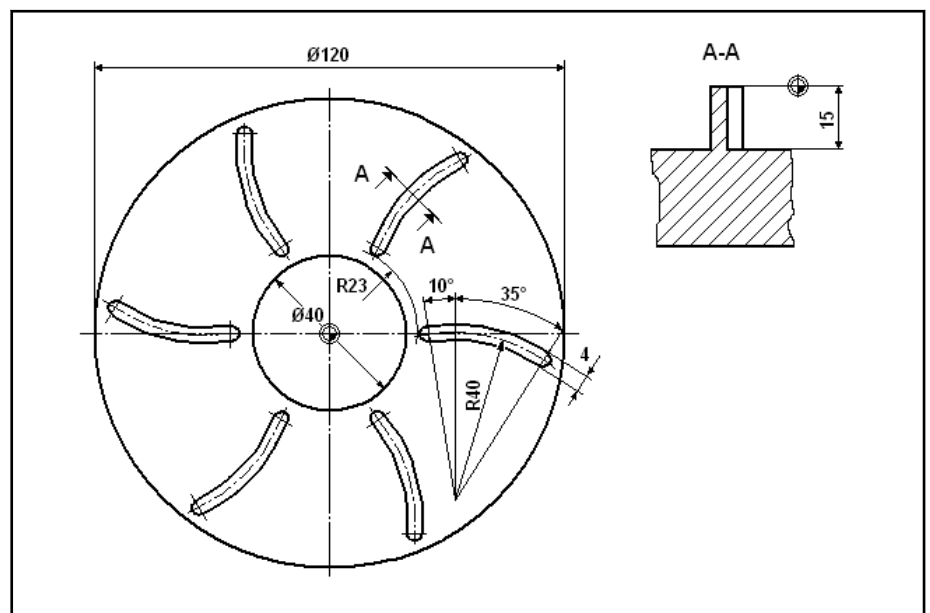


Fig. 7-44: Example of G280 Land curved milling
Program for G280 Land curved milling

```

N10 T1 M6 ; End mill D 10
N20 RAD G17 G90 G95 G97 S4500 M3 F900 ; Technology values
N30 G280 (CV3, WI4, R40, PA100, OA-45, UL0, DT-15, RSW0.8, FE2) ; Cycle Milling curved land
N40 G113 (MCP0, SCP0, PR25, NH6, ROT2) ; Execute cycle on partial circle
N50 G0 G80 X200 Y200 Z200 M5 ; Delete modal call and leave contour

```


8 Milling cycles (contour-dependent)

8.1 General notes

⚠ CAUTION

If technology parameters are changed outside the program the cycle is programmed in, save the program again.

The contour-dependent cycles create a cut segmentation and use it again if the cycle parameters were not changed. When saving the program, all old cut segmentations are discarded and newly once created if required.

8.2 Defining the contour

The contour of milling cycles is defined as the contour-dependent turning cycles (see [chapter 11 Turning cycles \(contour-dependent\)](#), page 251).

8.3 G273 User-defined contour pocket milling

The cycle "User-defined contour pocket milling" can mill a pocket with a user-defined contour by cut segmentation.

NOTICE

Due to the extraordinarily high number of possible combinations between geometry and machining variance, the calculated cut segmentation can sometimes not meet the user requirements or even violate the contour.

Due to this reason, the NC programmer has to carefully check and implement the results of the cycle. A recommended tool is the NC simulation.

Milling cycles (contour-dependent)

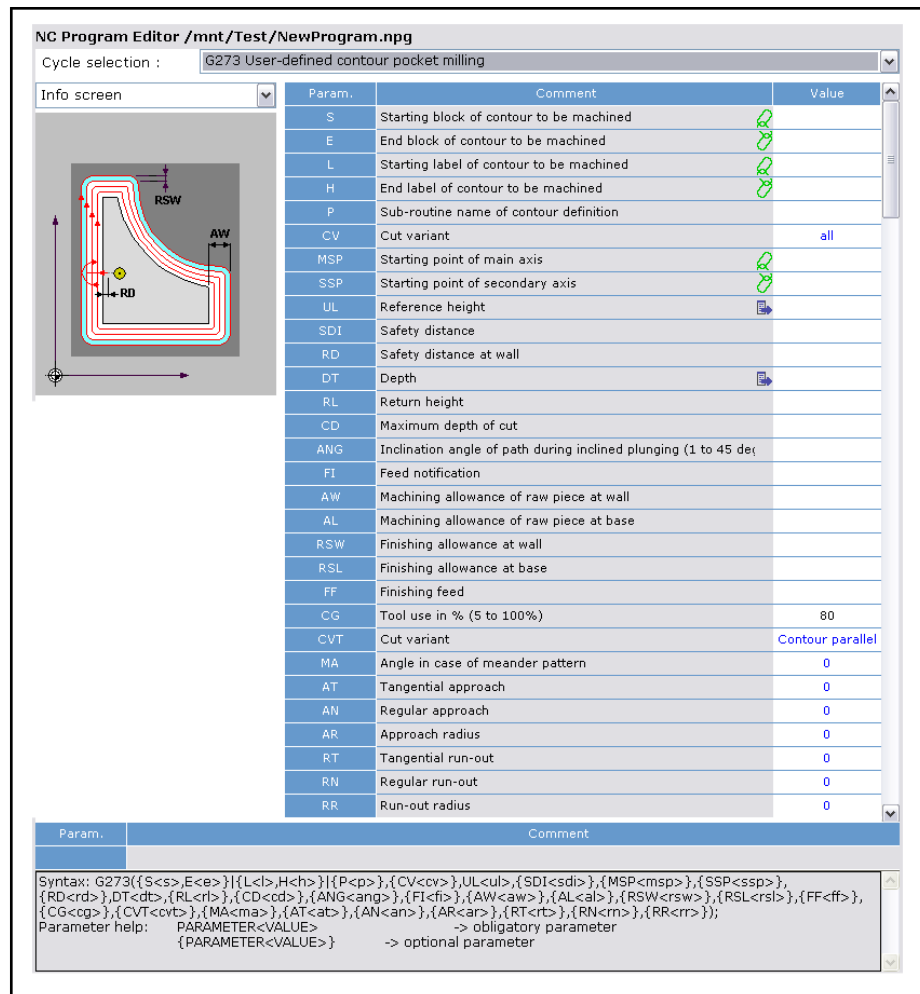


Fig. 8-1: G273 User-defined contour pocket milling

G273	User-defined contour pocket milling	G273(S,E,L,H,P,CV,CVT,CD,FF,AW,AL,RSW,RSL,UL,DT,RL,CG,SDI,RD,AT,AN,AR,RT,RN,RR,ANG,FI)		
S	Starting block of the contour to be machined	String	Optional	
E	End block of the contour to be machined	String	Optional	
L	Starting label of the contour to be machined	String	Optional	
H	End label of the contour to be machined	String	Optional	
P	Subroutine name of the contour definition	String	Optional	
CV	Machining (cutting) variant	Integer	Optional	1 Roughing 2 Complete 3 Finishing 2 is the default setting

G273	User-defined contour pocket milling	G273(S,E,L,H,P,CV,CVT,CD,FF,AW,AL,RSW,RSL,UL,DT,RL,CG,SDI,RD,AT,AN,AR,RT,RN,RR,ANG,FI)		
CVT	Milling algorithm	Integer	Optional	1 Equidistant 2 Meander 1 is preset
CD	Maximum infeed depth	Real	Optional	Incremental >0 One roughing cut in infeed direction by default
FF	Finishing feed	Real	Optional	>0 Last programmed F-value is set by default
AW	Machining allowance of the unmachined part at the wall	Real	Optional	>0
AL	Machining allowance of the unmachined part at the base	Real	Optional	>0
RSW	Finishing allowance at the wall	Real	Optional	>0 No allowance by default
RSL	Finishing allowance at the base	Real	Optional	>0 No allowance by default
UL	Reference height	Real	Mandatory	Absolute
DT	Depth	Real	Mandatory	Absolute
RL	Retraction height	Real	Optional	Absolute UL+-SDI by default
CG	Tool use	Real	Optional	5 to 100 % 80 % is preset
SDI	Safety distance vertical to the machining plane	Real	Optional	>=0 2 mm by default
RD	Safety distance in plane	Real	Optional	>=0 2 mm by default
AT	Tangential approach length	Real	Optional	>=0 Default setting is 0
AN	Regular approach length	Real	Optional	>=0 Default if neither AT/AN/AR -> AN=RD
AR	Approach radius	Real	Optional	>= Tool radius or 0 Default setting is 0
RT	Tangential run-out length	Real	Optional	>=0 Default setting is 0
RN	Regular run-out length	Real	Optional	>=0 Default if neither RT/RN/RR -> RN=RD
RR	Run-out radius	Real	Optional	>= Tool radius or 0 Default setting is 0

G273	User-defined contour pocket milling	G273(S,E,L,H,P,CV,CVT,CD,FF,AW,AL,RSW,RSL,UL,DT,RL,CG,SDI,RD,AT,AN,AR,RT,RN,RR,ANG,FI)		
ANG	Angle during plunging	Real	Optional	$0 < ANG \leq 90$ Default 90
FI	Infeed feed when plunging	Real	Optional	> 0 Default setting: last programmed F-value
MA	Meander angle with main axis MA	Real	Optional	$-360 \leq MA \leq 360$ Default setting is 0
MSP	Starting or plunging position in main axis	Real	Optional	Absolute
SSP	Starting or plunging position in secondary axis	Real	Optional	Absolute

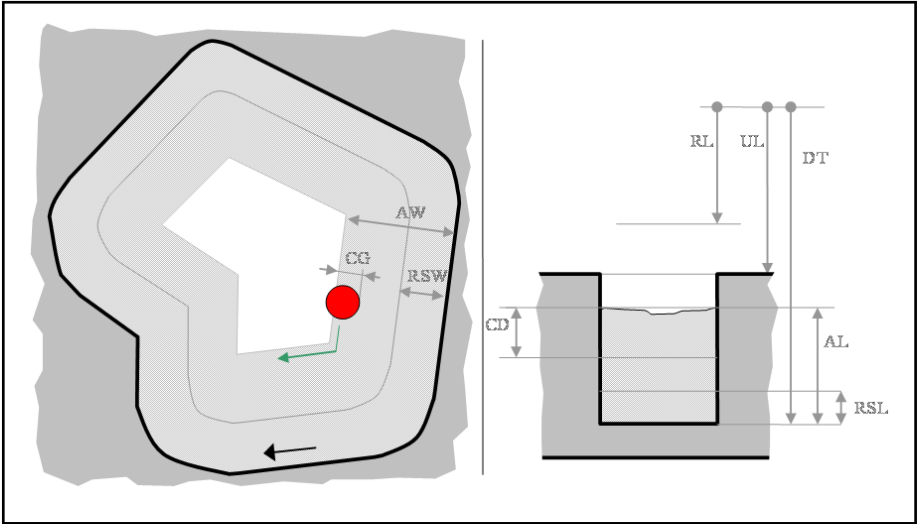


Fig. 8-2: Parameters of G273

The definition of the contour is described in [chapter 8.2 "Defining the contour"](#) on page 147.

- 

If the contour is not closed [first point (line) and last point (line) are identical], a runtime error is generated.
- 

A single circle is no valid contour. A valid contour has thus to comprise of at least two elements.

Unprogrammed parameters:

If the machining allowance of the raw piece at base **AL** is not programmed, the cycle assumes that the pocket is completely filled. In this case, the pocket is roughed from **UL** to **DT**.

If the machining allowance of the raw piece at the wall **AW** is not programmed, a "full" pocket is assumed.

If the maximum infeed depth **CD** is not programmed, it is perpendicularly infeed only once to the machining plane. As a result, the parameter **AL** is irrelevant and is thus ignored. Roughing in $Z=DT+RSL$; finishing in $Z=DT$.

If the retract height **RL** is not programmed, the tool is only retracted to the upper height **UL** plus the safety distance **SDI**. The tool is generally not in the pocket anymore.

If no starting point **MSP/SSP** is specified, the cycle automatically selects a suitable starting point. Usually, this is the position at the tool with the greatest distance from the contour on all sides.



If a starting point is to be specified, both parameters **MSP/SSP** have to be programmed. It is not permitted to only specify one parameter.



If only roughing is selected via the cutting variant **CV**, a warning on the finishing parameters (e.g., finishing infeed, approach, run-out) is not output. These parameters are ignored. In the opposite case (cutting variant = finishing), a warning on roughing parameters (e.g., machining allowance at wall, tool usage) is not output either.

The milling algorithm parameter CVT can be used to define how the cycle machines a plane (depth infeed).

Possible algorithms are:

- Machining along the equidistant (CVT = 1)
- Machining by means of meander patterns (CVT = 2)

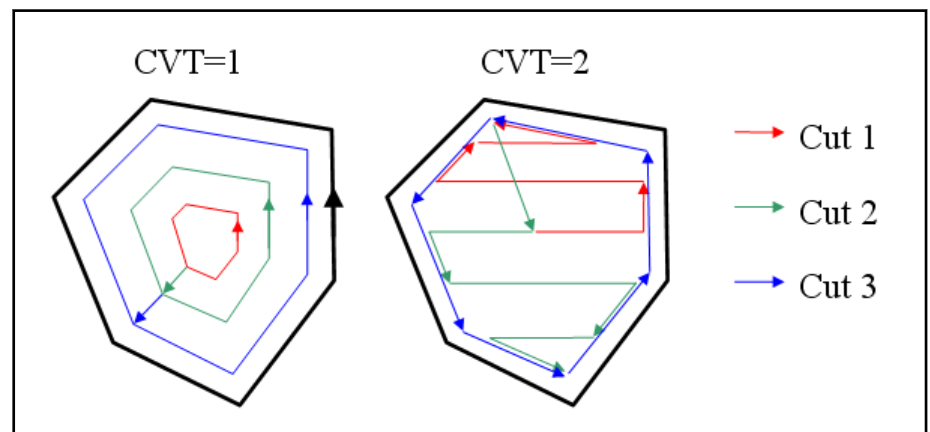


Fig. 8-3: Equidistant (CVT = 1) and meander (CVT = 2) milling algorithms

If machining along the **equidistant** is selected, the tool is always infeed as usual to the next equidistant towards the contour.

If the plane is machined on **meander patterns**, the tool starts from the center (or from the starting position),

1. machines one half of the pocket,
2. then the other half of the pocket,
3. and finally mills the resulting step profile along the contour.

The meander angle **MA** can rotate the meander patterns in the machining plane. If 0° (default) is selected, the meander patterns extend in parallel to the main axis.

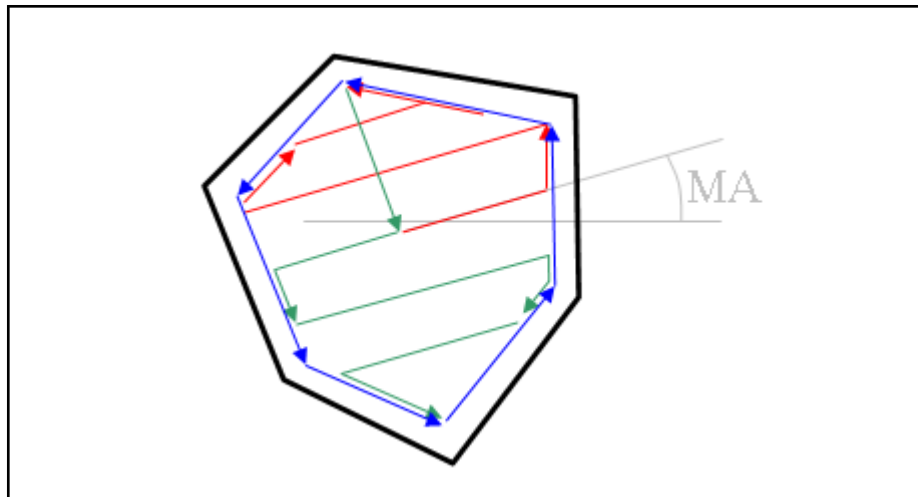


Fig. 8-4: Rotating meander patterns around the angle MA

Sequence: The cycle machines the pocket in the selected plane with main and secondary axis. After a depth has been completely milled, a depth infeed follows perpendicularly to the machining plane. The number of depth infeeds is calculated based on the maximum depth of the cut **CD**. The cycle machines all cuts up with the same depth.

Roughing outside the raw piece

In the first step, the tool is infeed (rapid traverse) in the current machining plane next to the contour (distance **RSW+RD** or at the starting point). A possible active tool radius correction is switched off. The tool is infeed perpendicularly to the plane in rapid traverse according to **CD**. The cycle mills to the contour as usual using the active feed until **RSW** is reached.

Finally, the contour is machined with equidistant allowance **RSW** and the active feed is traveled.



The machining direction in the pocket is equal to the direction in which the contour was defined.

After the contour has been milled, the tool is retracted from the contour by **SDI** and **RD**.

Roughing in the raw piece

First, there is an infeed (rapid traverse) in the current machining plane. A possible active tool radius correction is switched off.

The tool is infeed perpendicularly to the plane according to **CD**. The infeed method is based on the infeed angle **ANG** and the raw piece allowance at the wall **AW**. If it is infeed with feed and the infeed feed **FI** parameter was programmed, the cycle enables this feed for depth infeed.

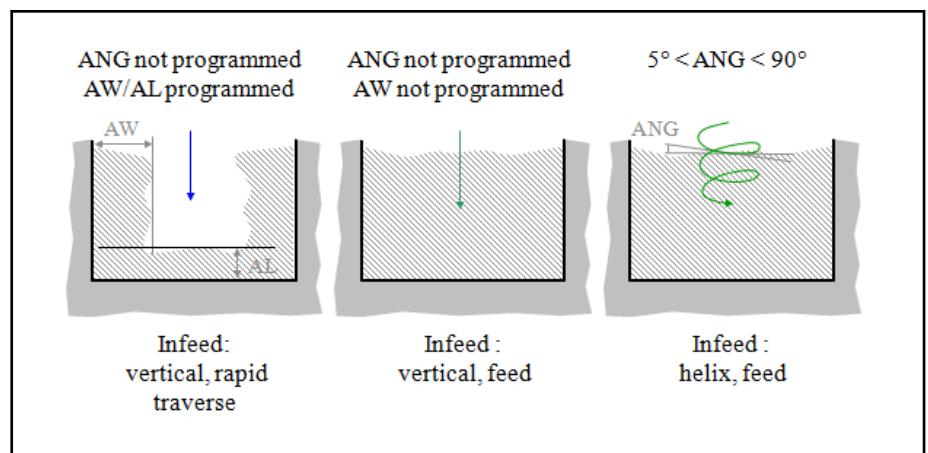


Fig. 8-5: Infeed variants for G273

In the following steps and depending on the milling algorithm **CVT**, the tool is either repeatedly infed as usual to the contour according to the programmed **CG** using the active feed and then mills on an equidistant around the contour or the material is milled out in a meander motion.



The number of infeeds in the machining plane for equidistant machining is calculated based on the maximum infeed:

CG × tool diameter

The cycle performs all infeeds becoming the same depth.

The same applies to the distance of the meander patterns.



To ensure that collisions between contour and tool are detected, the cycle uses the collision monitoring NC function "Collision, CLN". After the cycle, collision monitoring is set to two monitoring blocks with "CLN(DEF)" and remains active.

This sequence is repeated until the finishing allowance **RSW** is reached in the current depth infeed.

After the contour has been milled, the tool is retracted from the contour by **SDI** and **RD**.

Finishing at base

As in [Roughing in raw piece, page 152](#), the tool is infed in the contour and the pocket is machined on equidistant or meander patterns. The approach and run-out motion according to the parameters **AN/AR/AT** and **RN/RR/RT** are only made directly at the contour in the last cut.

Milling cycles (contour-dependent)

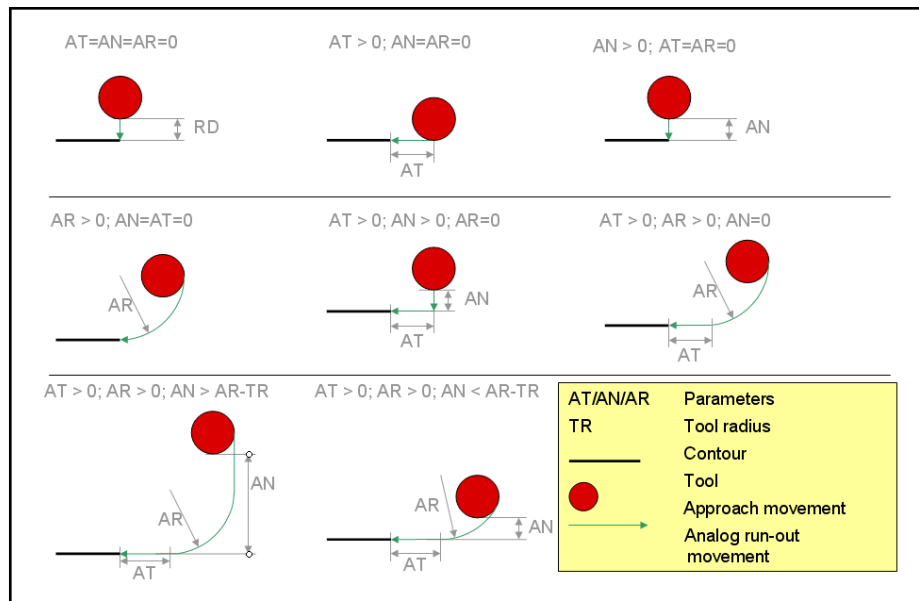


Fig. 8-6: Approach and run-out motion variants and parameters

The tool radius correction is only activated in this last step. The tool radius is compensated before by cut segmentation.



The feed rate of the center point path (feed correction NC function, cutter center point "G46") is specified for cutting along the contour (with active G41/G42). After the cycle, the initial state of the modal function is restored.

As in the following figure, "full" sections of concave contours are sometimes milled or parts of the raw piece are not machined completely. In this case, we recommend to machine the "pockets" of the convex contour before with a separate call.

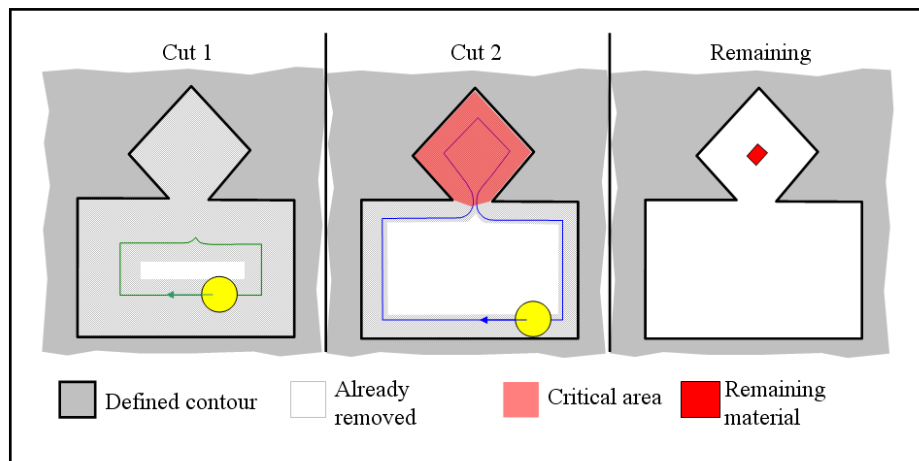


Fig. 8-7: Example of a critical area in the cycle G273 Contour pocket milling



The cycle does not calculate or machine off-cuts. This can cause (especially in case of equidistant patterns) incompletely machined pockets. In the figure (fig. 8-8 "Presentation of the corner problem" on page 155), material parts remain in acute angles. If the tool use CG is smaller than $(\sin(\text{Angle}/2) + 1)/2 \cdot 100\%$, no material remains in the angle. In the example applies: $\text{CG} < (\sin(60^\circ / 2) + 1)/2 \cdot 100\% = 75\%$.

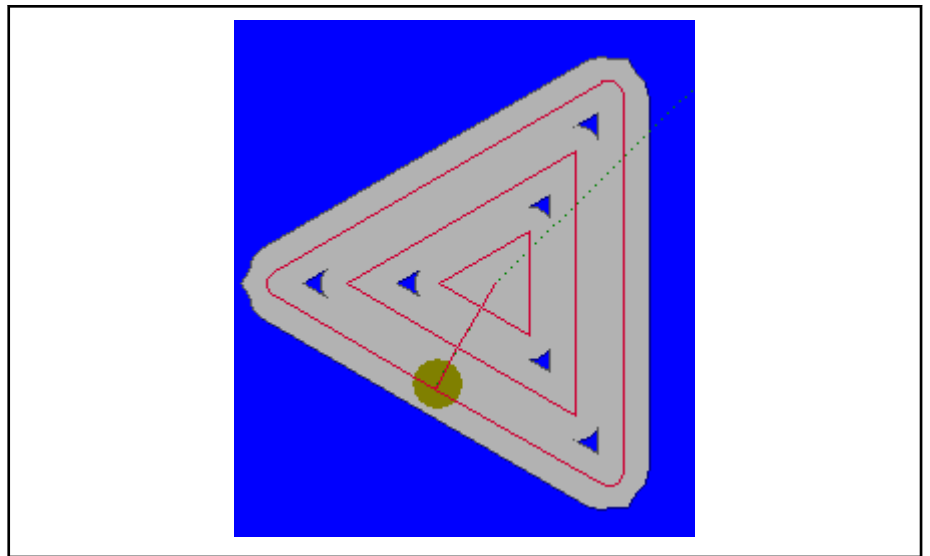


Fig. 8-8: Presentation of the corner problem

Special features/restrictions:

- The radius during approach and run-out (AR, RR) has to be higher than the tool radius.
- The contour may not exceed 50 elements including the input help (RND, CHL, CHS, straight moves).
- The number of valid motions is limited to 500.

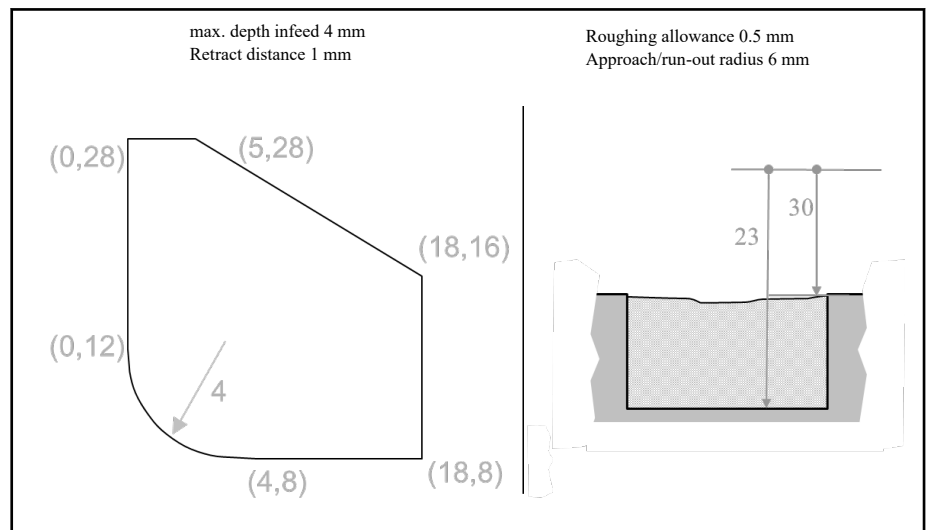
Example:

Fig. 8-9: Example of the cycle G273

Program:

```

N10 G17 G90 G94 F650 D1 ;Initialization
N20 G0 X5 Y14 Z35 ;Approach starting point
N30 G273 (S100,E160,UL30,DT23,CD4,RSL0.5,RSW0.5,SDI1,RD1,AR6,RR6) ;Call cycle
N40 M30 ;Program end

;*****
;*** Contour description ***
;*****
N100 G0 X5 Y28
N110 X0
N120 Y12
N130 G3 X4 Y8 R4
N140 G0 X18
N150 Y16
N160 X5 Y28

```

8.4 G278 User-defined contour pivot milling

The cycle "User-defined contour pivot milling" can mill a pivot with a user-defined contour by cut segmentation.

NOTICE

Due to the extraordinarily high number of possible combinations between geometry and machining variance, the calculated cut segmentation can sometimes not meet the user requirements or even violate the contour.

Due to this reason, the NC programmer has to carefully check and implement the results of the cycle. A recommended tool is the NC simulation.

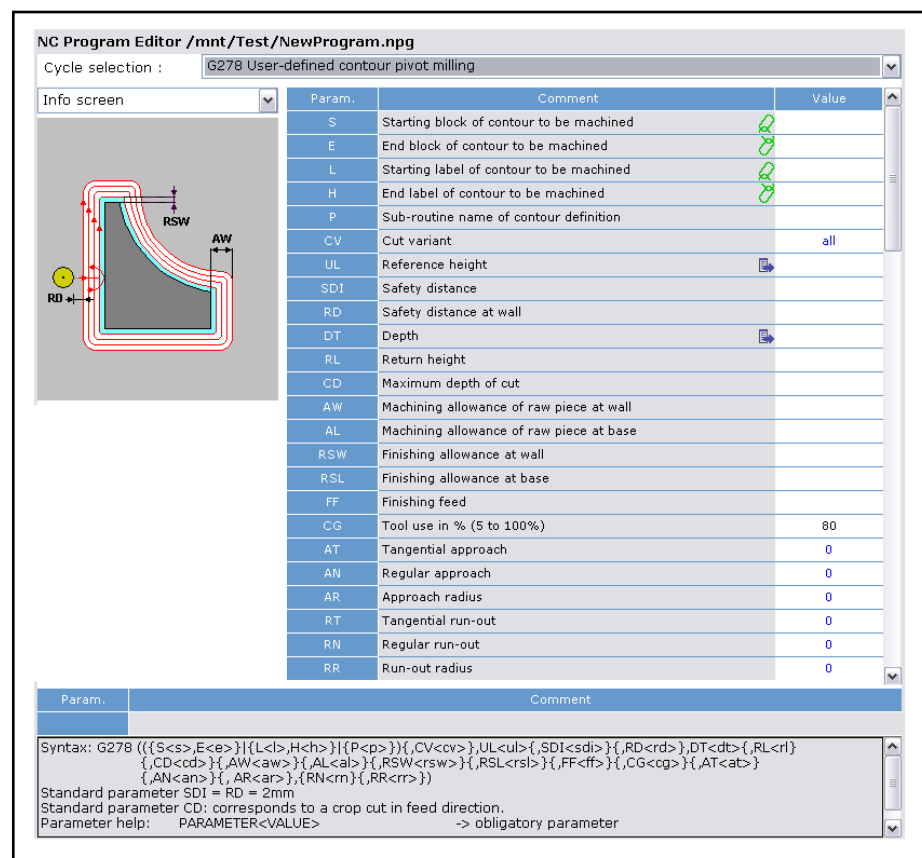


Fig. 8-10: G278 User-defined contour pivot milling

G278	User-defined contour pivot milling	G278(S,E,L,H,P,CV,CD,FF,AW,AL,RSW,RSL,UL,DT,RL,CG,SDI,RD,AT,AN,AR,RT,RN,RR)		
S	Starting block of the contour to be machined	String	Optional	
E	End block of the contour to be machined	String	Optional	
L	Starting label of the contour to be machined	String	Optional	
H	End label of the contour to be machined	String	Optional	

G278	User-defined contour pivot milling	G278(S,E,L,H,P,CV,CD,FF,AW,AL,RSW,RSL,UL,DT,RL,CG,SDI,RD,AT,AN,AR,RT,RN,RR)		
P	Subroutine name of the contour definition	String	Optional	
CV	Machining (cutting) variant	Integer	Optional	1 Roughing 2 Complete 3 Finishing 2 is the default setting
CD	Maximum infeed depth	Real	Optional	Incremental >0 One roughing cut in infeed direction by default
FF	Finishing feed	Real	Optional	>0 Default setting: last programmed F-value
AW	Machining allowance of the unmachined part at the wall	Real	Optional	>0
AL	Machining allowance of the unmachined part at the base	Real	Optional	>0
RSW	Finishing allowance at the wall	Real	Optional	>0 No allowance by default
RSL	Finishing allowance at the base	Real	Optional	>0 No allowance by default
UL	Reference height	Real	Mandatory	Absolute
DT	Depth	Real	Mandatory	Absolute
RL	Retraction height	Real	Optional	Absolute UL+-SDI by default
CG	Tool use	Real	Optional	5 to 100 % 80 % is preset
SDI	Safety distance vertical to the machining plane	Real	Optional	>=0 2 mm by default
RD	Safety distance in plane	Real	Optional	>=0 2 mm by default
AT	Tangential approach length	Real	Optional	>=0 Default setting is 0
AN	Regular approach length	Real	Optional	>=0 Default if neither AT/AN/AR -> AN=RD
AR	Approach radius	Real	Optional	>= Tool radius or 0 Default setting is 0
RT	Tangential run-out length	Real	Optional	>=0 Default setting is 0

G278	User-defined contour pivot milling	G278(S,E,L,H,P,CV,CD,FF,AW,AL,RSW,RSL,UL,DT,RL,CG,SDI,RD,AT,AN,AR,RT,RN,RR)		
RN	Regular run-out length	Real	Optional	>=0 Default if neither RT/RN/RR -> RN=RD
RR	Run-out radius	Real	Optional	>= Tool radius or 0 Default setting is 0

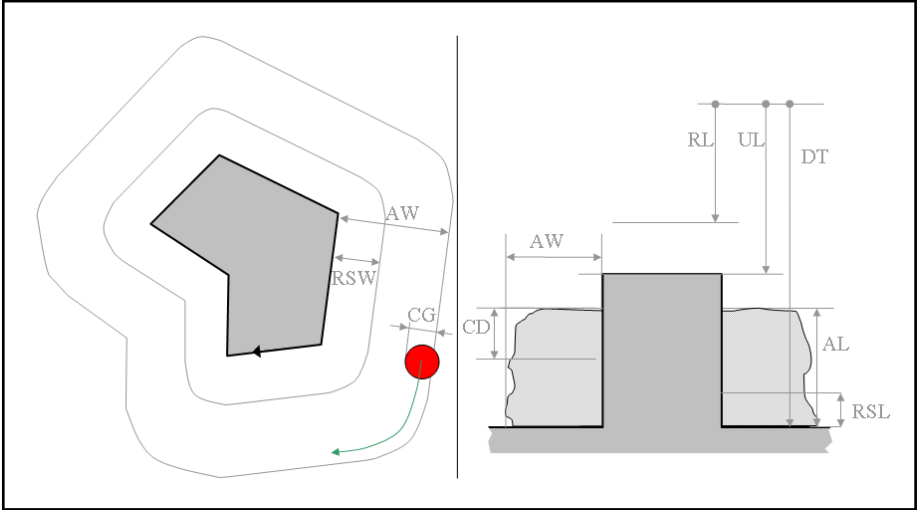


Fig. 8-11: Parameters of G278

The contour definition is described in the chapter [8.2 Defining the contour](#), page 147.

 If the contour is not closed [first point (line) and last point (line) are identical], a runtime error is generated.

 A single circle is no valid contour. A valid contour has thus to comprise of at least two elements.

Unprogrammed parameters:

If the machining allowance of the raw piece at the base **AL** is not programmed, the complete pin is machined (from **UL** to **DT**).

If the machining allowance of the raw piece at the wall **AW** is not programmed, the tool is traveled around the contour only once (per depth infeed). Tool use parameter **CG** is of no relevance in this case and thus ignored. In this case, roughing is performed around the contour and along an equidistant with distance $D = RSW + \text{tool radius}$, finishing with tool radius correction (G41(G42) on the contour.

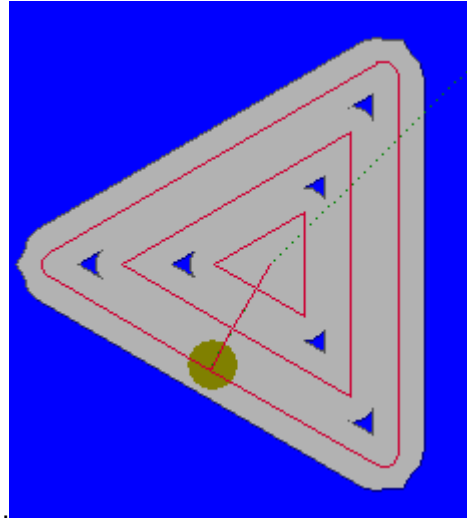
If the maximum infeed depth **CD** is not programmed, it is perpendicularly infeed only once to the machining plane. As a result, the parameter **AL** is irrelevant and is thus ignored. Roughing in $Z=DT+RSL$; finishing in $Z=DT$.

 If the cutting variant **CV** is selected and is thus only to be roughed, a warning on the finishing parameters (e.g., finishing infeed, approach, run-out) is not output. These parameters are ignored. In the opposite case (cutting variant = finishing), a warning on roughing parameters (e.g., machining allowance at wall, tool usage) is not output either.

	Sequence: The cycle machines the pin in the selected plane with a main axis and a secondary axis. After a depth has been completely milled, a depth infeed follows perpendicularly to the machining plane. The number of depth infeeds is calculated based on the maximum depth of the cut CD . The cycle machines all cuts up with the same depth.
Roughing outside the raw piece	In the first step, the tool is infed (rapid traverse) in the current machining plane next to the contour (distance RSW+RD). A possible active tool radius correction is switched off. The tool is infed perpendicularly to the plane in rapid traverse according to CD . The cycle mills to the contour as usual using the active feed until RSW is reached.
	 Regular infeed to the contour is always executed towards the starting point of the defined contour.
	Finally, the contour is machined with equidistant allowance RSW and the active feed is traveled.
	 The machining direction around the pin is the direction in which the contour was defined.
	After the contour has been milled, the tool is retracted from the contour by SDI and RD .
Roughing in the raw piece	In the first step, the tool is infed (rapid traverse) in the current machining plane next to the raw piece (distance AW+RD). A possible active tool radius correction is switched off. The tool is infed perpendicularly to the plane in rapid traverse according to CD . In the following steps, the tool is repeatedly infed as usual to the contour according to the programmed CG using the active feed and it is then milled on an equidistant around the contour.
	 The number of infeeds in the machining plane is calculated based on the maximum infeed CG * tool diameter. The cycle machines all cuts up with the same depth.
	 To ensure that collisions between contour and tool are detected, the cycle uses the collision monitoring NC function "Collision, CLN". After the cycle, collision monitoring is set to two monitoring blocks with "CLN(DEF)" and remains active.
	This sequence is repeated until the finishing allowance RSW is reached in the current depth infeed.
	After the contour has been milled, the tool is retracted from the contour by SDI and RD .
Finishing at base	As in Roughing in raw piece, page 159 , the tool is infed next to the raw piece and then the pin is milled on equidistant patterns. The approach and run-out

Milling cycles (contour-dependent)

motion according to the parameters **AN/AR/AT** and **RN/RR/RT** are only made



directly at the contour in the last cut.

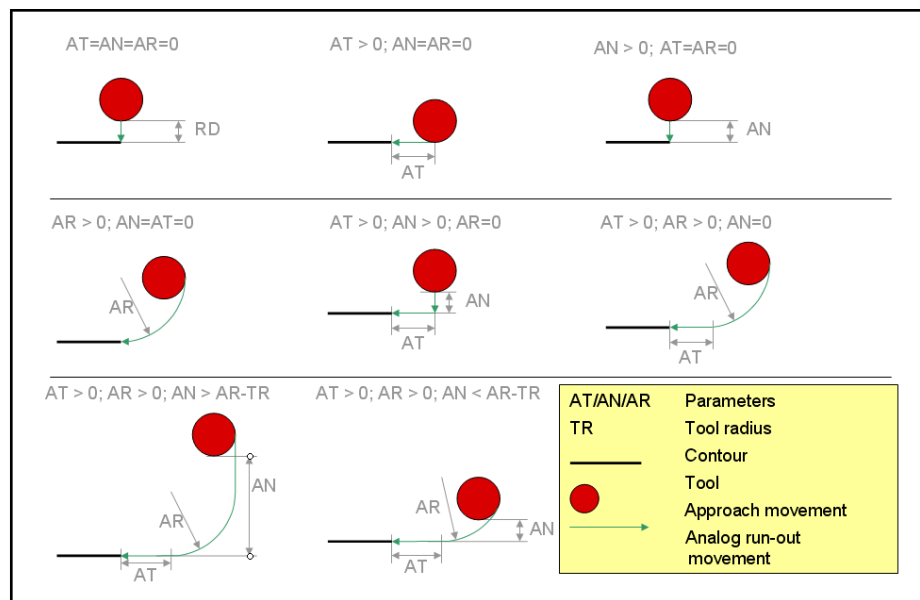


Fig. 8-12: Approach and run-out motion variants and parameters

The tool radius correction is only activated in this last step. The tool radius is compensated before by cut segmentation.



The feed rate of the center point path (feed correction NC function, cutter center point "G46") is specified for cutting along the contour (with active G41/G42). After the cycle, the initial state of the modal function is restored.

As in the following figure, "full" sections of concave contours are sometimes milled or parts of the raw piece are not machined completely. It is recommended in this case to machine the "pockets" of the convex contour before using the cycle [G273 User-defined contour pocket milling, page 147](#).

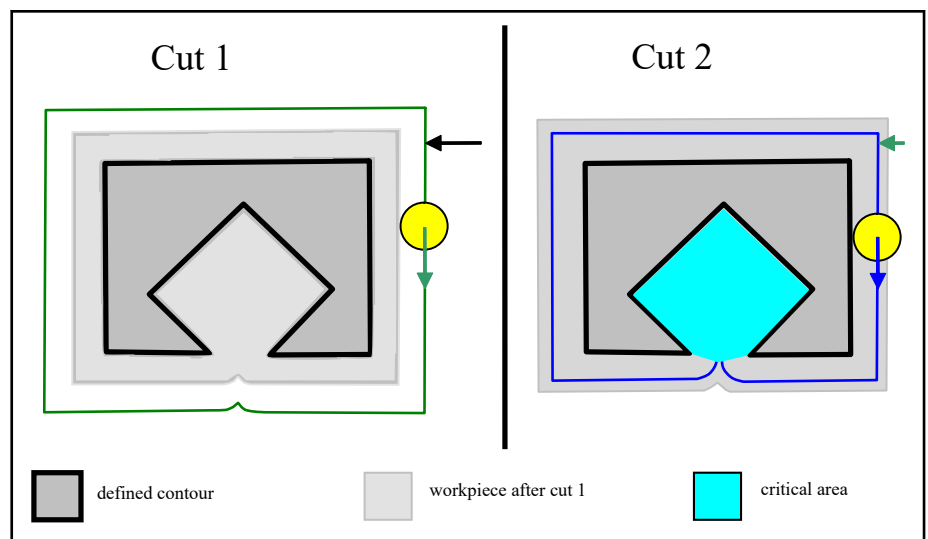


Fig. 8-13: Problem at concave contours

Special features/restrictions:

- It has to be ensured that approach and run-out as well as infeed motions can be made without collision.
- The radius during approach and run-out (**AR**, **RR**) has to be higher than the tool radius.
- The contour may not exceed 50 elements including the input help (RND, CHL, CHS, straight moves).
- The number of valid motions is limited to 500.

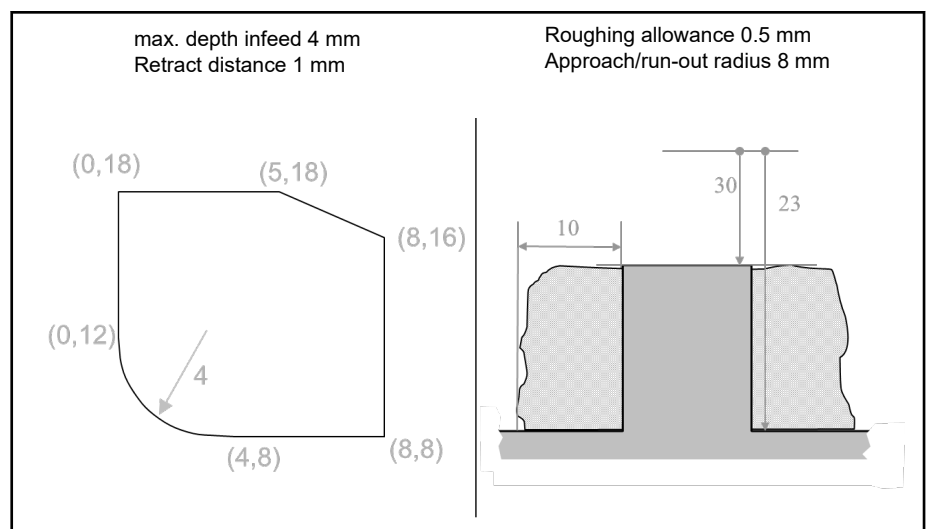
Example:

Fig. 8-14: Example of the cycle G278 Contour pivot milling

Program for G278 Contour pivot milling

```

N10 G17 G90 G94 F650 D1
N20 G0 X0 Y0 Z35
N30 G278 (S100,E160,UL30,DT23,AW10,CD4,RSL0.5,RSW0.5,SDI1,RD1,AR8,RR8)
N40 M30

;*****
;*** Contour description ***
;*****
N100 G0 X5 Y18
N110 X0
N120 Y12

;Initialization
;Approach starting point
;Call cycle
;Program end

```

```
N130 G3 X4 Y8 R4
N140 G0 X8
N150 Y16
N160 X5 Y18
```

8.5 G290 Path milling

The cycle "Path milling" can mill along a contour specified by cut segmentation. The contour is machined with multiple depth infeeds. Depending on the parameterization, roughing cuts (with finishing allowance) and finishing cuts are machined.

NOTICE

Due to the extraordinarily high number of possible combinations between geometry and machining variance, the calculated cut segmentation can sometimes not meet the user requirements or even violate the contour.

Due to this reason, the NC programmer has to carefully check and implement the results of the cycle. A recommended tool is the NC simulation.

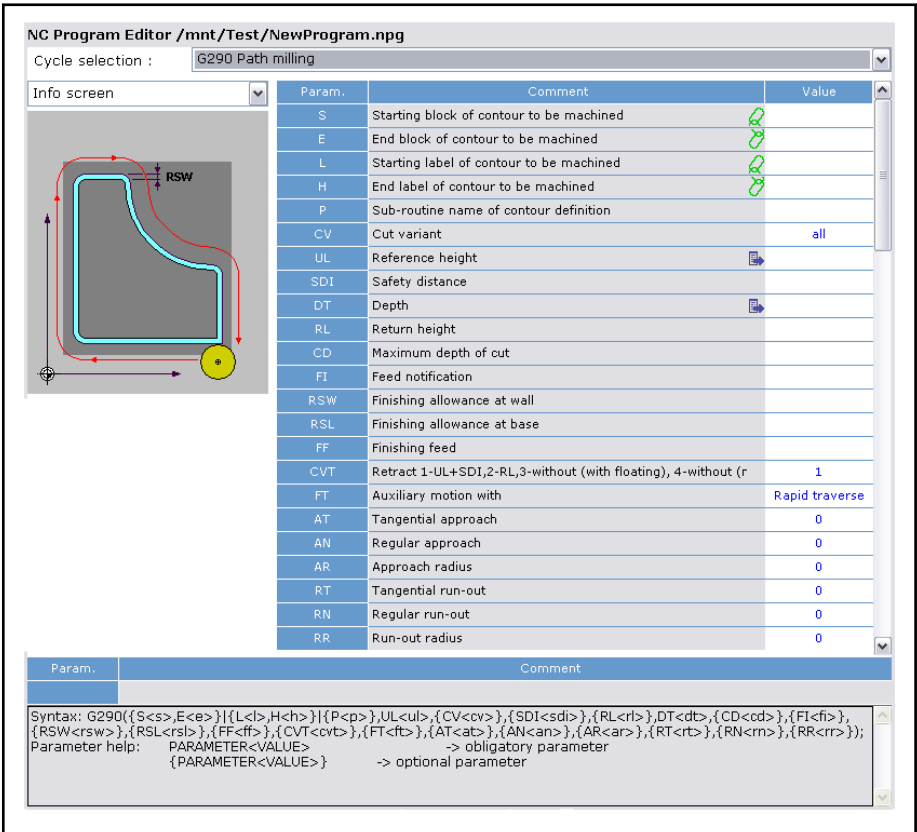


Fig. 8-15: G290 Path milling

G290	Path milling	G290(S,E,L,H,P,CV,CVT,FT,CD,FF,FI,RSW,RSL,UL,DT,RL,SDI,AT,AN,AR,RT,RN,RR)		
S	Starting block of the contour to be machined	String	Optional	
E	End block of the contour to be machined	String	Optional	
L	Starting label of the contour to be machined	String	Optional	
H	End label of the contour to be machined	String	Optional	
P	Subroutine name of the contour definition	String	Optional	
CV	Machining (cutting) variant	Integer	Optional	1 Roughing 2 Complete 3 Finishing 2 is the default setting
CVT	Cut segmentation algorithm	Integer	Optional	1 Retract to UL+SDI 2 Retract to RL 3 Bidirectional 4 No retract 1 is preset
FT	Type of interim motions	Integer	Optional	1 In rapid traverse 2 With feed 1 is preset
CD	Maximum infeed depth	Real	Optional	Incremental >0 One roughing cut in infeed direction by default
FF	Finishing feed	Real	Optional	>0 Default setting: last programmed F-value
FI	Depth infeed feed	Real	Optional	>0 or -1 for rapid traverse Default setting: last programmed F-value
RSW	Finishing allowance at the wall	Real	Optional	>0 No allowance by default
RSL	Finishing allowance at the base	Real	Optional	>0 No allowance by default
UL	Reference height	Real	Mandatory	Absolute
DT	Depth	Real	Mandatory	Absolute
RL	Retraction height	Real	Optional	Absolute UL+-SDI by default
SDI	Safety distance vertical to the machining plane	Real	Optional	>=0 2 mm by default

Milling cycles (contour-dependent)

G290	Path milling	G290(S,E,L,H,P,CV,CVT,FT,CD,FF,FI,RSW,RSL,UL,DT,RL,SDI,AT,AN,AR,RT,RN,RR)		
AT	Tangential approach length	Real	Optional	≥ 0 Default setting is 0
AN	Regular approach length	Real	Optional	≥ 0 with G41/G42 Default setting is 0
AR	Approach radius	Real	Optional	$\geq$ Tool radius or 0 Default setting is 0
RT	Tangential run-out length	Real	Optional	≥ 0 Default setting is 0
RN	Regular run-out length	Real	Optional	≥ 0 with G41/G42 Default setting is 0
RR	Run-out radius	Real	Optional	$\geq$ Tool radius or 0 Default setting is 0

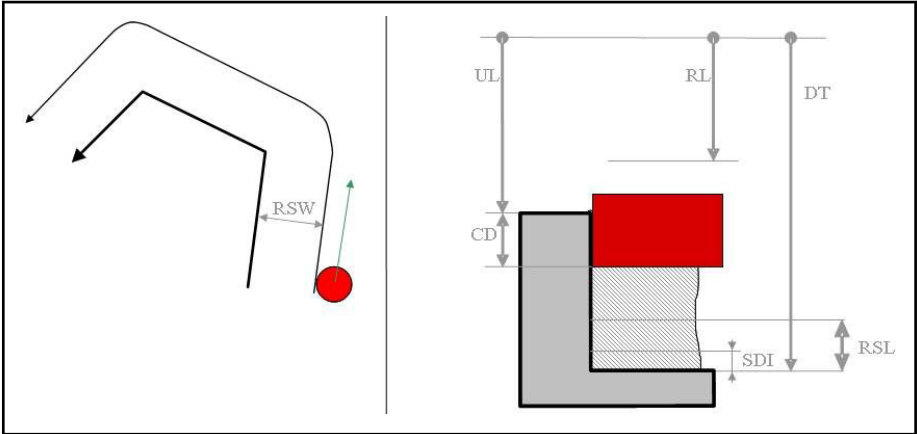


Fig. 8-16: Parameters for G290

The definition of the contour is described in [chapter 8.2 "Defining the contour"](#) on page 147.



The contour can be open or closed.

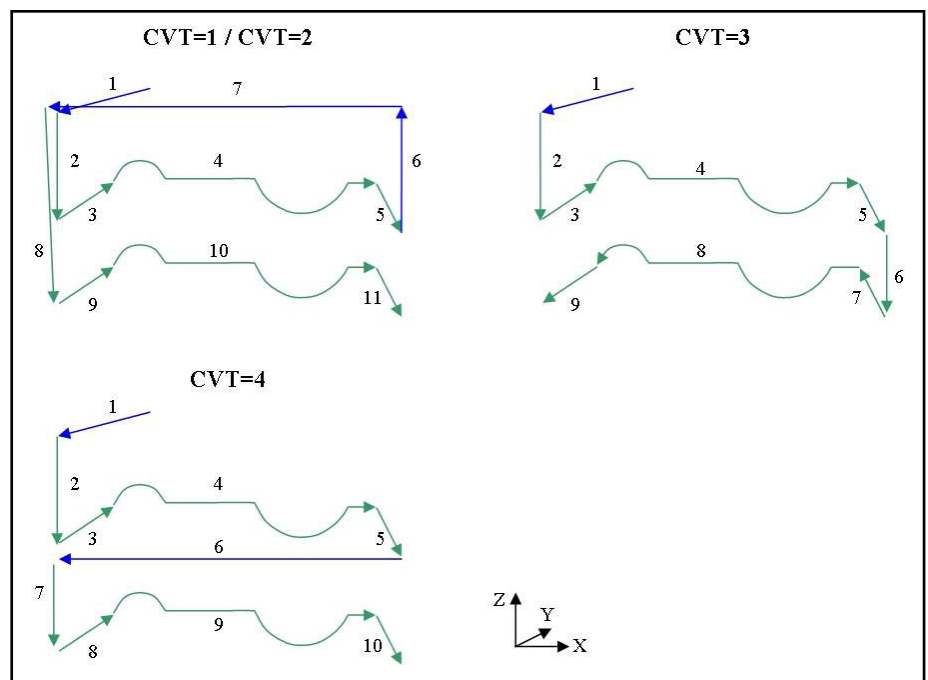


Fig. 8-17: Cut segmentation algorithm parameter for G290 (blue: rapid traverse, green: feed)

Unprogrammed parameters:

If maximum infeed depth **CD** is not programmed, only one infeed cut is executed perpendicularly to the machining plane. Roughing: $Z = DT + RSL$; finishing: $Z = DT$.

If the retract height **RL** was not programmed, the cycle moves to the upper height plus **UL** the safety distance **SDI** after machining. In this case, the tool is positioned at $Z = UL + SDI$ after the cycle.



If only roughing is selected via the cutting variant **CV**, a warning on the finishing parameters (e.g., finishing infeed, approach, run-out) is not output. These parameters are ignored. In the opposite case (cutting variant = finishing), a warning on roughing parameters (e.g., finishing allowances) is not output either.

Sequence:

The cycle machines the specified path in the selected plane with a main axis and a secondary axis. After a depth has been completely milled, a depth infeed follows perpendicularly to the machining plane. The number of depth infeeds is calculated based on the maximum depth of the cut **CD**. The cycle machines all cuts up with the same depth.

Roughing

In the first step, the tool is infeed (rapid traverse) to the starting point for the approach motion in the current machining plane. Then, the tool is moved perpendicularly to the plane according to **CD** in rapid traverse or with feed for the maximum depth of cut **FI**.

This is followed by an approach motion according to **AT/AN/AR** and the cycle mills along the contour (offset by finishing allowance **RSW** and tool radius) without tool radius correction.



The machining direction along the contour is the direction in which the contour was defined. The only exception are straight paths at bidirectional machining.

After the end of the contour has been reached, the cycle generates the run-out motion according to **RT/RN/RR**.

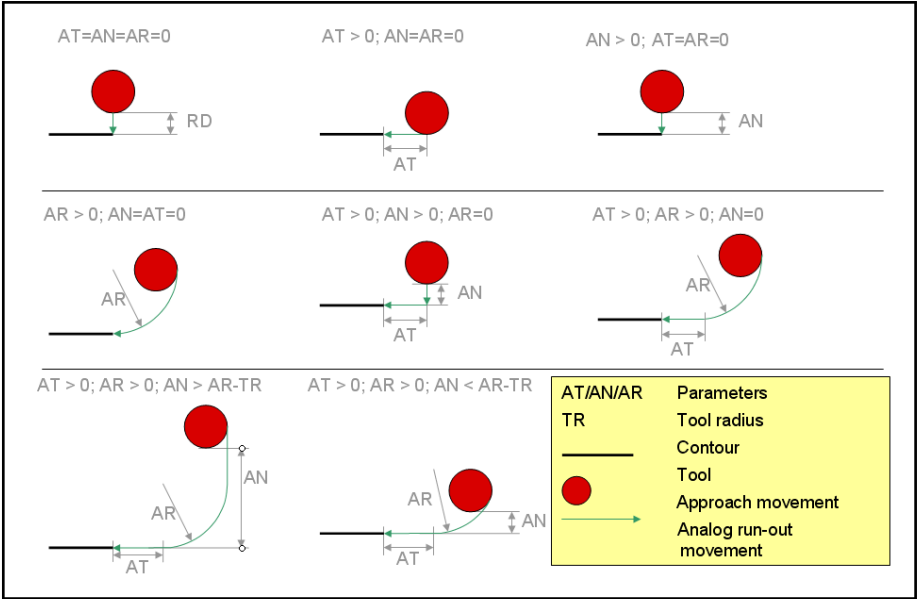


Fig. 8-18: Approach and run-out motion variants and parameters

After milling, it is either retracted or returned directly to the starting position of the following approach motion depending on **CVT** (see Fig. 8-17 [Cut segmentation algorithm parameter for G290](#) (blue: input, green: feed), page 165). Further depth infeds follow until finishing allowance **RSL** has been reached.

Finishing

Finishing takes place as roughing. In this machining step, however, the tool mills directly along the contour (without finishing allowance) with active tool radius correction. The tool is infed perpendicularly to the plane in rapid traverse according to **CD**. At this point, the tool radius correction (G41/G42) is activated. Subsequently, the tool mills on the contour.



The feed rate of the center point path (feed correction NC function, cutter center point "G46") is specified for cutting along the contour (with active G41/G42). After the cycle, the initial state of the modal function is restored.

Effect of tool radius correction
G40/G41/G42:

The cycle defines cut segmentation according to the tool radius correction that was active when the cycle was called. This defines finishing allowance at the wall **RSW** as well as the direction of the approach and run-out motions. The appropriate tool radius correction is activated during finishing along the contour.

If, for example, G42 (machining to the right of the workpiece) is active when the cycle is called, the finishing allowance acts to the right (as seen in direction of the contour), approach and run-out motions from the right and G42 is activated during finishing.

A special case is a deactivated tool radius correction (G40). In this case, the tool is traveled along the defined contour. If programmed, finishing allowance at the wall **RSW** results in a runtime error. The signs of the normal lengths of approach **AN** and run-out **RN** can be used to specify the direction from which approach and run-out are generated. If a radius for approach and run-out

AR/RR was programmed, a normal length **AN** and **RN** has to be specified as well to ensure that the direction of the arc is known to the cycle.

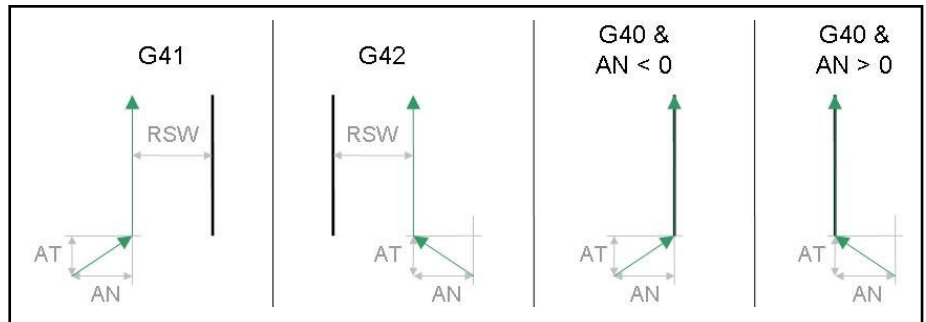


Fig. 8-19: Effect of tool radius correction G40/G41/G42

Special features/restrictions:

- It has to be ensured that approach and run-out as well as infeed motions can be made without collision.
- The radius during approach and run-out (**AR, RR**) has to be higher than the tool radius.
- The contour may not exceed 50 elements including the input help (RND, CHL, CHS, straight moves).
- The number of valid motions is limited to 500.
- There is no monitoring for collision with the contour while traveling to the starting point.

Example:

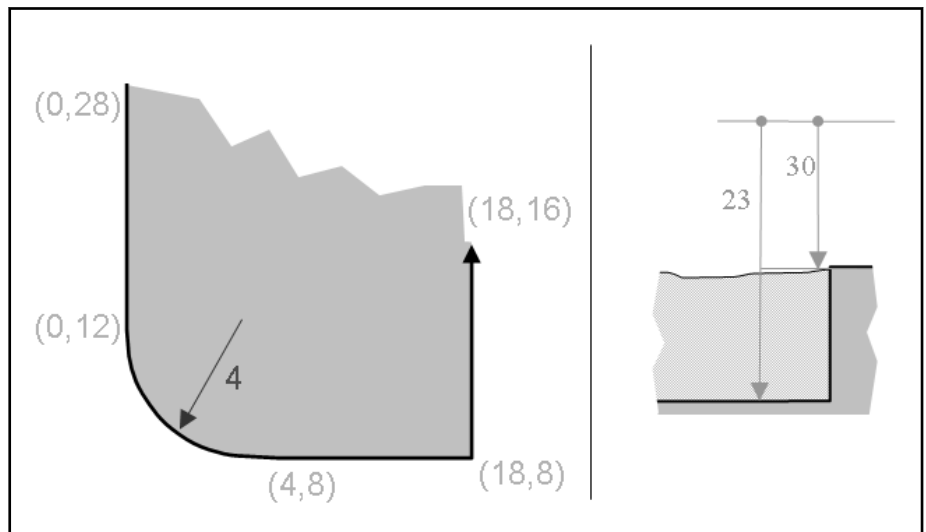


Fig. 8-20: Path milling example

The path is to be milled with a maximum infeed depth of 4 mm. The safety distance is 1 mm, the roughing allowances are 0.5 mm each and the approach and run-out radiuses are 6 mm each. After the cycle, the tool is to be traveled to Z = 40.

Program for G290 Path milling

```
G0 X0 Y40 Z40 G17 G90 D1 F1000 G42
N20 G0 X0 Y30 Z35
G290(S100, E150, UL30, DT23, CD4, AR 6, RR6, SDI1, RSW0.5, RSL0.5, RL40)
G40
N40 M30
;*****
;*** Contour description ***
;Initialization
;Approach starting point
;Call cycle
;Switch off tool correction
;Program end
```

Milling cycles (contour-dependent)

```

;*****
N100 G0 X0 Y28
N120 Y12
N130 G3 X4 Y8 R4
N140 G0 X18
N150 Y16

```

8.6 Milling contour pocket with islands cycle

8.6.1 General information

The cycle [chapter 8.6.4 G281 Milling contour pocket with islands, page 172](#) can mill a pocket with a user-defined contour by cut segmentation. The pocket can include islands also defined by any other contours.

Contour islands for the pocket with islands are defined using the following cycle G288. The cycle G289 (see [chapter 8.6.3 G289 Defining Starting Points, page 170](#)) can also specify starting points for the pocket with islands.

These two auxiliary cycles do not execute any motion. They only provide data for the cycle "Milling contour pocket with islands".



Both auxiliary cycles save their data (island contours and starting points) in an internal memory. This memory is only deleted if the cycle is programmed with parameter R.

It is recommended to initialize the relevant memory upon each new island and starting point sequence and a pocket. Parameter R is programmed with the first island definition and the first starting point definition (see examples).

If this internal memory is to be deleted during home positioning, G288 (R) or G289 (R) has to be programmed in "State after channel home position" (machine parameter CHAN/Ch[k]/Ini/ChResetState) or in the init program.

8.6.2 G288 Defining island contour

The cycle "Define contour islands" is an auxiliary cycle to "mill a contour pocket with islands". Islands traced out when G281 is called are defined here.



This cycle only defines data. No motion is generated.

G288	Define islands	G288(R,S,E,L,H,P)		
R	Deleting all previous island contours	-	Optional	
S	Starting block of the island contour	String	Optional	Together with E
E	End block of the island contour	String	Optional	Together with S
L	Starting label of the island contour	String	Optional	Together with H
H	End label of the island contour	String	Optional	Together with L
P	Subroutine name of the island contour	String	Optional	

The island contour is defined as the pocket contour described in [chapter 8.6.4 G281 Milling contour pockets with islands, page 172](#).



Before loading the island contour, all input helps (moving, mirroring, rotating) are switched off. After the cycle, the state at cycle call is restored.



If the contour is not closed [first point (line) and last point (line) are identical], a runtime error is generated.



A single circle is no valid contour. A valid contour has thus to comprise of at least two elements.

One island contour can be defined per G288 call. It is not allowed to program several G288 in one block. G288 causes a stop on the path.



In a block with contour reference (as G288 with island definition), no other NC function with contour reference (e.g. another G288 or G273) may be programmed. Therefore, no other NC function is to be programmed in a block with G288.

The defined island contours are stored in an internal memory. This memory is deleted when programming the parameter **R** (without value). If **R** is programmed and an island is defined in a G288 cycle, the cycle deletes the memory first and then stores the specified island contour.

If an island contour is located completely outside the pocket contour, this island is ignored. An island contour can be completely or partially located in the pocket contour. This allows an easy creation of complex pocket contours.



Starting points and island contours are also retained after calling G281.

Special features/restrictions:

- The sum of the contour elements (distances and arc segments) of all island is limited to 50.
- Only one G288 may be programmed per block.
- All island contours have to be closed.

Milling cycles (contour-dependent)

Example:

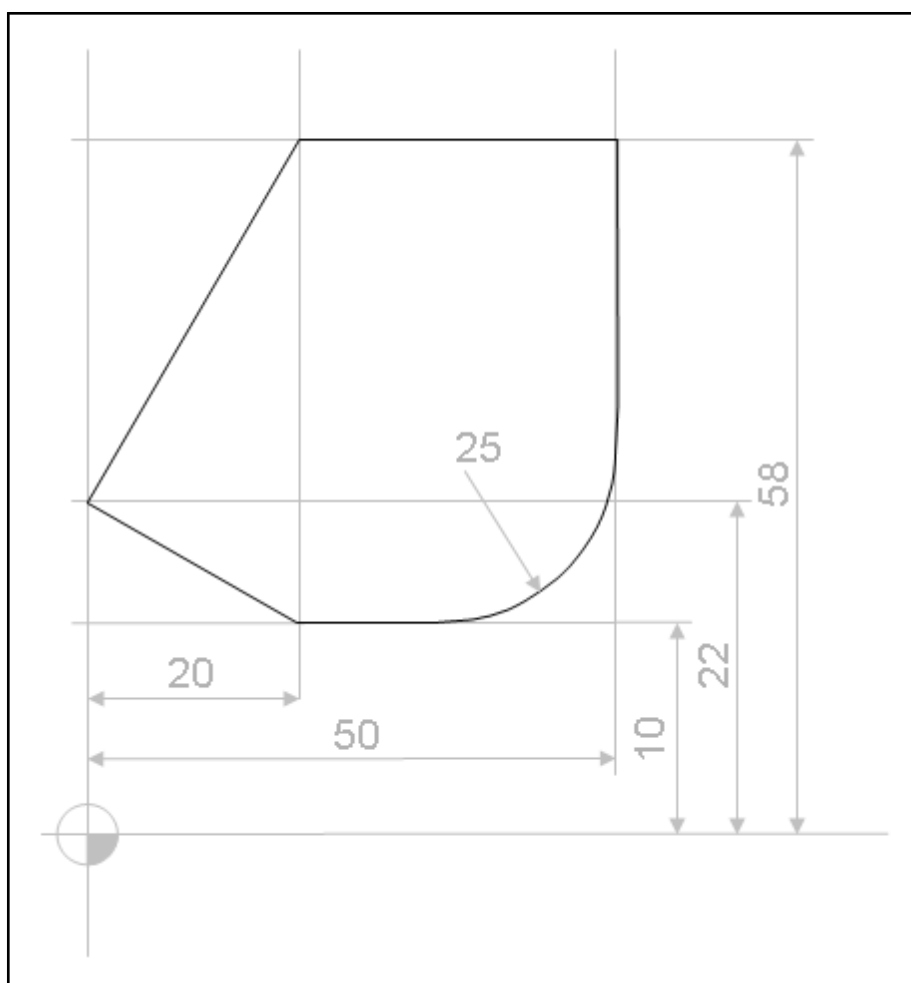


Fig. 8-21: G288 Define Island Contour
Program for G288 Define island contour

```

...
N100 G288 (S1000,E1050,R)          ; Define island; delete memory before
...
N600 M30                            ; Program end

;*****
;*** Contour description ***
;*****
N1000 G1 X20 Y10
N1010 X50 RND(25)
N1020 Y58 RND()
N1030 X20
N1040 X0 Y22
N1050 X20 Y10

```

8.6.3 G289 Defining starting points

The cycle "Define Starting Points" is an auxiliary cycle for [chapter 8.6 G281 Milling contour pockets with islands, page 168](#). The starting points at which the cycle G281 is to plunge in are defined here.



This cycle only defines data. No motion is generated.

G289	Defining starting points	G289(R,S,E,L,H,P,MSP,SSP)		
R	Deleting all previous starting points	-	Optional	
S	Starting block of a drilling program	String	Optional	Together with E
E	End block of a drilling program	String	Optional	Together with S
L	Starting label of a drilling program	String	Optional	Together with H
H	End label of a drilling program	String	Optional	Together with L
P	Subroutine name of a drilling program	String	Optional	
MSP	Starting position in the main axis	Real	Optional	Absolute
SSP	Starting position in the secondary axis	Real	Optional	Absolute

There are two options to define starting points:

- via a drilling program or
- as individual points.

Only one of the two variants may be programmed in a G289.

If the plunging points were machined by a drilling program in a previous process step, the drilling program can be used again to define the starting points for the cycle. The drilling program is specified in G289 like a contour definition. The cycle determines the end points of each motion and stores them as starting points.

The individual points can also be specified as main and secondary axis value.

One drilling program can be specified per G289 call. Only one G289 is thus to be programmed per block. The NC function G289 causes a stop in the block.



In a block with contour reference (as G289 with drilling program), no other NC function with contour reference (e.g. another G288 or G273) may be programmed. Therefore, no other NC function is to be programmed in a block with G289.

The defined starting points are stored in an internal memory. This memory is deleted when programming the parameter **R** (without value). If **R** is programmed and a starting point is defined in a G289, the cycle deletes the memory first and then stores the specified starting point.

If a starting point is located completely outside the pocket contour, this point is ignored. If a starting point is closer at a pocket or islands contour than the $\text{ToolRadius} + \text{RD}$, the cycle outputs a warning.

If starting points are defined, the cycle **G281** uses only these positions for plunging. It is always plunged in vertically (irrespective of **ANG**). The plunging velocity depends on the parameter **FI**.



Areas, which cannot be reached by the specified points using the current tool, are not machined.



Starting points and island contours are also retained after calling G281.

Special features/restrictions:

- The number of starting points is limited to 30.

Milling cycles (contour-dependent)

- Only one G289 may be programmed per block.

Example: The following example defines these points as plunging points:

	X	Y
1. Point	50	28
2. Point	0	28
3. Point	0	112
4. Point	40	85
5. Point	35	47

*Tab. 8-1: G289 Defining Starting Points
Program for G288 Defining starting points*

```

...
N10 G17
...
N100 G289 (S100,E130,R)      ; Specify drilling program; delete memory before
N110 G289 (MSP35,SSP47)      ; Define starting point
...
N1000 M30                    ; Program end

;*****
;*** Drilling program ***
;*****
N100 G0 X50 Y28
N110 X0
N120 Y112
N130 X40 Y85

```

8.6.4 G281 Milling contour pocket with islands

NOTICE

Due to the extraordinarily high number of possible combinations between geometry and machining variance, the calculated cut segmentation can sometimes not meet the user requirements or even violate the contour.

Due to this reason, the NC programmer has to carefully check and implement the results of the cycle. A recommended tool is the NC simulation.

G281	Milling contour pocket with islands	G281(S,E,L,H,P,CV,CVT,CD,FF,AW,AL,RSW,RSL,UL,DT,RL,CG,SDI, RD,ANG,FI)		
S	Starting block of the contour to be machined	String	Optional	
E	End block of the contour to be machined	String	Optional	
L	Starting label of the contour to be machined	String	Optional	
H	End label of the contour to be machined	String	Optional	
P	Subroutine name of the contour definition	String	Optional	

G281	Milling contour pocket with islands	G281(S,E,L,H,P,CV,CVT,CD,FF,AW,AL,RSW,RSL,UL,DT,RL,CG,SDI, RD,ANG,FI)		
CV	Machining (cutting) variant	Integer	Optional	1 Roughing 2 Complete 3 Finishing 4 Predrilling 2 is the default setting
CVT	Cutting variant	Integer	Optional	1 Equidistant 2 Meander 1 is preset
CD	Maximum infeed depth	Real	Optional	Incremental >0 One roughing cut in infeed direction by default
FF	Finishing feed	Real	Optional	>0 Latest programmed F-value by default
AW	Machining allowance of the raw piece at the wall	Real	Optional	>0
AL	Machining allowance of the raw piece at the base	Real	Optional	>0
RSW	Finishing allowance at the wall	Real	Optional	>0 No allowance by default
RSL	Finishing allowance at the base	Real	Optional	>0 No allowance by default
UL	Reference height	Real	Mandatory	Absolute
DT	Depth	Real	Mandatory	Absolute
RL	Retraction height	Real	Optional	Absolute UL+-SDI by default
CG	Tool use	Real	Optional	5 to 100 % 80 % is preset
SDI	Safety distance vertical to the machining plane	Real	Optional	>=0 2 mm by default
RD	Safety distance in plane	Real	Optional	>=0 2 mm by default
ANG	Angle during plunging	Real	Optional	0 < ANG <= 90 Default 90
FI	Infeed feed when plunging	Real	Optional	> 0 50% of the previously programmed F-value by default or -1 for rapid traverse

G281	Milling contour pocket with islands	G281(S,E,L,H,P,CV,CVT,CD,FF,AW,AL,RSW,RSL,UL,DT,RL,CG,SDI, RD,ANG,FI)		
MA	Meander angle with main axis MA	Real	Optional	-360 <= MA <= 360 0 by default
LTR	Latest tool radius (residual cut)	Real	Optional	> 0

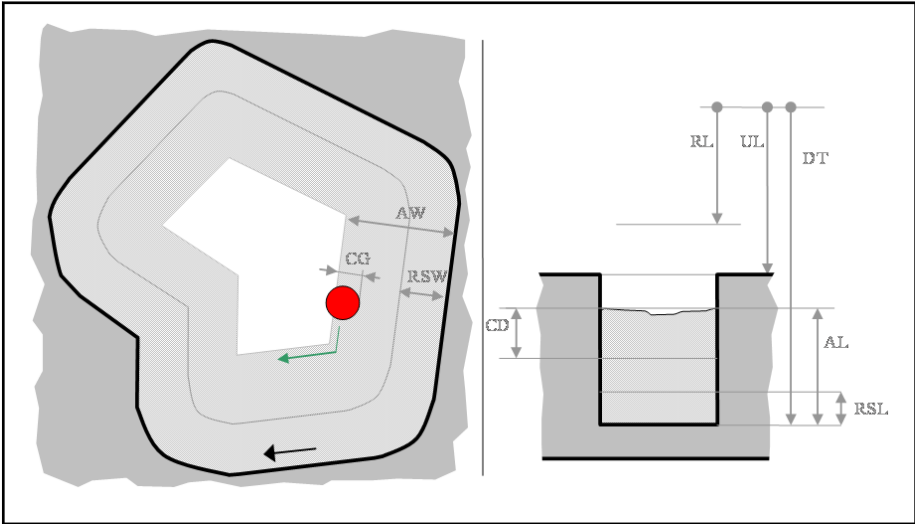


Fig. 8-22: Parameters of G281

The contour definition is described in the chapter [8.2 Defining the Contour](#), [page 147](#).

-  If the contour is not closed [first point (line) and last point (line) are identical], a runtime error is generated.
-  A single circle is no valid contour. A valid contour has thus to comprise of at least two elements.

Unprogrammed parameters:

If the machining allowance of the raw piece at base **AL** is not programmed, the cycle assumes that the pocket is completely filled. In this case, the pocket is roughed from **UL** to **DT**.

If the machining allowance of the raw piece at the wall **AW** is not programmed, it is only traveled once along the contour above **AL**. Below **AL**, a "full" pocket is assumed.

If the maximum infeed depth **CD** is not programmed, it is perpendicularly infeed only once to the machining plane. As a result, the parameter **AL** is irrelevant and is thus ignored. Roughing in $Z=DT+RSL$; finishing in $Z=DT$.

 If the machining variant **CV** is selected and thus only to be roughed, a warning on the finishing parameters (e.g., finishing infeed, approach, run-out) is not output. These parameters are ignored. In the opposite case (machining variant = finishing), a warning on roughing parameters (e.g., machining allowance at wall, tool use) is not output either.

The parameter of the cutting variant **CVT** can define how the cycle machines a plane (depth infeed).

Possible variants are:

- Machining along the equidistant (CVT = 1)

- Machining by means of meander patterns (CVT = 2)

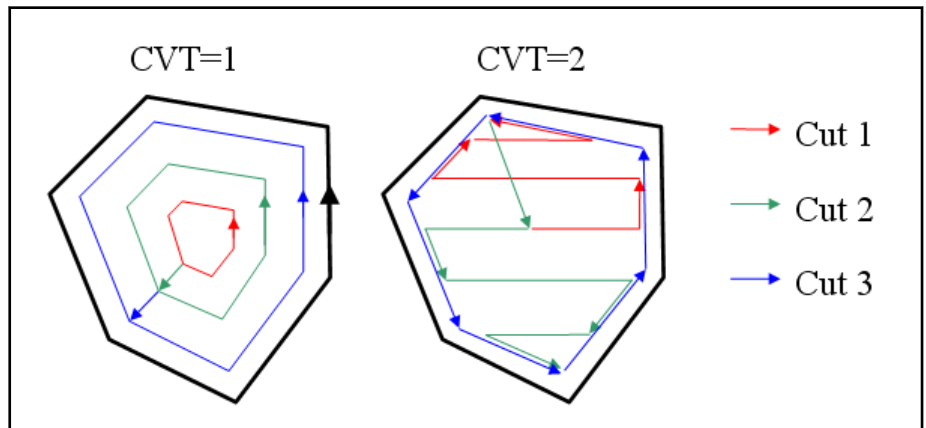


Fig. 8-23: Cutting variant: Equidistant (CVT = 1) and meander (CVT = 2)

If machining along the **equidistant** is selected, the tool is always infed as usual to the next equidistant towards the contour.

If the plane is machined on **meander patterns**, the tool starts at one side (or at the starting point) and

1. machines the pocket along the meander patterns,
2. then mills the resulting step profile along the pocket contour
3. and finally along the island contours of the step profile.

The meander angle **MA** can rotate the meander patterns in the machining plane. If 0° (default) is selected, the meander patterns extend in parallel to the main axis.

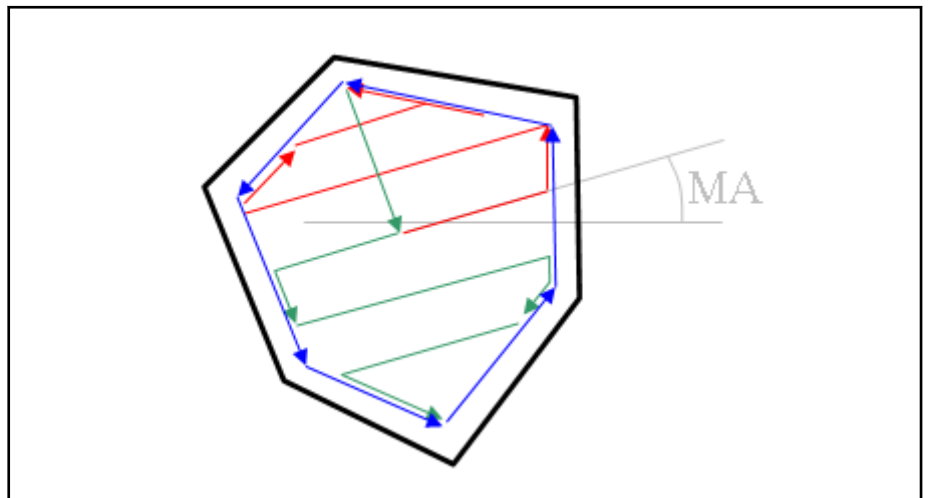


Fig. 8-24: Rotating meander patterns around the angle MA

The infeed method vertical to the selected plane depends on the parameters "infeed angle" **ANG** and "raw piece allowance" at the wall **AW**. If it is infed with feed and the infeed feed **FI** parameter was programmed, the cycle enables this feed for depth infeed. A value of **FI**=-1 means "rapid traverse".

Milling cycles (contour-dependent)

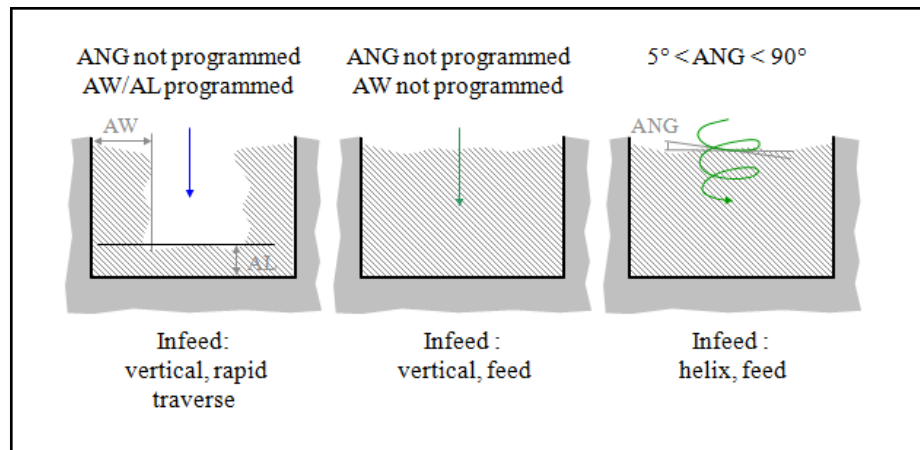


Fig. 8-25: Infeed variants for G273

Machining variants of G281

There are four different cycle modes. The technological sequence is:

1. 1. Predrilling
2. 2. Roughing
3. 3. Machining remaining material
4. 4. Finishing

Predrilling

Cycle:

All relevant parameters for a later machining are transferred to the cycle (incl. parameter **CV=4**). The cycle uses them to calculate a cut segmentation including suitable plunging points. The generated subroutine only contains these plunging points and is automatically processed. The parameter **LTR** indicates the size of the radius of the future milling tool. All points drilled like that are automatically defined as starting points.

When calculating drilling points, a full pocket is always assumed. The parameters **AW** and **AL** are ignored.

Application:

Prior to starting, define all islands. Change the drilling tool, select a modal drill cycle and enable the desired modal function for the motions between the drilling points (e.g. "G0(IPS1)"). Subsequently, call the cycle in predrilling mode. The drilling points of the generated program are approached and drilled at those respective points.

Roughing

Cycle:

Roughing parameters (incl. parameter **CV=1** or **CV=2**, **LTR** may not be programmed) are transferred to the cycle. The cycle uses this to calculate a cut segmentation. Depending on the parameter **CVT**, this is based on the equidistant or meander patterns. The cut segmentation is automatically machined.

Application:

Define islands and starting points (both optional). Then, fire the roughing tool and call the cycle. The cycle removes from the pocket what the tool (due to its radius) allows.

Machining remaining material

Cycle:

Roughing or finishing parameters are transferred to the cycle (incl. parameter **CV=1** or **CV=3**, **LTR** has to be programmed). The cycle determines the material which could not be machined by the latest roughing tool (radius **LTR**). These parts are milled (as far as possible) using the current tool.

Bigger areas of the remaining material are machined on equidistant patterns.

If finishing is used in the "machine remaining material" mode, it includes base and wall finishing. At base finishing, the remaining material in the pocket is removed. At wall finishing, it is traveled once along all contours.

The parameters **AW** and **AL** as well as the starting points are ignored when cutting the residual cut. A full pocket already machined with a tool with the radius **LTR** is always assumed.

Application:

The cycle has already been executed with **CV=1**. Then, fit a roughing tool with a smaller radius and call the cycle again. This time, the radius of the old (big) tool is transferred as parameter **LTR**. The cycle removes from the pocket what the current tool (due to its radius) allows.

Finishing

Cycle:

Finishing parameters are transferred to the cycle (incl. parameter **CV=2** or **CV=3**). The cycle executes a wall finishing (**RSW>0**) and/or a base finishing (**RSL>0**). Base finishing always includes a wall finishing (last cut). If no **RSW** was specified, the tool radius*0.5 is assumed.

The base finishing uses the same cutting variants (equidistant/meander) as the roughing. Thus, the respective parameters have to be programmed. At wall finishing, it is traveled along the contour via a quadrant with the radius "tool radius+2mm". Traveling is carried out analogously.

For all island contours at which can be traveled along using the finishing tool, a separate wall finishing is performed. This is preferably executed in the direction the island was defined.

Defined starting points are ignored while finishing.

Application:

The cycle has already been executed with **CV=1**. Then, fit the finishing tool and call the cycle again. The cycle finishes the pocket.

Special features/restrictions:

- The pocket contour may not exceed 50 elements including the input tool (RND, CHL, CHS, straight moves).
- The pocket contour has to be closed.
- The number of valid motions is limited to 1500.

If starting points are defined, the cycle **G281** uses only these positions for plunging. It is always plunged in vertically (irrespective of **ANG**). The plunging velocity depends on the parameter **FI**.



Areas, which cannot be reached by the specified points using the current tool, are not machined.

Simple example:

The following pocket (with island) is to be machined. First, rough, then finish. The milling tool (radius 5mm) is already fit in. The finishing allowance at the wall is to be 0.8 mm and 0.5 mm at the base. The maximum depth of cut of the tool is 6 mm. Perform roughing and finishing at a feed of 500 mm/min.

Milling cycles (contour-dependent)

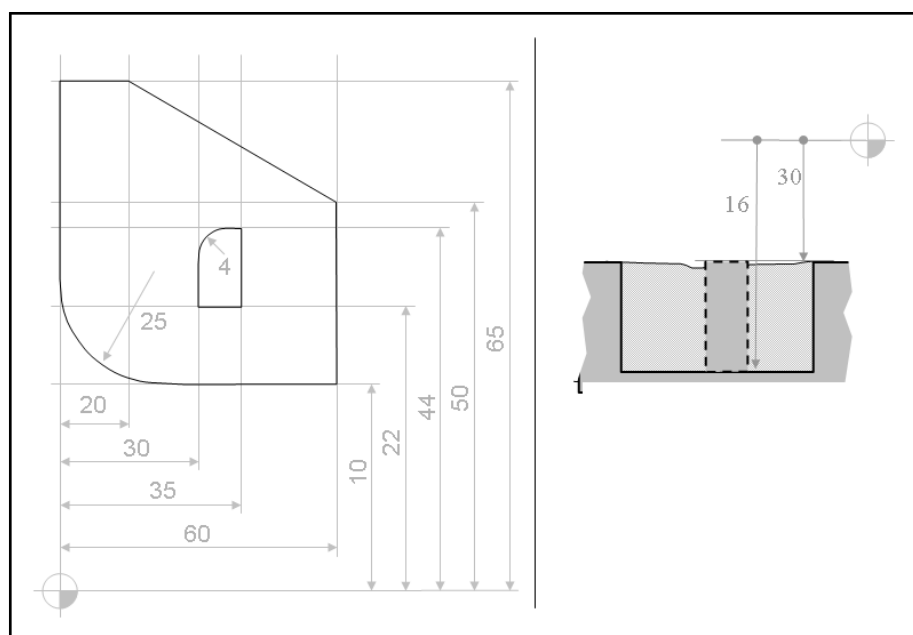


Fig. 8-26: G281 Milling contour pocket with islands

Simple program for G281 Milling contour pocket with islands

```

N10 ED1 G90 G94 G17 G0 X0 Y0 Z30 F500      ; Initialization
N20 G288(R, S500, E540)                    ; Defining an island, prior to this, delete memory
N30 G289(R)                                ; If required, delete existing starting points
N40 M3 S500                                ; Start spindle
N50 G281(L PocketStart, H PocketEnd, UL30, DT16, CD6, RSW0.8, RSL0.5) ; Cycle call
N60 M30                                    ; Program end

;*****
;*** Island description ***
;*****
N500 G1 X30 Y25
N510 X35
N520 Y44
N530 X30 RND(4)
N540 Y25 RND()

;*****
;*** Pocket description ***
;*****
N600 PocketStart:
G1 X0 Y65
X20
X60 Y50
Y10 G1(RND25) X0
Y65
N610 PocketEnd:

```

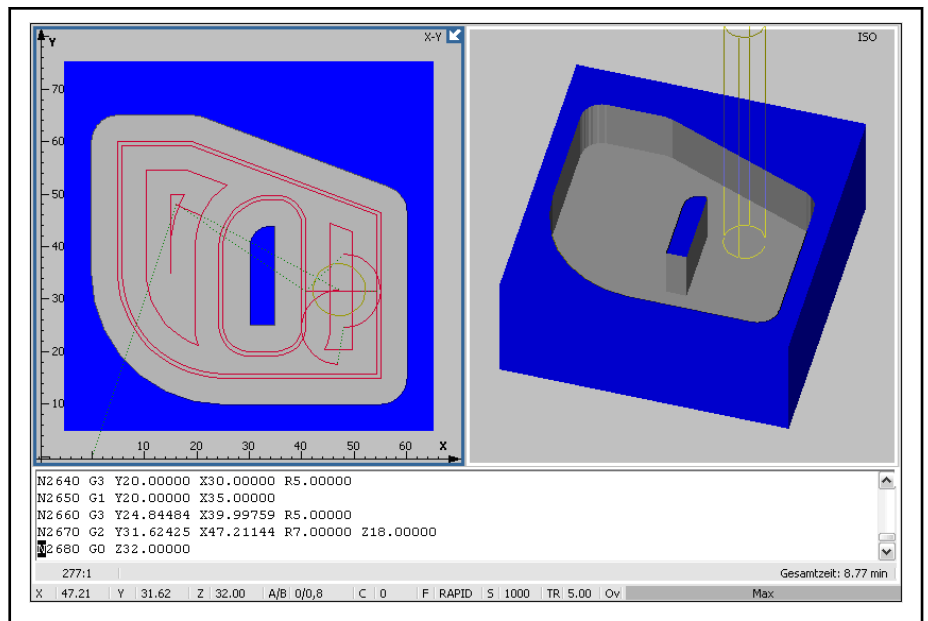



Fig. 8-27: G281 Milling contour pocket with islands

Complex example:

A pocket with islands is to be machined. In the first step, plunging points are drilled with a drilling tool (T5). In the second step, it is roughed with a milling tool with a big radius (T1, ToolRadius 25). Then, a smaller roughing tool is fit in (T2, ToolRadius 5) to machine the remaining material. At the end, it is finished with a separate finishing tool (T4, ToolRadius 5).



The example shows a schematic procedure. Thus, all irrelevant data (such as parameter values, pocket/islands contours) is removed. Thus, the example is not executable anymore.

Complex program for G281 Milling contour pocket with islands

```

N10 T5 M6 ED5                ; Fit predrilling tool
N20 G17 G90 X... Y... Z...    ; Approach position via pocket
N30 G288(P Insel1.npg, R)     ; Define island contour (delete old one)
N40 G288(P Insel2.npg)        ; Define island contour
N50 G288(P Insel3.npg)        ; Define island contour
N60 G289(R)                   ; Delete previously defined starting points
N70 F...                      ; Select drilling feed
N80 G81 (...)                 ; Select modal drilling cycle
N90 G281(P Tasche.npg, LTR..., CV4, RSW...)

; Cycle call with:
; P -> Define pocket contour
; LTR-> TR of RoughingTool 1
; CV4-> Predrilling mode
; RSW-> Later finishing allowance, wall
; The drilling points are automatically defined as starting points
; Disable modal drilling cycles
; Fit RoughingTool 1
; Select roughing feed
N100 G80
N110 T1 M6 ED1
N120 F...
N130 G281(P Tasche.npg, DT..., UL..., CV1, CD..., RSW..., RSL...)

; Cycle call with:
; P -> Define pocket contour
; DT -> Pocket bottom absolute
; UL -> Upper pocket border absolute
; CV1-> "Roughing" mode
; RSW-> Finishing allowance wall
; RSL-> Finishing allowance bottom
; Fit RoughingTool 2
N140 T2 M6 ED2
N150 G281(P Tasche.npg, DT..., UL..., LTR..., CD..., RSW..., RSL...)

; Cycle call with:
; P -> Define pocket contour
; DT -> Pocket bottom absolute
; UL -> Upper pocket border absolute
; LTR-> "Residual cut" mode, tool radius

```

Milling cycles (contour-dependent)

```

;      RSW-> Finishing allowance wall
;      RSL-> Finishing allowance bottom
N160 T4 M6 ED4
; Fit RoughingTool 4
N170 G281(P Tasche.npg, DT..., UL..., CV3, RSW..., RSL...)
; Cycle call with:
;   P  -> Define pocket contour
;   DT -> Pocket bottom absolute
;   UL -> Upper pocket border absolute
;   CV3-> "Finishing" mode
;   RSW-> Finishing allowance wall
;   RSL-> Finishing allowance bottom
```

9 Engraving

9.1 General information

Basic settings

Letter height	10
Distance	2,222
Subordinate line	2,222
Letter width	See table Character set for engraving cycles

Tab. 9-1: Specification for basic settings in mm

Supported characters of the MTX character set

Label ASCII / ANSI	Charac- ters	Character width (at a character height of 10 mm)
32	SP	5.556
33	!	0
34	"	2.222
35	#	5.556
36	\$	5.778
37	%	10
38	&	5.778
39	'	5.778
40	(	1.111
41	)	1.111
42	*	4.445
43	+	4.445
44	,	1.111
45	-	4.445
46	.	0
47	/	2.222
48	0	4.445
49	1	2.222
50	2	4.445
51	3	4.445
52	4	5.556
53	5	4.445
54	6	4.445
55	7	4.445
56	8	4.445
57	9	4.445

Engraving

Label ASCII / ANSI	Charac- ters	Character width (at a character height of 10 mm)
58	:	0
59	;	1.111
60	<	4.445
61	=	4.445
62	>	4.445
63	?	4.445
64	@	5.778
65	A	6.667
66	B	6.667
67	C	5.556
68	D	6.667
69	E	5.556
70	F	5.556
71	G	6.667
72	H	6.667
73	I	0
74	Y	4.445
75	K	4.445
76	L	4.445
77	M	7.778
78	N	5.556
79	O	5.556
80	P	5.556
81	Q	6.667
82	R	5.556
83	S	5.556
84	T	5.556
85	U	5.556
86	V	6.667
87	W	8.889
88	X	7.888
89	Y	6.667
90	Z	5.556
91	[	1.111
92	\	2.222

Label ASCII / ANSI	Charac- ters	Character width (at a character height of 10 mm)
93	]	1.111
94	^	6.042
95	_	4.445
96	'	1.111
97	a	5.278
98	b	4.445
99	c	3.334
100	d	4.445
101	e	3.334
102	f	3.334
103	g	4.445
104	h	4.445
105	i	0
106	y	1.667
107	k	4.445
108	l	1.111
109	m	6.667
110	n	4.445
111	o	4.445
112	p	4.445
113	q	5.556
114	r	3.334
115	s	4.445
116	t	3.334
117	u	4.445
118	v	4.445
119	w	6.667
120	x	4.445
121	y	4.445
122	z	4.445
123	{	4.445
124		0
125	}	4.445
126	~	5.778

Tab. 9-2: Character set for engraving cycles

Example with special characters

Text to be engraved: **Please refresh directory "C:\tmp\user"**

NC block: N40 G201 (TX **"Please refresh directory \"C:\tmp\user\"**, LE30, PA45, MSP80, SSP20, SL2, DT-1, FI300)



Strings should always be programmed between inverted commas "**.**".

If a string is written between inverted commas and an additional inverted comma is required in this string, the additional inverted comma must be preceded by a backslash "\". Backslashes in turn are represented by two backslashes.

If strings are very simple, no leading or closing blanks, no comma ",", equal-to signs "=" and brackets close ")", the inverted commas may be omitted. In this case, backslashes "\" are interpreted as regular backslashes.

9.2 G201 Linear text engraving

Syntax G201(TX,LE,PA,PD,MSP,SSP,SL,DT,FI,FF,ORI,TYP,DWT,IPS)

Short description

The cycle "G201 Linear text engraving" engraves a text using an engraving tool whose characters are aligned in a linear way.

Overview

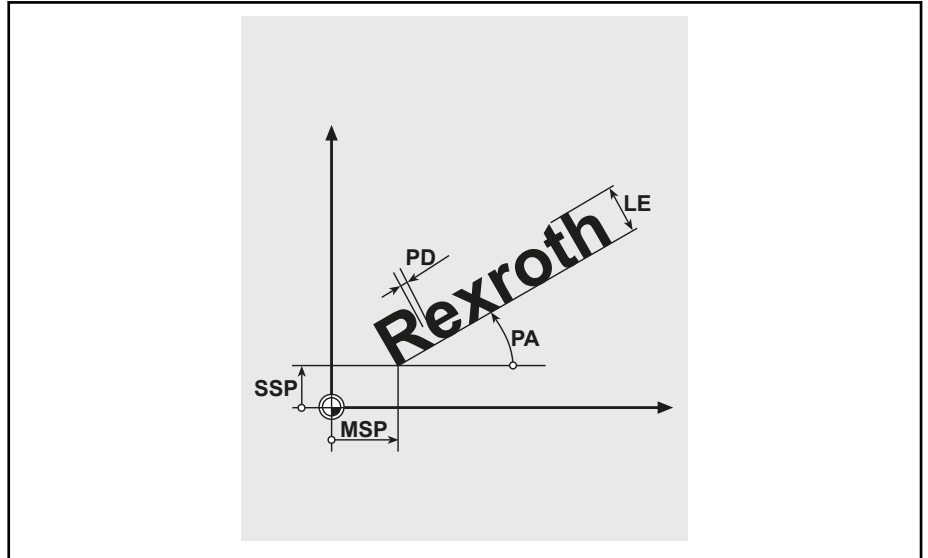


Fig. 9-1: Cycle G201 Linear text engraving

Parameter

Parameter	Name	Properties	Values	Pre-assignment
TX	Text	Mandatory	String, Max. 128 characters	
LE	Text height	Mandatory	>0	
PA	Text angle	optional	Absolute, Degree	0
PD	Character spacing	optional	Incremental, ≥0	0
MSP	Starting point: main axis	Mandatory	Absolute	
SSP	Starting point: secondary axis	Mandatory	Absolute	
SL	Safety level	Mandatory	Absolute	
DT	Depth	Mandatory	Absolute	
FI	Plunge feed	optional	>0	Current feed
FF	Feed plane	optional	>0	Current feed
ORI	Position	optional		0
TYP	Type	optional		0

DWT	Dwell time at points	optional	≥ 0 , Seconds	0
IPS	Exact stop	optional		1

Tab. 9-3: Parameter call

ORI	Effect
0	Horizontal
1	Vertical

ORI 0

Bosch

MTX

ORI 1

B
o
s
c
h

X-MT

Tab. 9-4: Description of parameter ORI - Position

TYPE	Effect
0	Standard print
1	Mirror writing

TYP 0

Bosch

TYP 1

Bosch

Tab. 9-5: Description of parameter TYPE - Type

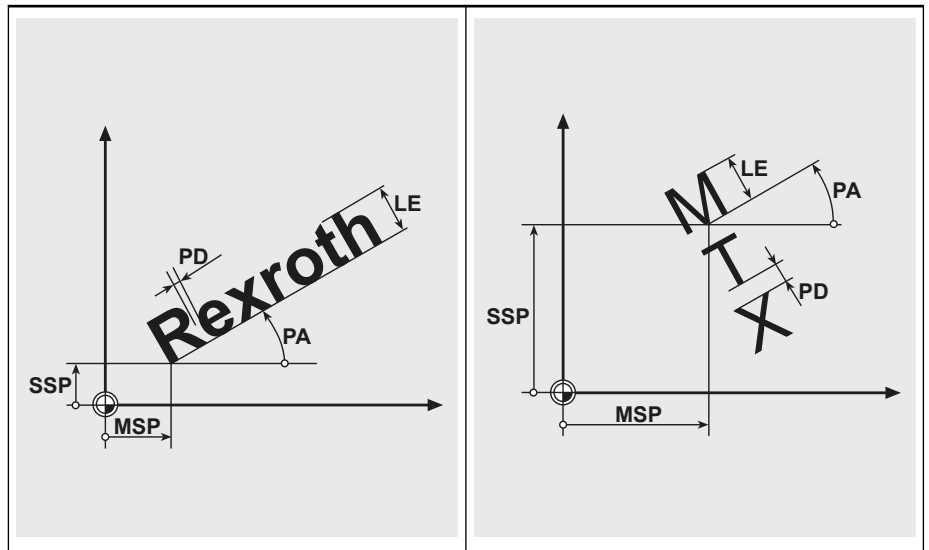
IPS	Effect
0	Exact stop OFF
1	Exact stop ON

Tab. 9-6: Description of parameter IPS - Exact stop

Description

TX text

Contains text to be engraved (one line)



Tab. 9-7: Parameters in horizontal and vertical position

LE text height

Height of capitals, measured from the base line

PA text angle

The text is aligned under this absolute angle to the main axis

PD character spacing

Distance between two adjacent characters of the text in the write direction that is added to the distance defined in the font

MSP Starting point main axis

The first character can be found at this absolute position of the main axis

SSP Starting point secondary axis

The first character can be found at this absolute position of the secondary axis

SL Safety level

Plunging in feed starts at this absolute position on the feed axis. After milling of each polyline, it is returned to this absolute position

DT Depth

The text is engraved on this absolute position of the feed axis

FI Plunge feed

Infeed takes place to depth DT using this feed

FF Feed level

Characters are engraved using this feed

	ORI position	ORI determines whether the characters are aligned horizontally or vertically
	TYPE type	
		Specifies the use of mirror writing
	DWT Dwell time at points	
		Waiting time: time interval at full stop, semicolon, after plunging and before retracting to reduce cutting pressure and to improve the quality of the typeface.
	IPS Exact stop	
		The exact stop controls if the NC has to decelerate to a new polyline during positioning to reach the end position in the exact stop window
Special features		G40 (no edge radius correction) is not to be active prior to the cycle. The cycle deactivates global transition elements (RND, RNE, CHF, and CHS).
		The machining time can be reduced by using exact stop Off (IPS0). Requirement: positioning with lag errors has to be possible (instruction: FeedForward/FFW).
Sequence/processing		Positioning takes place in rapid traverse to the start position of the first polyline in the interpolation plane. Subsequently, it is positioned to the safety level in the feed axis. Subsequently, it is traveled to the engraving depth with the plunging feed, the polyline is engraved and moved back to the safety level. This procedure is repeated until all characters are engraved.
Example		The text engraving depth is 1 mm.

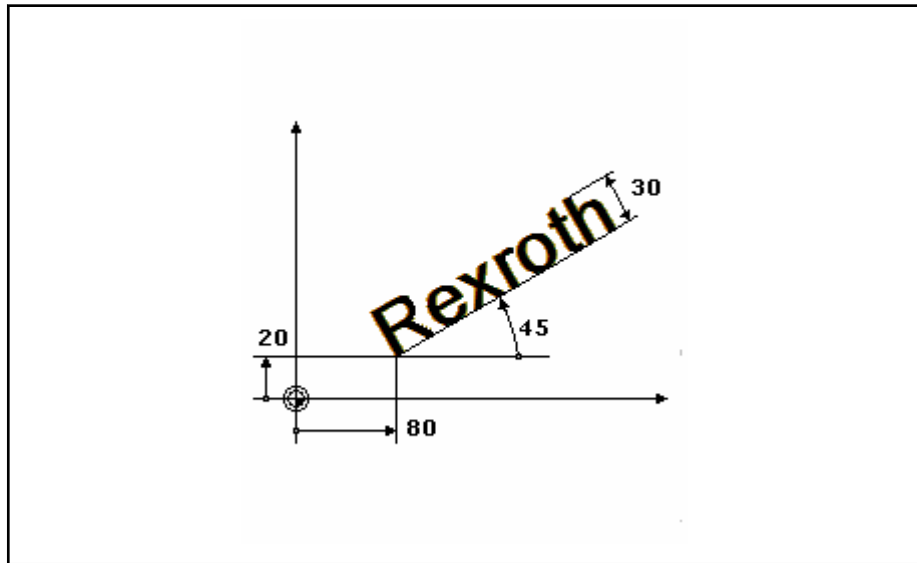


Fig. 9-2: Example: G201 Linear text engraving
Program for G201 Linear text engraving

```

N10 T1 M6 ; Tool change
N20 G94 G97 S3500 F500 M3 ; Technology values
N30 G0 G17 G90 RAD X80 Y20 Z20 ; Prepositioning
N40 G201 (TX Rexroth, LE30, PA45, MSP80, SSP20, SL2, DT-1, FI300)
; Cycle Linear text engraving
N50 X300 Y300 Z200 M5 ; Leave contour

```

9.3 G202 Circular text engraving

Syntax G202(TX,LE,PA,MCP,SCP,R,FA,CDI,SL,DT,FI,FF,TYP,DWT,IPS)
Short description

The cycle "G202 Circular text engraving" engraves a text using an engraving tool whose characters are aligned along a circular arc.

Overview

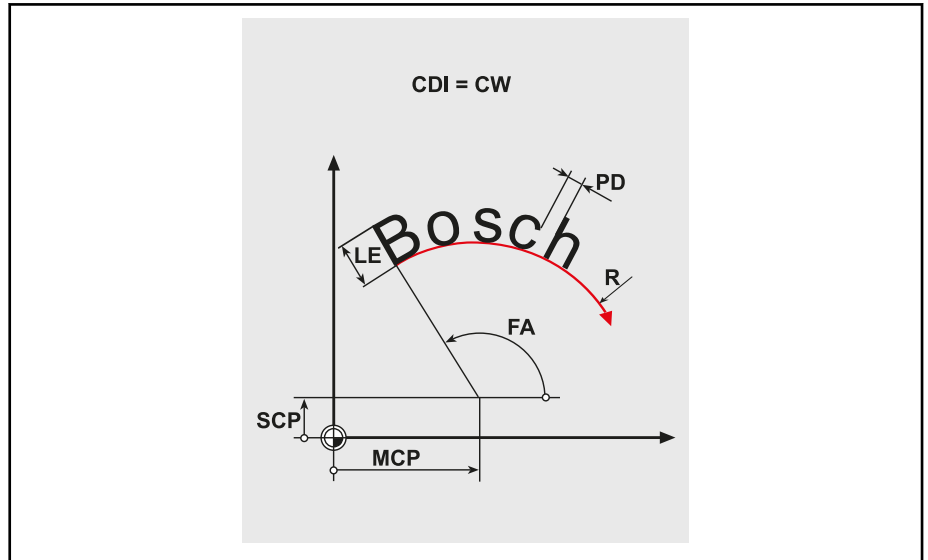


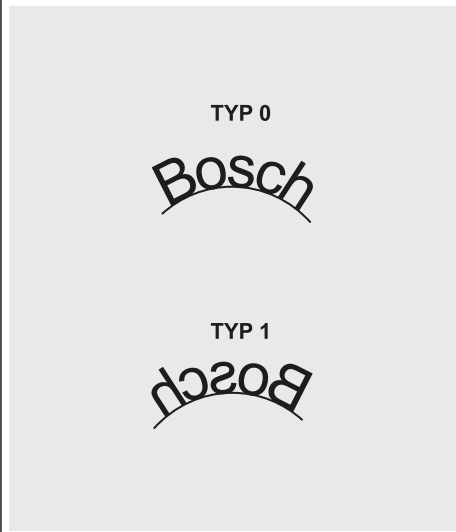
Fig. 9-3: Cycle G202 Circular text engraving

Parameter

Parameter	Name	Properties	Values	Pre-assignment
TX	Text	Mandatory	String, Max. 128 characters	
LE	Text height	Mandatory	>0	
PD	Character spacing	optional	Incremental, >=0	0
MCP	Center point: main axis	Mandatory	Absolute	
SCP	Center point: secondary axis	Mandatory	Absolute	
R	Radius	Mandatory	Incremental, >0	
FA	Starting angle	optional	Absolute	0
CDI	Direction of rotation	optional	Counterclockwise direction (CCW) / Clockwise direction (CW)	CCW
SL	Safety level	Mandatory	Absolute	
DT	Depth	Mandatory	Absolute	

FI	Plunge feed	optional	>0	Current feed
FF	Feed plane	optional	>0	Current feed
TYPE	Type	optional		0
DWT	Dwell time at points	optional	>=0, Seconds	0
IPS	Exact stop	optional		1

Tab. 9-8: Parameter call

TYPE	Effect
0	Standard print
1	Mirror writing
 The image shows two examples of the Bosch logo engraved on a light gray background. The top example, labeled 'TYP 0', shows the logo in its standard orientation. The bottom example, labeled 'TYP 1', shows the logo mirrored horizontally.	

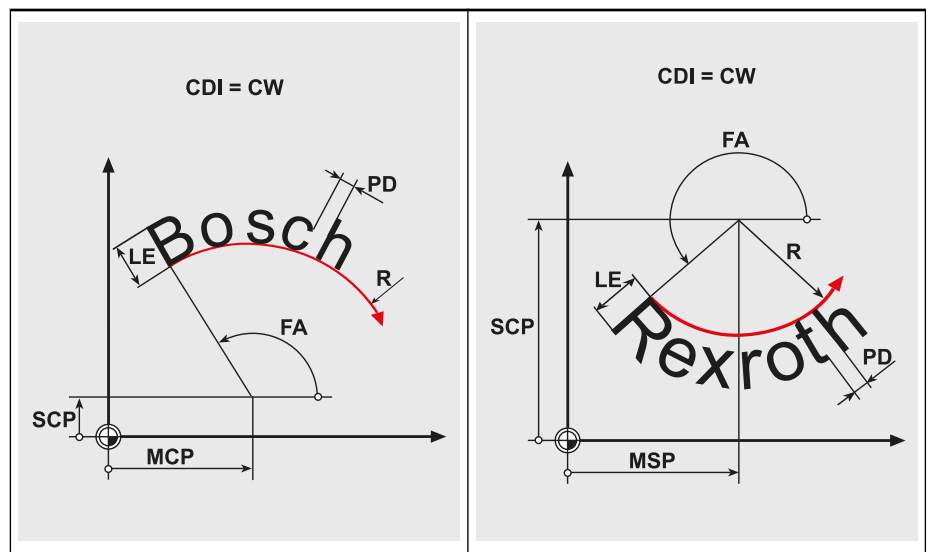
Tab. 9-9: Description of parameter TYPE - Type

IPS	Effect
0	Exact stop OFF
1	Exact stop ON

Tab. 9-10: Description of parameter IPS - Exact stop

Description**TX text**

Contains text to be engraved (one line)



Tab. 9-11: Parameters in clockwise direction and in counterclockwise direction

LE text height

Height of capitals, measured from the base line

PD character spacing

Distance between two adjacent characters of the text in the write direction that is added to the distance defined in the font

MCP Center point main axis

The circular arc center can be found at this absolute position of the main axis

SCP Center point secondary axis

The circular arc center can be found at this absolute position of the secondary axis

R Radius

Radius of the base line in direction of rotation in clockwise direction or radius of head line in direction of rotation in counterclockwise direction

FA Starting angle

The first character of the text starts at this absolute angle to the main axis

CDI Direction of rotation

Direction of rotation of the text

SL Safety level

Plunging in feed starts at this absolute position on the feed axis. After milling of each polyline, it is returned to this absolute position

DT Depth

The text is engraved on this absolute position of the feed axis

FI Plunge feed

Infeed takes place to depth DT using this feed

FF Feed level

Characters are engraved using this feed

TYPE type

Specifies the use of mirror writing

DWT Dwell time at points

Waiting time: time interval at full stop, semicolon, after plunging and before retracting to reduce cutting pressure an to improve the quality of the typeface.

IPS Exact stop

The exact stop controls if the NC has to decelerate to a new polyline during positioning to reach the end position in the exact stop window

Special features G40 (no edge radius correction) is not to be active prior to the cycle. The cycle deactivates global transition elements (RND, RNE, CHF, and CHS).

The machining time can be reduced by using exact stop Off (IPS0). Requirement: positioning with lag errors has to be possible (instruction: FeedForward/FFW).

Sequence/processing Positioning takes place in rapid traverse to the start position of the first polyline in the interpolation plane. Subsequently, it is positioned to the safety level in the feed axis. Subsequently, It is traveled to the engraving depth with the plunging feed, the polyline is engraved and moved back to the safety level. This procedure is repeated until all characters are engraved.

Example The text engraving is too deep by 0.5 mm.

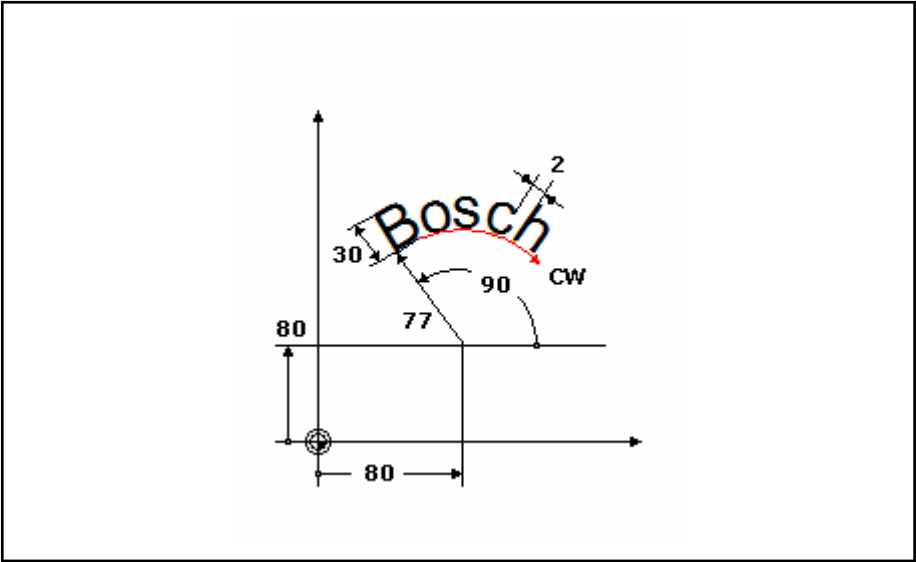


Fig. 9-4: Example: G202 Circular text engraving
Program for G202 Circular text engraving

```
N10 T1 M6 ; Tool change
N20 G94 G97 S3500 F500 M3 ; Technology values
N30 G0 G17 G90 RAD X80 Y160 Z20 ; Prepositioning
N40 G202 (TX Bosch, LE30, PD2, MCP80, SCP80, R77, FA90, CDI CW, SL2, DT-0.5) ; Cycle Circular text engraving
N50 X300 Y300 Z200 M5 ; Leave contour
```

10 Turning cycles

10.1 G160 Longitudinal turning

Syntax G160(SR,SA,ER,EA,CD,CR,CA)

Brief description Use the cycle G160 "Longitudinal turning" to machine a cylindrical plane or an offset by a radial cut segmentation.

Overview

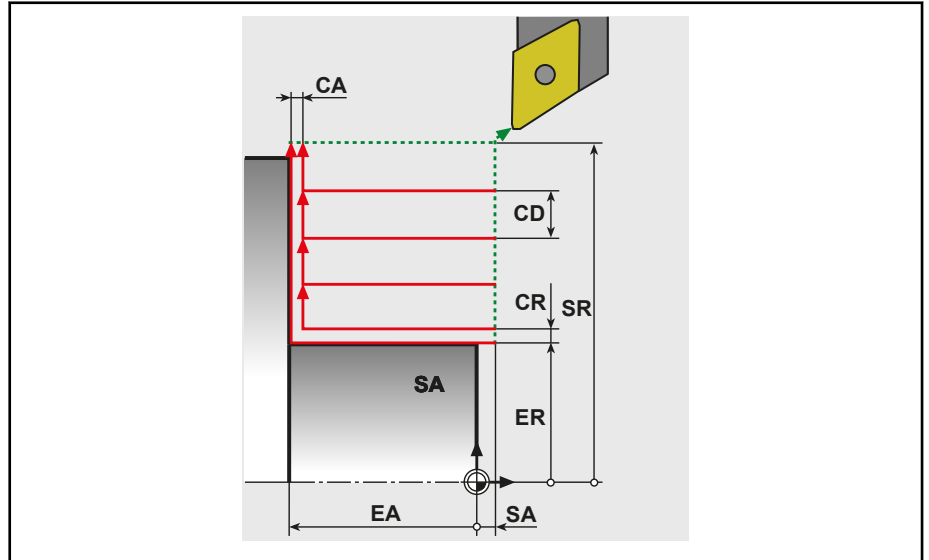


Fig. 10-1: Cycle G160 Longitudinal turning

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
SR	Starting point, radial	Mandatory	Absolute	
SA	Starting point, axial	Mandatory	Absolute	
ER	End point, radial	Mandatory	Absolute	
EA	End point, axial	Mandatory	Absolute	
CD	Maximum depth of cut	Mandatory	Incremental >0	
CR	Depth of last cut, longitudinal	Optional	Incremental ≥0	0 or no finishing cut
CA	Depth of cut of last cut, plane	Optional	Incremental ≥0	0 or no finishing cut

Tab. 10-1: Parameter call

Description

Edge position 4 10 <u>START?</u> = FALSE  Edge position 1 10 <u>START?</u> = FALSE  	Edge position 3 10 <u>START?</u> = FALSE  Edge position 2 10 <u>START?</u> = FALSE 

Tab. 10-2: Parameter for longitudinal turning in four edge positions

SR Starting point, radial

The corner point of the rectangular cross section to be machined and where the first cut starts is located at this absolute radial position.

If required, a longitudinal allowance and an overflow have to be taken into consideration in the parameter value

SA Starting point, axial

The corner point of the rectangular cross section to be machined and where the first cut starts is located at this absolute axial position.

If required, take a safety distance and a plane allowance into consideration

ER End point, radial

This absolute radial position is the end diameter

EA End point, axial

This absolute axial position is the end point of the plane surface

CC Maximum depth of cut

Greatest possible radial infeed

CR Depth of cut of last cut, longitudinal

Radial infeed for the last cut

CA Depth of cut of last cut, plane

Machining allowance for the last plane cut

Special features

The cycle can be used in every interpolation plane, although turning is usually performed in plane G18. The main axis of the plane is the axial axis and the secondary axis is the radial axis. If the cycle is for once called in G19, the z-axis is the axial axis and the y-axis is the radial axis.

Sequence/processing

The cycle begins at the starting point resulting from SR and SA. It has to be possible to travel to the starting point straightforwardly from the last programmed position.

Infeed takes place in rapid traverse for each cut. The total depth is distributed into equally deep cuts by taken into consideration the last depth of cut CR. This cut is executed with the feed programmed in the cycle and consists of an longitudinal section cut and a plane cut to the initial diameter or the diameter of the previous cut. Subsequently, retracting takes place in axial direction to SA. This procedure is repeated cyclically until the finished size or the machining allowance for the last cut is available. If CA>0 or CR>0, a finishing cut is executed. Only in case of CA>0, the total plane surface is rotated.

At the end of the cycle, the tool is moved back to the initial position before the cycle.

Example

Turn a recess. The cutting depth should be 4.5 mm. A cutting depth of 0.3 mm has to be complied with for the last cut. Subsequent turning of the plane

surface should be avoided. A safety distance of 1 mm has to be ensured at the front end

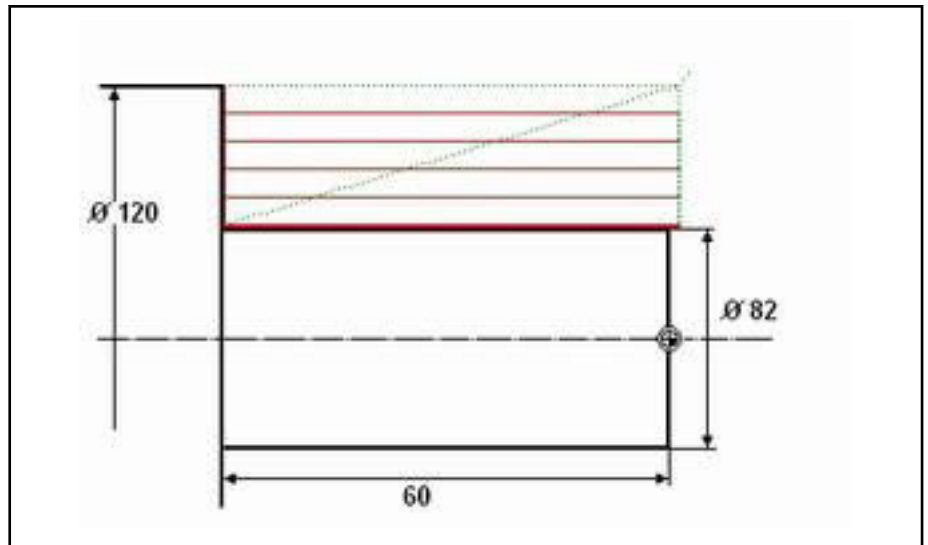


Fig. 10-2: Example G160 Longitudinal turning
Program for G160 Longitudinal turning

```

N10 T1 M6                                ;Tool change
N20 G18 DIA G95 G96 S250 F0.2 M4         ; Technology values
N30 G0 X125 Z3                           ; Approach initial position
N40 G160 (SR120, SA1, ER82, EA-60, CD4.5, CR0.3)
                                           ;Cycle Longitudinal turning
N50 X200 Z100 M5                         ;Leave contour

```

10.2 G161 Face turning

Syntax G161(SR,SA,ER,EA,CD,CR,CA)

Brief description Use the cycle "G160 Longitudinal turning" to machine a plane or an offset by an axial cut segmentation.

Overview

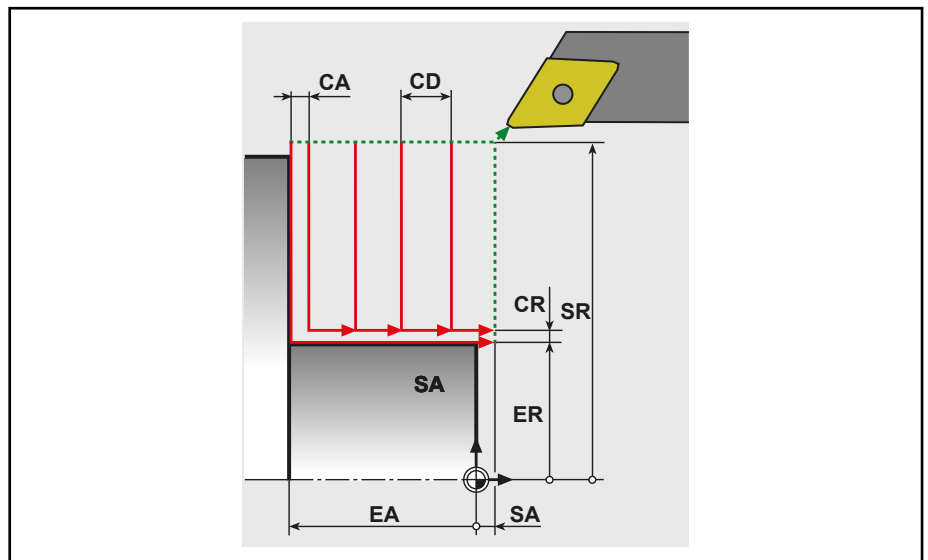


Fig. 10-3: G161 Face turning

Turning cycles

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
SR	Starting point, radial	Mandatory	Absolute	
SA	Starting point, axial	Mandatory	Absolute	
ER	End point, radial	Mandatory	Absolute	
EA	End point, axial	Mandatory	Absolute	
CD	Maximum depth of cut	Mandatory	Incremental >0	
CR	Depth of last cut, longitudinal	Optional	Incremental >=0	0 or no finishing cut
CA	Depth of cut of last cut, plane	Optional	Incremental >=0	0 or no finishing cut

Tab. 10-3: Parameter call

Description

Edge position 4 10 <u>START?</u> = FALSE BOOLEAN variable	Edge position 3 10 <u>START?</u> = FALSE BOOLEAN variable
Edge position 1 10 <u>START?</u> = FALSE BOOLEAN variable	Edge position 2 10 <u>START?</u> = FALSE BOOLEAN variable

Tab. 10-4: Parameter for face turning in four edge positions

SR Starting point, radial

The corner point of the rectangular cross section to be machined and where the first cut starts is located at this absolute radial position.

If required, take a safety distance and a longitudinal allowance into consideration

SA Starting point, axial

The corner point of the rectangular cross section to be machined and where the first cut starts is located at this absolute axial position.

If required, a plane allowance and an overflow have to be taken into consideration in the parameter value

ER End point, radial

This absolute radial position represents the end of machining or the position of the generated cylinder surface

EA End point, axial

This absolute axial position represents the end of machining or the position of the generated plane surface

CC Maximum depth of cut

Greatest possible axial infeed

CR Depth of cut of last cut, longitudinal

Machining allowance for the last longitudinal cut

CA Depth of cut of last cut, plane

Axial infeed for the last plane cut

Special features

The cycle can be used in every interpolation plane, although turning is usually performed in plane G18. The main axis of the plane is the axial axis and the secondary axis is the radial axis. If the cycle is for once called in G19, the z-axis is the axial axis and the y-axis is the radial axis.

Sequence/processing

The cycle begins at the starting point resulting from SR and SA. It has to be possible to travel to the starting point straightforwardly from the last programmed position.

Infeed takes place axially in rapid traverse for each cut. The total depth is distributed into equally deep cuts by taken into consideration the last depth of cut CA. This cut is executed with the feed programmed in the cycle and consists of an plane cut and a longitudinal cut to the initial plane surface or the plane surface of the previous cut. Subsequently, retracting takes place in axial direction to SR. This procedure is repeated cyclically until the finished size or the machining allowance for the last cut is available. If CA>0 or CR>0, a finishing cut is executed. Only in case of CA>0, the total cylindrical surface is rotated in finished dimension.

At the end of the cycle, the tool is moved back to the initial position before the cycle.

Example

Turn a spigot. The cutting depth should be 4.5 mm. A cutting depth of 0.3 mm has to be complied with for the last cut (longitudinal and plane). Subsequent turning of the plane surface should be avoided. An allowance of 1 mm has to be ensured at the front end. The safety distance to the external diameter is 1 mm.

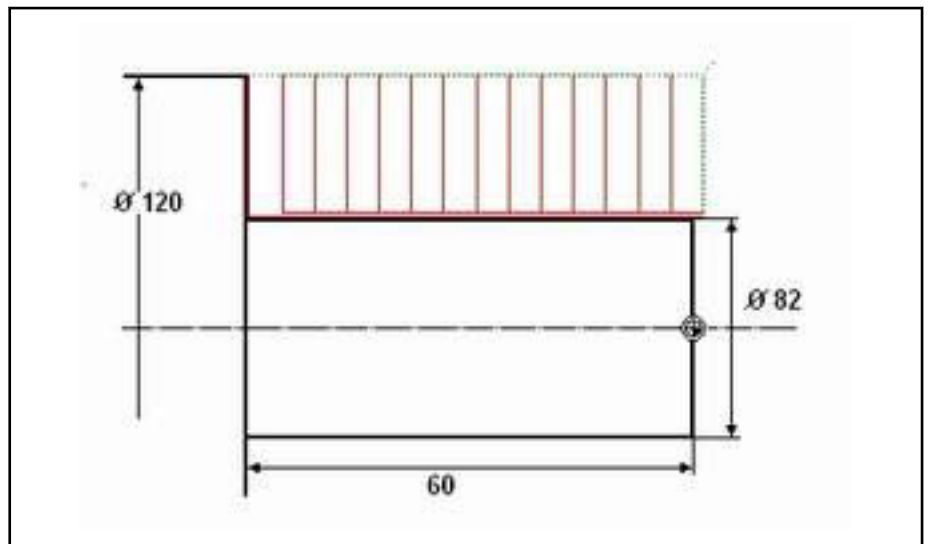


Fig. 10-4: Example G161 Face turning
Program for G161 Face turning

```

N10 T1 M6                ;Tool change
N20 G18 DIA G95 G96 S250 F0.2 M4 ; Technology values
N30 G0 X125 Z3            ; Prepositioning
N40 G161 (SR120, SA1, ER82, EA-60, CD4.5, CR0.3)
                          ;Cycle Face turning
N50 X200 Z100 M5         ;Leave contour
  
```

10.3 G162 Tapered turning

Syntax G162(SR,SA,ER,EA,CD,CC,RSF,RSL)

Brief description A cone is generated by contour-parallel cuts using the "G162 Tapered turning" cycle.

Overview

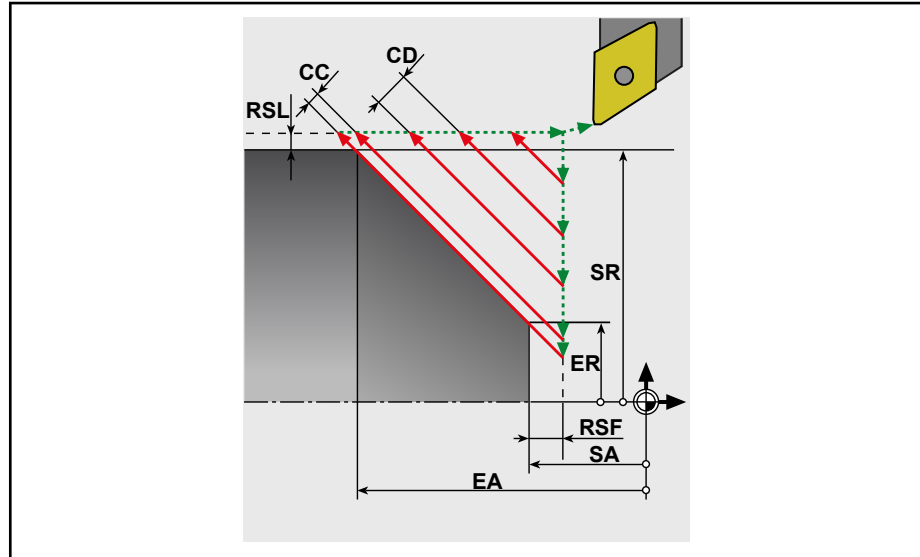


Fig. 10-5: G162 Tapered turning cycle

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
SR	Starting point, radial	Mandatory	Absolute	
SA	Starting point, axial	Mandatory	Absolute	
ER	End point, radial	Mandatory	Absolute	
EA	End point, axial	Mandatory	Absolute	
CD	Maximum depth of cut	Mandatory	Incremental >0	
CC	Depth of cut of last cut	Optional	Incremental ≥0	CD
RSF	Approach/allowance, plane	Optional	Incremental ≥0	0
RSL	Runout/allowance, longitudinal	Optional	Incremental ≥0	0

Tab. 10-5: Parameter call

Description

Edge position 4 10 <u>START?</u> = FALSE BOOLEAN variable	Edge position 3 10 <u>START?</u> = FALSE BOOLEAN variable
Edge position 1 10 <u>START?</u> = FALSE BOOLEAN variable	Edge position 2 10 <u>START?</u> = FALSE BOOLEAN variable

Tab. 10-6: Parameter for tapered turning in four edge positions

SR Starting point, radial

At this absolute radial position, the corner point of the rectangular triangle, generated by the cross section to be to be cut machined and where the first cut starts, is located at

SA Starting point, axial

At this absolute axial position, the corner point of the rectangular triangle, generated by the cross section to be to be cut machined and where the first cut starts, is located at

ER End point, radial

The first reference point of the taper curve is located at this absolute radial position

EA End point, axial

The second reference point of the tapered curve is located at this absolute axial position

CD Maximum depth of cut

Greatest possible contour-parallel infeed

CC Depth of cut of last cut

Contour-parallel infeed for the last cut

RSF Approach/allowance, plane

Axial increment to offset the actual machining start against SA to accommodate an approach, a plane allowance and/or a safety distance

RSL Run-out/allowance, longitudinal

Axial increment to offset the actual machining end against SR to accommodate a run-out and/or a vertical allowance.

Special features

The cycle can be used in every interpolation plane, although turning is usually performed in plane G18. The main axis of the plane is the axial axis and the secondary axis is the radial axis. If the cycle is for once called in G19, the z-axis is the axial axis and the y-axis is the radial axis.

Sequence/processing

The cycle begins at the starting point, resulting from SR, SA and RSF. It has to be possible to travel to the starting point straightforwardly from the last programmed position.

Infeed takes place in rapid traverse for each cut. The total depth is distributed into equally deep cuts by taken into consideration the last depth of cut CC. The cut is always executed with edge radius compensation and the programmed feed. Subsequently, retracting takes place in axial direction to the taper start. This process is repeated cyclically until the finished dimension is completed. Optionally, the cutting depth CC is realized for finishing during the last cut.

At the end of the cycle, the tool is moved back to the initial position before the cycle.

Example: Turn a spigot. The cutting depth for roughing is 5 mm, and 0.5 mm for a finishing cut. An axial approach of 1 mm has to be taken into consideration.

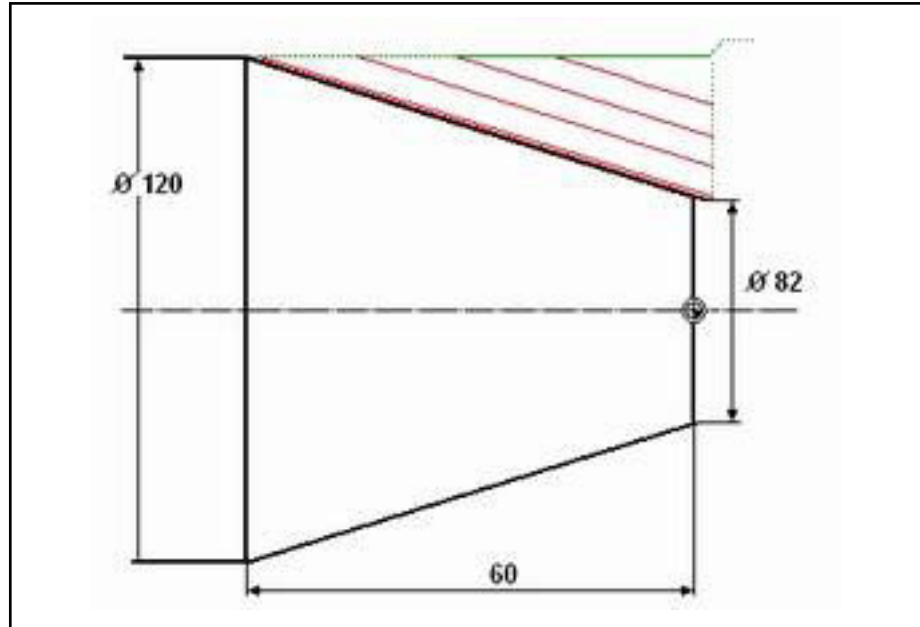


Fig. 10-6: Example of tapered turning

Program for G162 Tapered turning

```

N10 T1 M6                ; Tool change
N20 G18 DIA G95 G96 S250 F0.2 M4 ; Technology values N40 G162 (SR120, SA0, ER82, EA-60, CD5, CC0.5, RSF1)
N30 G0 X125 Z3           ; Prepositioning
N40 G162 (SR120, SA0, ER82, EA-60, CD5, CC0.5, RSF1)
                          ;Cycle Tapered turning
N50 X200 Z100 M5         ;Leave contour
  
```

10.4 G165 Thread single step

Syntax G165(Z,XR,TP,R,ROF)

Brief description Using the modal cycle "G165 Thread single step", cylindrical threads and tapered threads with a taper angle of up to 90° can be turned. Each cut has to be programmed individually in an NC block.

Overview

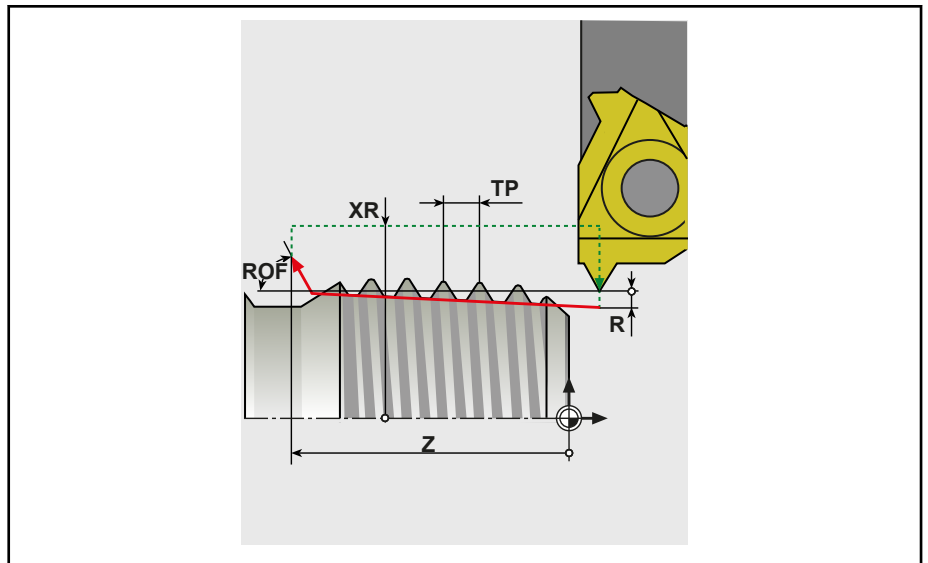


Fig. 10-7: G165 Thread single step

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
Z	End point, radial	Mandatory	Absolute	
XR	Retracting position, radial	Mandatory	Absolute	
TP	Thread pitch	Mandatory	>0	
R	Taper dimension	Optional	Incremental >=0	0
ROF	Retracting angle	Optional	0..70 degrees 0 indicates no slope	60

Tab. 10-7: Parameter call

Description

10 <u>START?</u> = FALSE BOOLEAN variable	10 <u>START?</u> = FALSE BOOLEAN variable

Tab. 10-8: Parameter for single step in external and internal thread

Z End point, axial

The thread ends at this absolute position.

XR Retracting position, radial

At this absolute position, retracting takes place after the thread cut
Include a sufficient safety distance in the available diameter

TP Thread pitch

Pitch of thread (axis-parallel)

R Taper dimension

Increment of the radius change for a tapered thread

ROF Retracting angle

Turning cycles

- If $ROF > 0$, a slope for radial retracting is inserted in the thread run-out. Thus, threads without undercut are machined
- Special features** Note that a modal cycle is not executed with the cycle call during programming, but only after the next traversing command.
G165 can only be used in the interpolation plane G18.
G165 cannot be parameterized via system data.
- Sequence/processing** The last programmed axial position determines the approach of the thread. If the cycle call is integrated in a motion block, only write the X-coordinate of the end point of the thread in the motion block. A Z-coordinate results in an unintentional motion before the thread cut. Otherwise, the first cut and all subsequent cuts can only be programmed by specifying X in a separate block. As all NC statements are executed before a subroutine jump, the programmed end coordinate in X is reached before the cycle.
- If a tapered thread is to be machined (taper dimension $R > 0$), positioning takes place radially to the starting point of the thread opposite the initial position. Subsequently, a thread cut is machined which ends at the programmed X-coordinate and the Z-coordinate provided as parameter. If a retraction angle ($ROF > 0$) has been programmed, the end point of the thread is not reached but a slope is inserted for radial retracting by retaining the axis coupling. After the cut, it is traveled to the retraction position XR in rapid traverse and subsequently back to the axis initial position.
- Example** A tapered fine thread that is to end before the adjacent cylinder is to be turned in 4 cuts. If no thread undercut is to be machined, inclined retracting is required. The approach should be 2 mm. The individual cuts should end on the following X-coordinates:
- 18.6 (0.7 mm infeed)
 - 18.0 (0.3 mm infeed)
 - 17.7 (0.15 mm infeed)
 - 17.546 (approx 0.08 mm infeed corresponds to the kernel diameter)

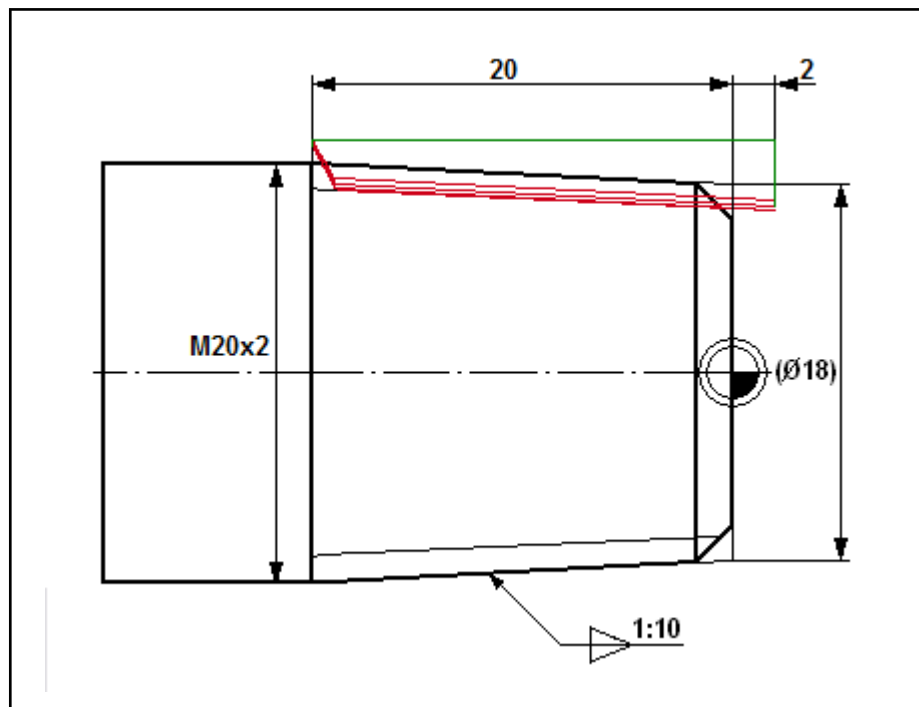


Fig. 10-8: G165 Thread single step

Program for G165 Face thread

```

N10 T1 M6 ; Tool change
N20 G18 DIA G97 S500 M3 ; Technology values
N30 G0 X22 Z2 ; Prepositioning, in Z to
; Start of approach
N40 X18.6 G165 (Z-20, XR22, TP2, R1.1) ; modal cycle call, 1. cut
N50 X18.0 ; 2. cut
N60 X17.7 ; 3. cut
N70 X17.546 ; 4. cut (kernel diameter)
N80 G80 X100 Z100 M5 ; Leave contour, delete modal call

```

10.5 G166 Thread sequence

Syntax G166(SR,SA,ER,EA,AR,AA,RR,RA,ROF,TP,DT,FD,NI,CDM,IT,CT,NFR,TFA,TSA,NT,MTR,TPA,TPR)

Brief description The cycle "G166 Thread sequence" can machine conical threads which can optionally be completed with a thread approach and/or thread run-out. This results in a line of up to three successive threads. The middle thread is named main thread.

Thread sequences are primarily used if the approach and run-out required for machining cannot directly be connected tangentially to the thread (location, no thread undercut, etc.). The approach and run-out can also be contours to be made on the finished part.

Overview

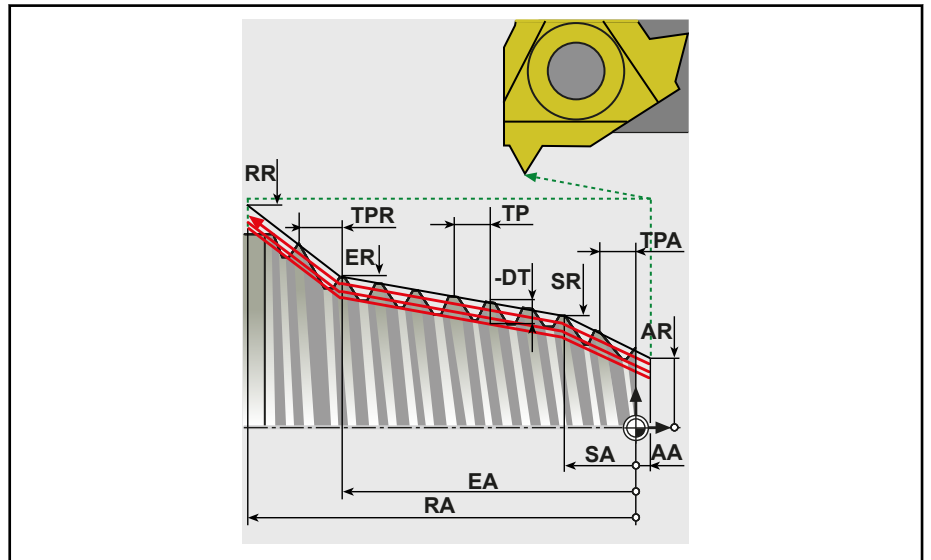


Fig. 10-9: G166 Thread sequence

Parameter

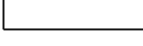
Parameter	Designation	Properties	Values	Pre-assignment
SR	Starting point, main thread radial	Mandatory	Absolute	
SA	Starting point, main thread axial	Mandatory	Absolute	
ER	End point main thread, radial	Mandatory	Absolute	
EA	End point main thread, axial	Mandatory	Absolute	

Turning cycles

Parameter	Designation	Properties	Values	Pre-assignment
AR	Starting point: approach radial	Optional Specify together with AA	Absolute	No separate approach
AA	Starting point, approach axial	Optional Specify together with AR	Absolute	No separate approach
RR	End point, run-out radial	Optional Specify together with RA	Absolute	No separate run-out path
RA	End point, run-out axial	Optional Specify together with RR	Absolute	No separate run-out path
ROF	Retracting angle	Optional Inactive when specifying RR and RA	0..70 degrees 0 indicates no slope	0
TP	Thread pitch	Mandatory	>0	
DT	Thread depth	Mandatory	Incremental < 0 external thread > 0 internal thread	
FD	Initial infeed depth	Optional optionally specify for NI	Incremental >0	Results from NI
NI	Number of cuts	Optional Optionally specify for FD	>0	Internal algorithm (empirical values)
CDM	Minimum infeed depth	Optional	>0	0.05 mm
IT	Infeed variant	Optional		2
CT	Cutting variant	Optional		4
NFR	Number of glazed cuts	Optional	>=0	0
TFA	Edge angle	Optional	Degree 0<=TFA<180	60
TSA	Starting angle	Optional	Degree	
NT	Number of turns	Optional	>0	1

Parameter	Designation	Properties	Values	Pre-assignment
MTR	Machining sequence of several turns of the thread	Optional only active in case NT>1	simultaneously (1) / subsequently (2)	1
TPA	Thread approach pitch	Optional	>0	TP
TPR	Pitch thread run-out	Optional	>0	TP

Tab. 10-9: Parameter call

IT	Effect
1	Degressive infeed strategy by complying with a constant chip cross section
2	Degressive infeed from the third cut with increasing chip cross section
3	Constant infeed around FD, recommended for tetragonal thread
10 <u>START?</u> = FALSE  BOOLEAN variable	

Tab. 10-10: Description of the parameter DEG - infeed variant

CT	Effect
1	Tool guidance, axis-parallel
2	Tool guidance along an edge
3	Tool guidance, alternating between both edges Not permitted for tapered thread
4	Tool guidance, alternating
10 <u>START?</u> = FALSE  BOOLEAN variable	

Tab. 10-11: Description of the parameter CT - cut variant

Description

10 <u>START?</u> = FALSE  BOOLEAN variable	10 <u>START?</u> = FALSE  BOOLEAN variable

Tab. 10-12: Dimensions of the thread sequence for external and internal thread

SR Starting point, main thread radial

The main thread starts at this absolute, radial position

SA Starting point, main thread axial

The main thread starts at this absolute, axial position

ER End point main thread, radial

The main thread ends at this absolute, radial position

EA End point main thread, axial

The main thread ends at this absolute, axial position

AR Starting point approach radial

The thread approach starts at this absolute, radial position

AA Starting point, approach axial

The thread approach starts at this absolute, axial position

RR End point, run-out radial

The thread run-out ends at this absolute, radial position

End point, run-out axial

The thread run-out ends at this absolute, axial position

ROF Retracting angle

If $ROF > 0$ and only if no run-out distance (RR, RA) is programmed, a slope for radial retracting is inserted at the end of the main thread. 0° points in axial direction

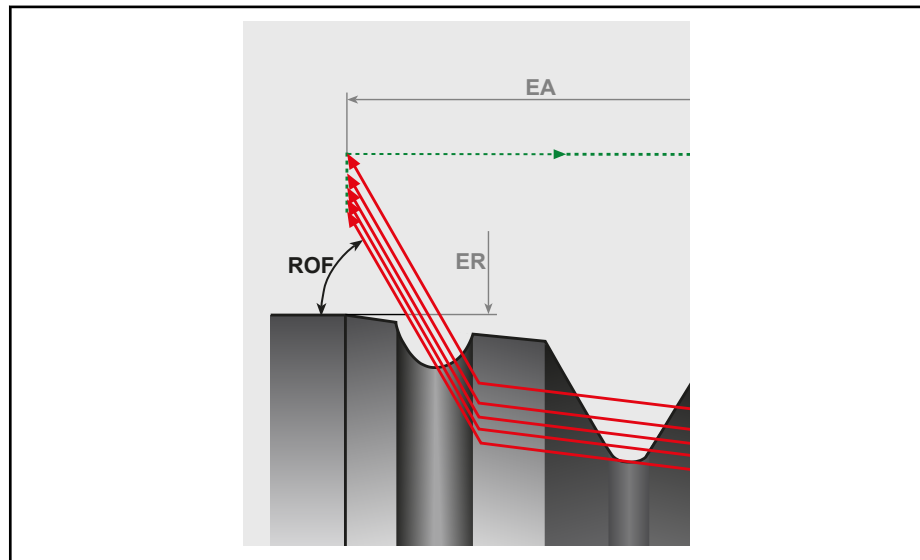


Fig. 10-10: Dimension of the retracting angle ROF at a taper angle of the main thread of $\leq 90^\circ$.

TP Thread pitch

Pitch of main thread

DT Thread depth

The sign determines the direction in which the depth infeed is made with each cut. This means that DT has to be < 0 for external threads and in the x-negative range < 0 for internal threads.

FD First infeed depth

Depth of first cut

NI Number of cuts

Defines the number of cuts without glazed cuts

CDM Minimum infeed depth

Minimum infeed depth that is not fallen below

FD or NI can be adjusted in the cycle to comply with the minimum infeed depth

IT Infeed variant

Determines the infeed depth from the second cut

CT Cutting variant

Specifies the position of the tool travels relatively to the completed thread profile.

NFR Number of glazed cuts

Number of cuts after the thread depth has been reached without infeed/chip removal.

TFA Edge angle

Opening angle between two edges of the thread profile. The bisector runs parallel to the axis

For cutting variant CT2, it is recommended to reduce the programmed edge angle (compared to the actual angle), to avoid friction at one side of the tool edge and the workpiece surface (modified edge infeed)

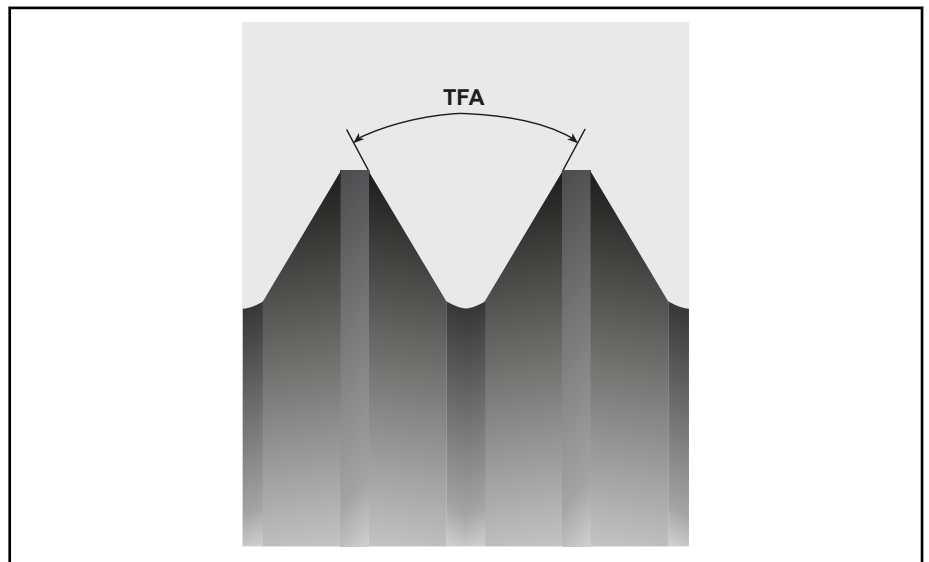


Fig. 10-11: Edge angle TFA

TSA Starting angle

Is to be programmed if the thread start has to pass through a specific reference point at the workpiece. The starting angle specified by TSA is implemented at the starting point AR/AA or SR/SA. The starting angle of the thread to be programmed is calculated from the distance of the starting point of the reference point on the workpiece, the thread pitch and the direction of rotation.

NT Number of turns

Defines the number of thread turns (for multiple threads)

MTR Processing sequence of several turns of the thread

Influences whether all turns of a thread are machined simultaneously or whether one turn of a thread is to be completed before the next one is machined.

TPA Thread approach pitch

Special pitch for thread approach, measured in the same axis direction as TP

TPR Thread approach pitch

Special pitch for thread run-out, measured in the same axis direction as TP

Turning cycles

Special features The cycle can be used in every interpolation plane, although turning is usually performed in plane G18. The main axis of the plane is the axial axis and the secondary axis is the radial axis. If the cycle is for once called in G19, the z-axis is the axial axis and the y-axis is the radial axis.

The thread pitches TP, TPA and TPR are measured in parallel to the axis. They are applied in case of a longitudinal pitch at a taper angle of the main thread of $\leq 90^\circ$. The tool is set radially. The pitch is considered to be a face pitch from a taper angle greater than 90 degrees. The tool is set axially.

Sequence/processing The cycle positions the tool at a starting point resulting from the thread run. The approach coordinate is AR/AA or the thread starting coordinate SA or SR in the direction of the pitch. In the infeed direction, it is the maximum (at $DT < 0$) or minimum of all the thread coordinates, which is additionally displaced by a retract amount of 0,5 mm towards the infeed direction. It has to be possible to approach this point directly and without collision starting from the position reached prior to the cycle. Then, the tool travels to the initial cut and to the initial infeed depth on the workpiece. A thread cut is performed, which comprises of an approach, the main thread and the run-out. If no run-out path is provided and if a retraction angle $ROF > 0$ has been programmed, the end point of the main thread is not reached but a slope is inserted for radial retracting by retaining the axis coupling. The tool is retracted in rapid traverse (whereby the retract coordinate corresponds to those of the starting point) and travels axis-parallelly to the infeed of the next cut. The individual thread cuts are performed cyclically until the end depth is reached. Glazed cuts can be optionally performed. The cycle ends with the tool traveling to the starting point of the cycle and then by returning to its position at the starting point before the cycle call.

Example Due to lack of space for a tangential approach and the lack of a thread undercut at the end, the thread is to be machined with the cycle for thread sequences. Execute seven cuts with constant chip cross section.

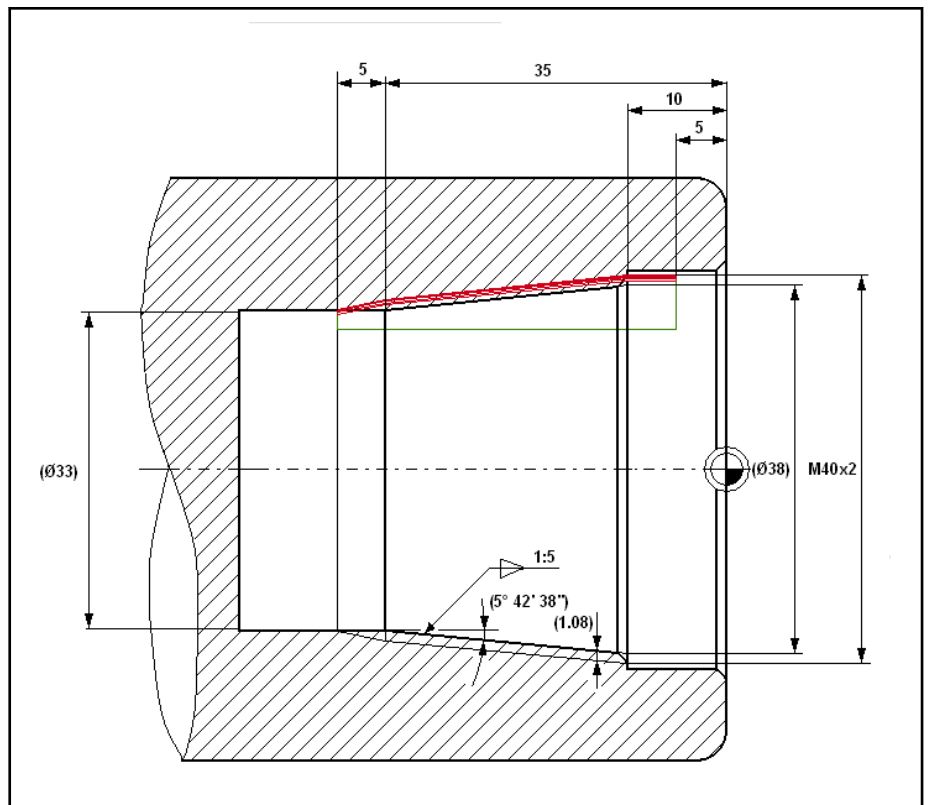


Fig. 10-12: Example G166 Thread sequence

Program for G166 Thread sequence

```

N10 T5 M6 ;Tool change
N20 DIA G18 G97 S500 M3 ;Technology values
N30 G0 X30 Z10 ;Prepositioning
N40 G166 (SR37.84, SA-10, ER32.84, EA-35,
        AR37.84, AA-5, RR30.84, RA-40,
        TP2, DT1.08, NI7, IT1) ;Cycle Thread chain
N50 X200 Z100 M5 ;Leave contour

```

10.6 G167 Thread turning

Syntax G167(SR,SA,EA,TP,DT,FD,NI,CDM,IT,CT,ROF,NFR,TFA,TSA,NT,MTR)

Brief description Cylindrical threads are machined using the cycle "G167 Thread turning"

Turning cycles

Overview

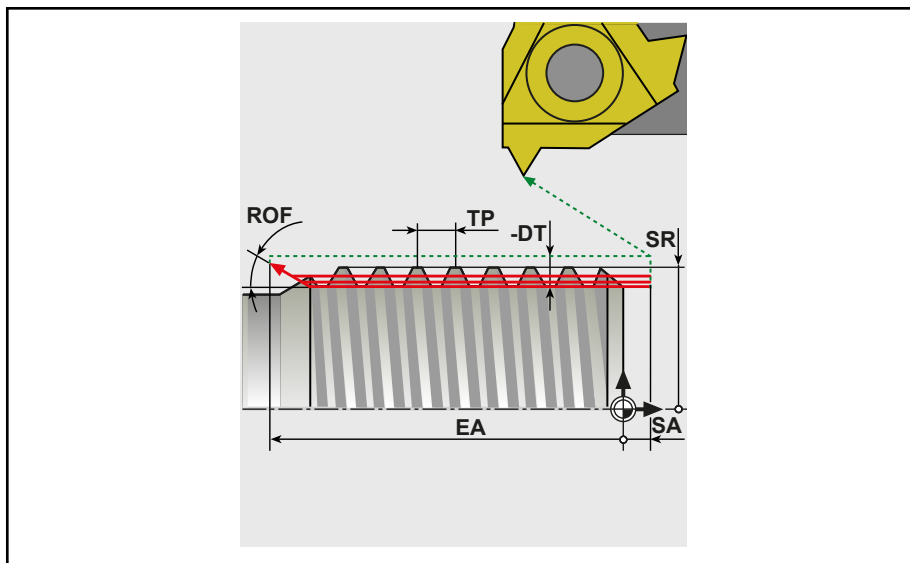


Fig. 10-13: G167 Thread turning

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
SR	Starting point, radial	Mandatory	Absolute	
SA	Starting point, axial	Mandatory	Absolute	
ER	End point, radial	Mandatory	Absolute	
TP	Thread pitch	Mandatory	>0	
DT	Thread depth	Mandatory	Incremental < 0 external thread > 0 internal thread	
FD	Initial infeed depth	Optional optionally specify for NI	Incremental >0	Results from NI
NI	Number of cuts	Optional Optionally specify for FD	>0	Internal algorithm (empirical values)
CDM	Minimum infeed depth	Optional	>0	0.05 mm
IT	Infeed variant	Optional		2
CT	Cutting variant	Optional		4
ROF	Retracting angle	Optional	0..70 degrees 0 indicates no slope	0

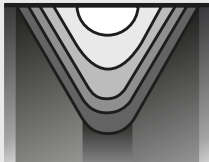
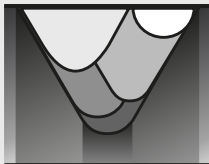
Parameter	Designation	Properties	Values	Pre-assignment
NFR	Number of glazed cuts	Optional	≥ 0	0
TFA	Edge angle	Optional	Degree $0 \leq TFA < 180$	60
TSA	Starting angle	Optional	Degree	
NT	Number of turns	Optional	> 0	1
MTR	Machining sequence of several turns of the thread	Optional only active in case $NT > 1$	simultaneously (1) / subsequently (2)	1

Tab. 10-13: Parameter call

IT	Effect
1	Degressive infeed strategy by complying with a constant chip cross section
2	Degressive infeed from the third cut with increasing chip cross section
3	Constant infeed around FD, recommended for tetragonal thread
<pre>10 START? = FALSE</pre>  BOOLEAN variable	

Tab. 10-14: Description of the parameter DEG - infeed variant

CT	Effect
1	Tool guidance, axis-parallel (radial infeed)
2	Tool guidance along an edge (edge infeed)
3	Tool guidance, alternating between both edges

CT	Effect
4	Tool guidance, alternating (radial/edge infeed)
CT1  CT2  CT3  CT4 	

Tab. 10-15: Description of the parameter CT - cut variant

Description

10 START? = FALSE BOOLEAN variable	10 START? = FALSE BOOLEAN variable

Tab. 10-16: Dimensions of the cylindrical internal and external thread

SR Starting point, radial

The thread starts at this absolute, radial position

SA Starting point, axial

The thread starts at this absolute, axial position

EA End point, axial

The main thread ends at this absolute, axial position

TP Thread pitch

Pitch of thread

DT Thread depth

The sign determines the direction in which the depth infeed is made with each cut. This means that DT has to be < 0 for external threads and in the x-negative range < 0 for internal threads.

FD First infeed depth

Depth of first cut

NI Number of cuts

Defines the number of cuts without glazed cuts

CDM Minimum infeed depth

Minimum infeed depth that is not fallen below

FD or NI can be adjusted in the cycle to comply with the minimum infeed depth

IT Infeed variant

Determines the infeed depth from the second cut

CT Cutting variant

Specifies the position of the tool travels relatively to the completed thread profile.

ROF Retracting angle

If $ROF > 0$, a slope for radial retracting is inserted at the end of the thread

NFR Number of glazed cuts

Number of cuts after the thread depth has been reached without infeed/chip removal.

TFA Edge angle

Opening angle between two edges of the thread profile

For cutting variant CT2, it is recommended to reduce the programmed edge angle (compared to the actual angle), to avoid friction at one side of the tool edge and the workpiece surface (modified edge infeed)

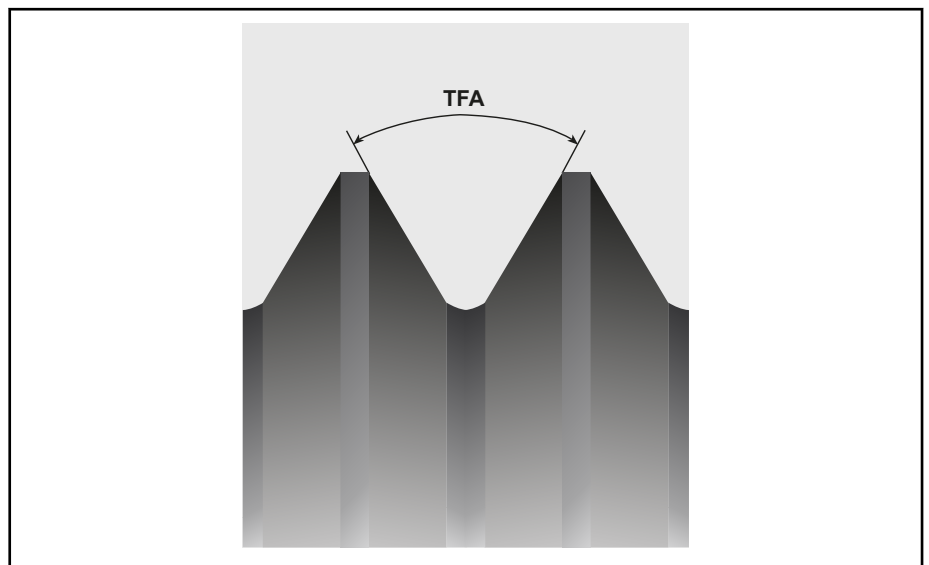


Fig. 10-14: Edge angle TFA

TSA Starting angle

Is to be programmed if the thread start has to pass through a specific reference point at the workpiece. The starting angle specified by TSA is executed at the starting coordinate SR/SA. The starting angle of the thread to be programmed is calculated from the distance of the starting point of the reference point on the workpiece, the thread pitch and the direction of rotation.

NT Number of turns

Defines the number of thread turns (for multiple threads)

MTR Processing sequence of several turns of the thread

Influences whether all turns of a thread are machined simultaneously or whether one turn of a thread is to be completed before the next one is machined.

Special features

The cycle can be used in every interpolation plane, although turning is usually performed in plane G18. The main axis of the plane is the axial axis and the secondary axis is the radial axis. If the cycle is for once called in G19, the z-axis is the axial axis and the y-axis is the radial axis.

Sequence/processing

The cycle positions the tool at the starting point SR and SA. The target coordinates of the motion are displaced by the retract amount of 0.5 mm in radial direction. It has to be possible to approach this point directly and without colli-

sion starting from the position reached prior to the cycle. Then, the tool travels to the initial cut and to the initial infeed depth on the workpiece. A thread cut is performed. If the retraction angle $ROF > 0$ has been programmed, the end point of the thread is not reached but a slope is inserted for radial retracting by retaining the axis coupling. The tool is retracted and travels to the infeed of the next cut. The individual thread cuts are performed cyclically until the end depth is reached. Glazed cuts can be optionally performed. The cycle ends with the tool traveling to the starting point of the cycle and then by returning to its position at the starting point before the cycle call.

Example The internal thread is to be turned with an overhead tool with edge infeed. This means that the tool operates in the x-negative area and the thread depth DT is to be specified as negative. The thread approach is 5 mm.

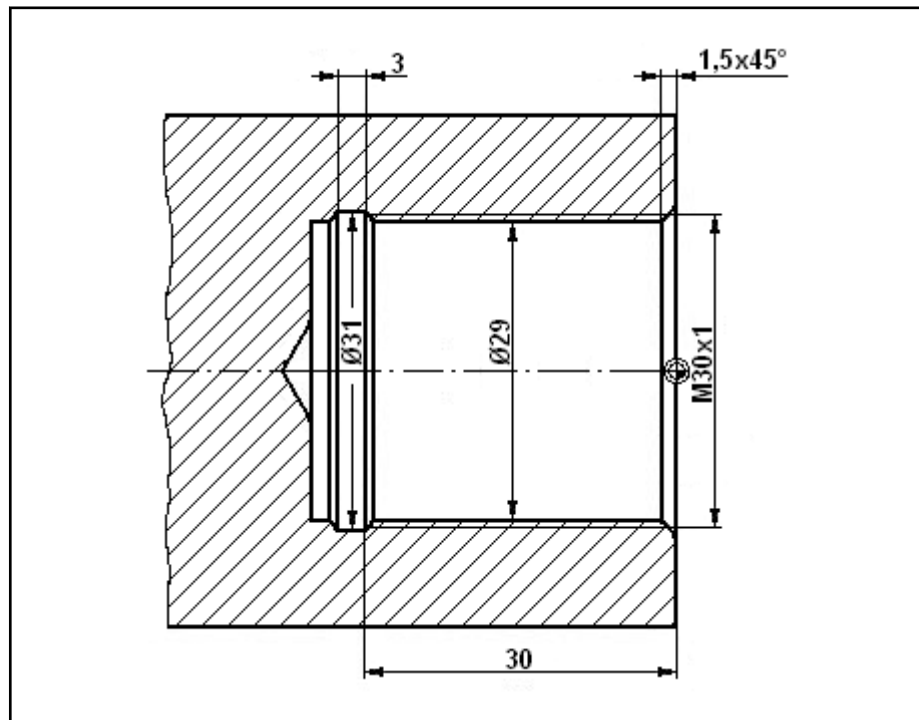


Fig. 10-15: Example G167 Thread turning

Program for G167 Thread turning

N10 T5 M6	;Tool change
N20 DIA G18 G97 S500 M3	;Technology values
N30 G0 X-28 Z10	;Prepositioning
N40 G167 (SR-28.917, SA5, EA-31.5, TP1, DT-0.541, CT2)	;Cycle Thread turning
N50 X200 Z100 M5	;Leave contour

10.7 G168 Face thread

Syntax G168(SR,SA,ER,TP,DT,FD,NI,CDM,IT,CT,ROF,NFR,TFA,TSA,NT,MTR)

Brief description The cycle "G168 Face thread" machines face threads, i.e. plane spirals by turning.

Overview

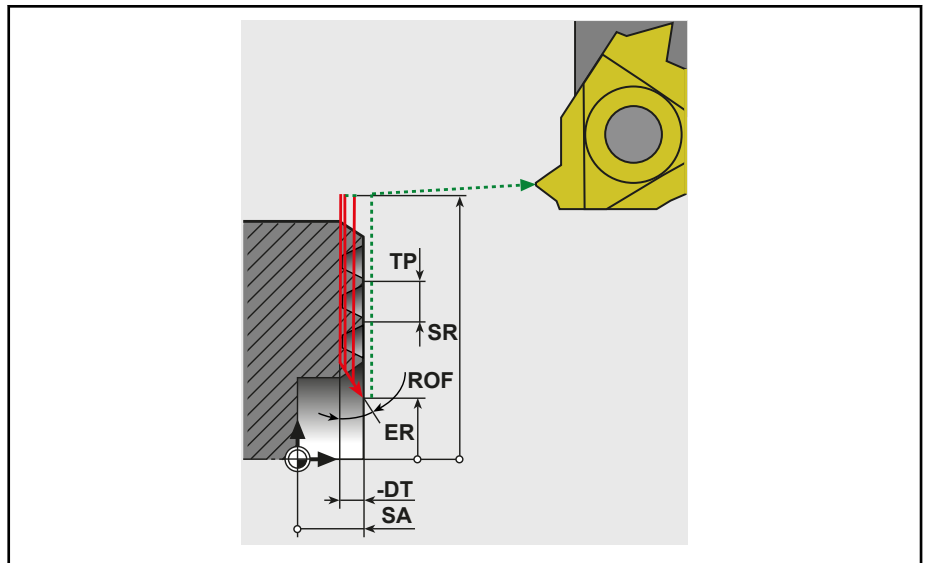


Fig. 10-16: G168 Face thread

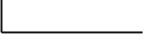
Parameter

Parameter	Designation	Properties	Values	Pre-assignment
SR	Starting point, radial	Mandatory	Absolute	
SA	Starting point, axial	Mandatory	Absolute	
EA	End point, axial	Mandatory	Absolute	
TP	Thread pitch	Mandatory	>0	
DT	Thread depth	Mandatory	Incremental < 0 right-sided > 0 left-sided	
FD	Initial infeed depth	Optional optionally specify for NI	Incremental >0	Results from NI
NI	Number of cuts	Optional Optionally specify for FD	>0	Internal algorithm (empirical values)
CDM	Minimum infeed depth	Optional	>0	0.05 mm
IT	Infeed variant	Optional		2
CT	Cutting variant	Optional		4
ROF	Retracting angle	Optional	0..70 degrees 0 indicates no slope	0

Turning cycles

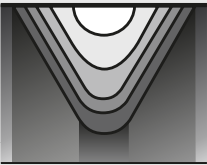
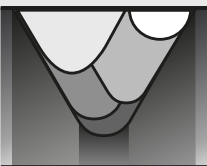
Parameter	Designation	Properties	Values	Pre-assignment
NFR	Number of glazed cuts	Optional	≥ 0	0
TFA	Edge angle	Optional	Degree $0 \leq TFA < 180$	60
TSA	Starting angle	Optional	Degree	
NT	Number of turns	Optional	> 0	1
MTR	Processing sequence of several turns of the thread	Optional only active in case $NT > 1$	simultaneously (1) / subsequently (2)	1

Tab. 10-17: Parameter call

IT	Effect
1	Degressive infeed strategy by complying with a constant chip cross section
2	Degressive infeed from the third cut with increasing chip cross section
3	Constant infeed around FD, recommended for tetragonal thread
<pre>10 START? = FALSE</pre>  BOOLEAN variable	

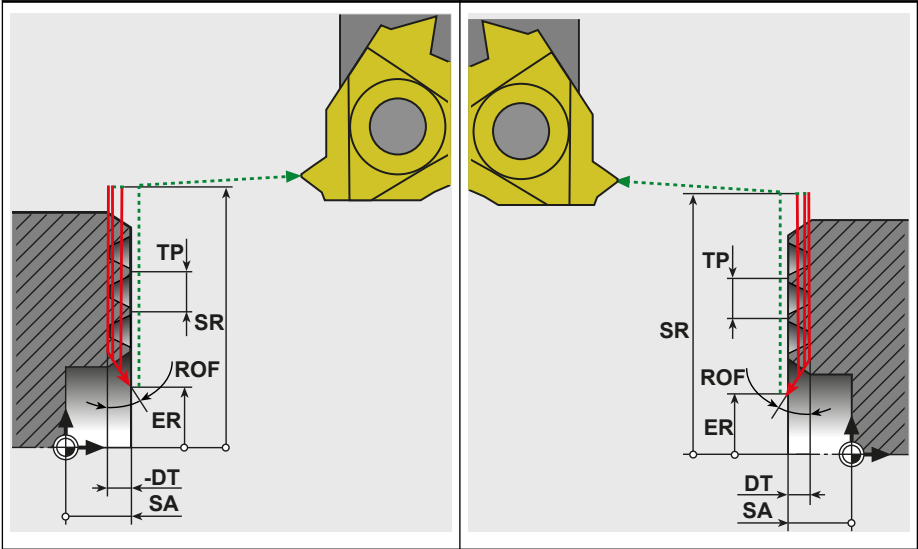
Tab. 10-18: Description of the parameter DEG - infeed variant

CT	Effect
1	Tool guidance, axis-parallel
2	Tool guidance along an edge
3	Tool guidance, alternating between both edges

CT	Effect
4	Tool guidance, alternating
CT1  CT2  CT3  CT4 	

Tab. 10-19: Description of the parameter CT - cut variant

Description



Tab. 10-20: Dimensions of the face thread on the right and left.

SR Starting point, radial

The thread starts at this absolute, radial position

SA Starting point, axial

The thread starts at this absolute, axial position

ER End point, radial

The thread ends at this absolute, radial position

TP Thread pitch

Pitch of thread

DT Thread depth

The sign determines the direction in which the depth infeed is made with each cut. This means that DT for face threads has to be < 0 on the right-hand (that is the z-positive side smaller than zero).

FD First infeed depth

Depth of first cut

NI Number of cuts

Defines the number of cuts without glazed cuts

CDM Minimum infeed depth

Minimum infeed depth that is not fallen below

FD or NI can be adjusted in the cycle to comply with the minimum infeed depth

IT Infeed variant

Determines the infeed depth from the second cut

CT Cutting variant

Specifies the position of the tool travels relatively to the completed thread profile.

ROF Retracting angle

If $ROF > 0$, a slope for radial retracting is inserted at the end of the thread

NFR Number of glazed cuts

Number of cuts after the thread depth has been reached without infeed/chip removal.

TFA Edge angle

Opening angle between two edges of the thread profile

For cutting variant CT2, it is recommended to reduce the programmed edge angle (compared to the actual angle), to avoid friction at one side of the tool edge and the workpiece surface (modified edge infeed)

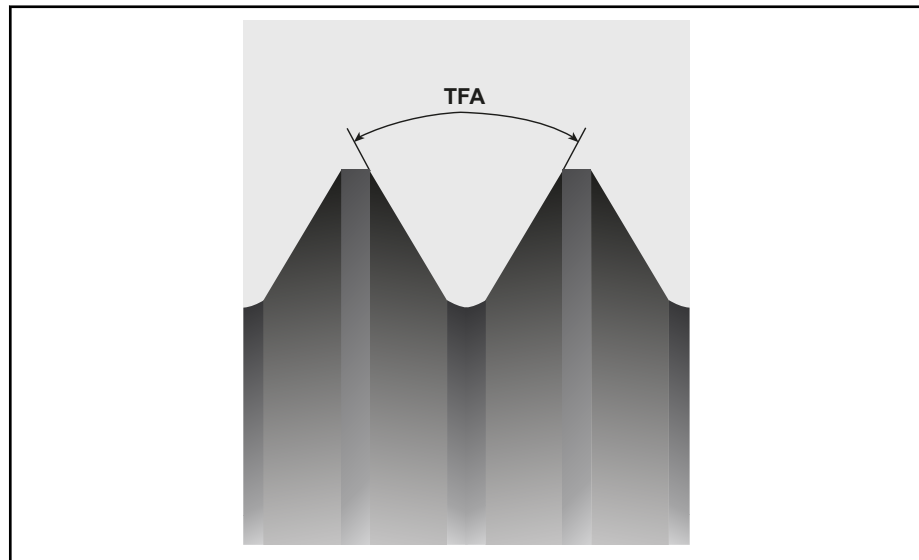


Fig. 10-17: Edge angle TFA

TSA Starting angle

Is to be programmed if the thread start has to pass through a specific reference point at the workpiece. The starting angle specified by TSA is executed at the starting coordinate SR/SA. The starting angle of the thread to be pro-

grammed is calculated from the distance of the starting point of the reference point on the workpiece, the thread pitch and the direction of rotation.

NT Number of turns

Defines the number of thread turns (for multiple threads)

MTR Processing sequence of several turns of the thread

Influences whether all turns of a thread are machined simultaneously or whether one turn of a thread is to be completed before the next one is machined.

Special features

The cycle can be used in every interpolation plane, although turning is usually performed in plane G18. The main axis of the plane is the axial axis and the secondary axis is the radial axis. If the cycle is for once called in G19, the z-axis is the axial axis and the y-axis is the radial axis.

Sequence/processing

The cycle positions the tool at the starting point SR and SA. The target coordinates of the motion are displaced by the retract amount of 0.5 mm in axial direction. It has to be possible to approach this point directly and without collision, starting from the position reached prior to the cycle. Then, the tool travels to the initial cut and to the initial infeed depth on the workpiece. A thread cut is performed. If the retraction angle $ROF > 0$ has been programmed, the end point of the thread is not reached but a slope is inserted for radial retracting by retaining the axis coupling. The tool is retracted and travels to the infeed of the next cut. The individual thread cuts are performed cyclically until the end depth is reached. Glazed cuts can be optionally performed. The cycle ends with the tool traveling to the starting point of the cycle and then by returning to its position at the starting point before the cycle call.

Example

Two-turn of thread spirals (face thread) with a trapezoid cross section of 5x5 are to be machined. The center of the thread groove has to cut the cross section 100 at the angular pitch of -30° or 150° .

A quarter revolution for approach and run-out are provided each (corresponding to a radial additional distance of 10 mm diameter in this case a pitch of 20 mm). The infeed is always carried out in axis-parallelly.

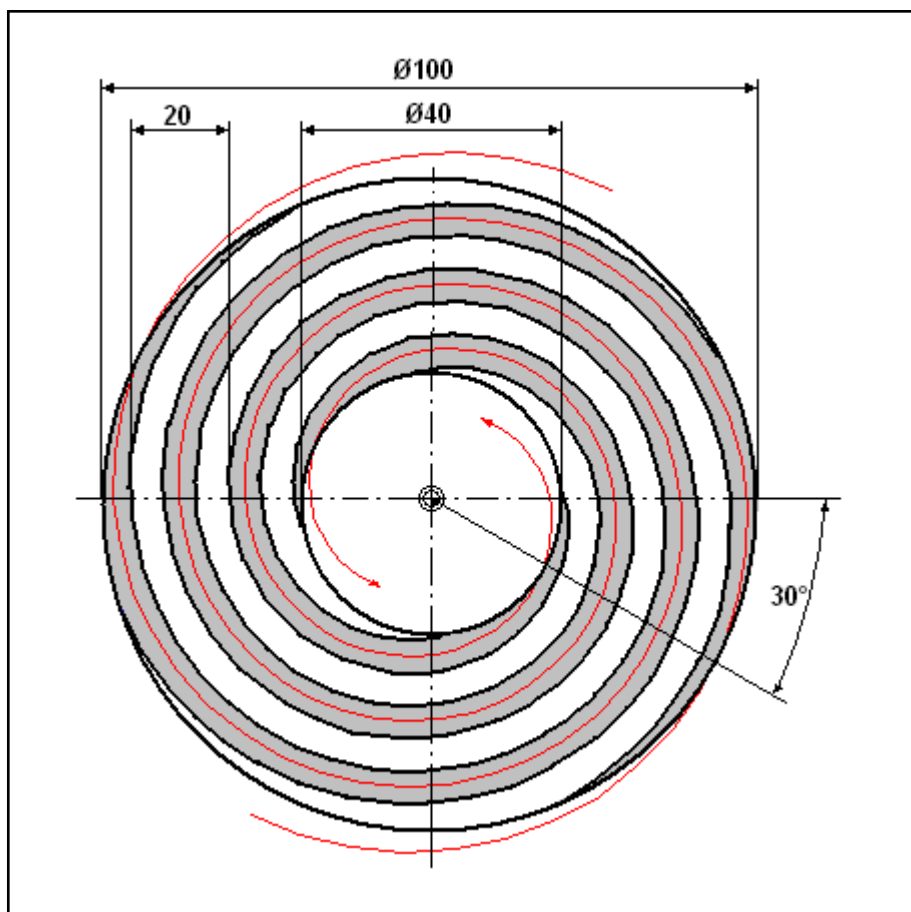


Fig. 10-18: Example G168 Face thread
Program for G168 Face thread

```

N10 T6 M6 ;Tool change
N20 G18 DIA G97 S200 M4 ;Technology values
N30 G0 X120 Z5 ;Prepositioning
N40 G168 (SR110, SA0, ER30, TP20, DT-5, FD0.5, IT3, CT1, TSA-120, NT2) ;Cycle Face thread
N50 X200 Z100 M5 ;leave contour

```

10.8 G169 Tapered thread

Syntax G169(SR,SA,ER,EA,TP,DT,FD,NI,CDM,IT,CT,ATH,RTH,ROF,NFR,TFA,TSA,NT,MTR)

Brief description The cycle "G169 Tapered thread" is used to turn conical threads including the borderline cases of cylindrical and face threads.

Overview

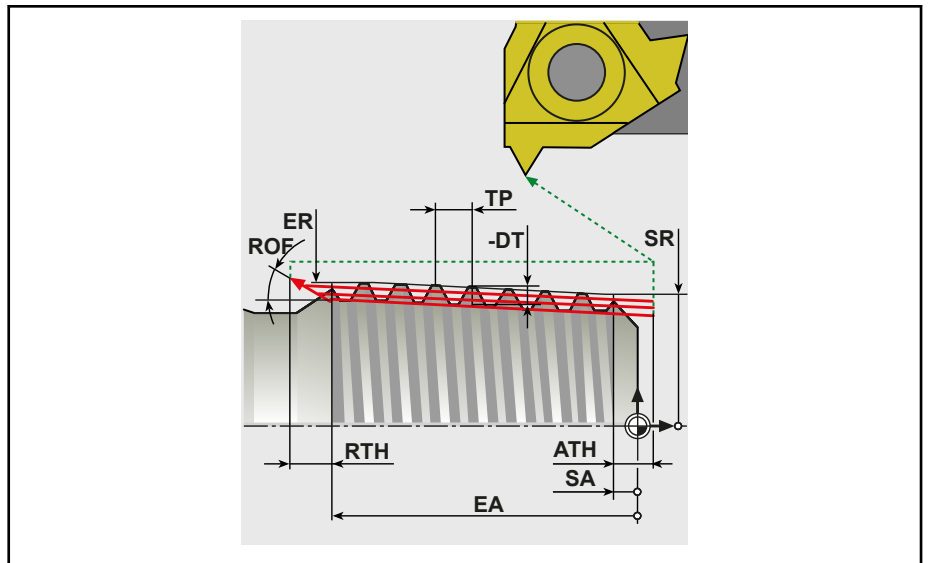


Fig. 10-19: G169 Tapered thread

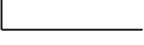
Parameter

Parameter	Designation	Properties	Values	Pre-assignment
SR	Starting point, radial	Mandatory	Absolute	
SA	Starting point, axial	Mandatory	Absolute	
ER	End point, radial	Mandatory	Absolute	
EA	End point, axial	Mandatory	Absolute	
ROF	Retracting angle	Optional Inactive when specifying RR and RA	0..70 degrees 0 indicates no slope	0
TP	Thread pitch	Mandatory	>0	
DT	Thread depth	Mandatory	Incremental < 0 external thread > 0 internal thread	
FD	Initial infeed depth	Optional optionally specify for NI	Incremental >0	Results from NI
NI	Number of cuts	Optional Optionally specify for FD	>0	Internal algorithm (empirical values)
CDM	Minimum infeed depth	Optional	>0	0.05 mm
IT	Infeed variant	Optional		2

Turning cycles

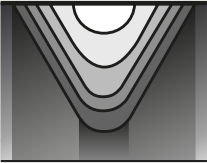
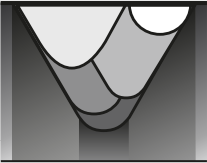
Parameter	Designation	Properties	Values	Pre-assignment
CT	Cutting variant	Optional		4
ATH	Approach	Optional	Incremental	0
RTH	Run-out	Optional	Incremental	0
ROF	Retracting angle	Optional	0..70 degrees 0 indicates no slope	0
NFR	Number of glazed cuts	Optional	>=0	0
TFA	Edge angle	Optional	Degree 0<=TFA<180	60
TSA	Starting angle	Optional	Degree	
NT	Number of turns	Optional	>0	1
MTR	Machining sequence of several turns of the thread	Optional only active in case NT>1	simultaneously (1) / subsequently (2)	1

Tab. 10-21: Parameter call

IT	Effect
1	Degressive infeed strategy by complying with a constant chip cross section
2	Degressive infeed from the third cut with increasing chip cross section
3	Constant infeed around FD, recommended for tetragonal thread
10 <u>START?</u> = FALSE  BOOLEAN variable	

Tab. 10-22: Description of the parameter DEG - infeed variant

CT	Effect
1	Tool guidance, axis-parallel
2	Tool guidance along an edge
3	Tool guidance, alternating between both edges Not permitted for tapered thread

CT	Effect
4	Tool guidance, alternating
CT1  CT2  CT3  CT4 	

Tab. 10-23: Description of the parameter CT - cut variant

Description

10 START? = FALSE BOOLEAN variable	10 START? = FALSE BOOLEAN variable

Tab. 10-24: Dimensions of the tapered thread in internal and external thread

SR Starting point, radial

This absolute radial position is a known reference point close to the start of thread

SA Starting point, axial

This absolute axial position is a known reference point close to the start of thread

ER End point, radial

This absolute radial position is a known reference point close to the end of thread

EA End point, axial

This absolute axial position is a known reference point close to the end of thread

TP Thread pitch

Pitch of main thread

DT Thread depth

The sign determines the direction in which the depth infeed is made with each cut. This means that DT has to be < 0 for external threads and in the x-negative range < 0 for internal threads.

FD First infeed depth

Depth of first cut

NI Number of cuts

Defines the number of cuts without glazed cuts

CDM Minimum infeed depth

Minimum infeed depth that is not fallen below

FD or NI can be adjusted in the cycle to comply with the minimum infeed depth

IT Infeed variant

Determines the infeed depth from the second cut

CT Cutting variant

Specifies the position of the tool travels relatively to the completed thread profile.

ATH Approach

Axis-parallel offset of the start of thread against SR or SA. ATH is a user-friendly function to facilitate defining the start of thread at the tapered thread.

RTH Approach

Axis-parallel offset of the end of thread against ER or EA. RTH is a user-friendly function to facilitate defining the end of thread at the tapered thread.

ROF Retracting angle

If $ROF > 0$, a slope for radial retracting is inserted at the end of the thread

NFR Number of glazed cuts

Number of cuts after the thread depth has been reached without infeed/chip removal.

TFA Edge angle

Opening angle between two edges of the thread profile. The bisector runs parallel to the axis

For cutting variant CT2, it is recommended to reduce the programmed edge angle (compared to the actual angle), to avoid friction at one side of the tool edge and the workpiece surface (modified edge infeed)

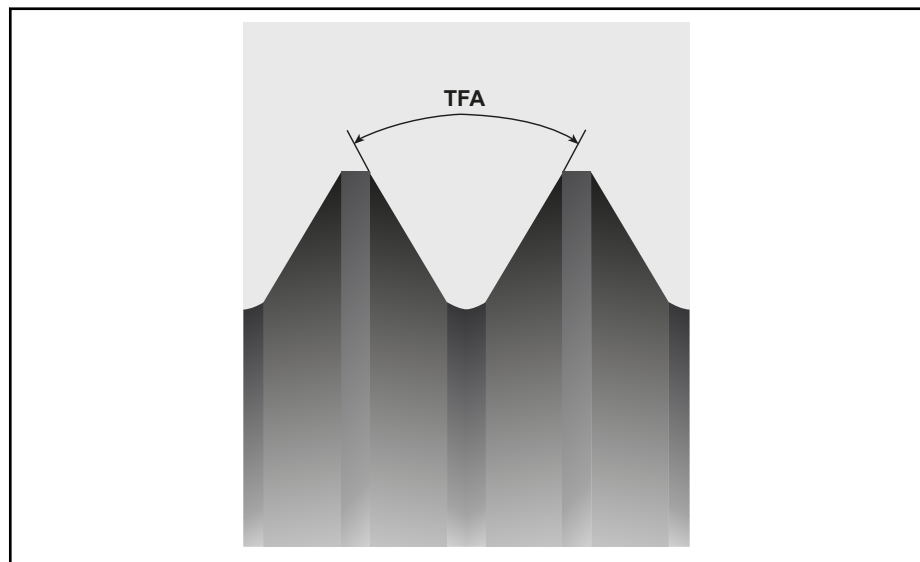


Fig. 10-20: Edge angle TFA

TSA Starting angle

Is to be programmed if the thread start has to pass through a specific reference point at the workpiece. The starting angle specified by TSA is implemented at the starting point AR/AA or SR/SA. The starting angle of the thread to be programmed is calculated from the distance of the starting point of the

reference point on the workpiece, the thread pitch and the direction of rotation.

NT Number of turns

Defines the number of thread turns (for multiple threads)

MTR Processing sequence of several turns of the thread

Influences whether all turns of a thread are machined simultaneously or whether one turn of a thread is to be completed before the next one is machined.

TPA Thread approach pitch

Special pitch for thread approach

TPR Thread approach pitch

Special pitch for thread approach

Special features

The cycle can be used in every interpolation plane, although turning is usually performed in plane G18. The main axis of the plane is the axial axis and the secondary axis is the radial axis. If the cycle is for once called in G19, the z-axis is the axial axis and the y-axis is the radial axis.

The thread pitch TP is measured in parallel to the axis. They are applied in case of a longitudinal pitch at a taper angle of the main thread of $\leq 90^\circ$. The tool is set radially. The pitch is considered to be a face pitch from a taper angle greater than 90 degrees. The tool is set axially.

Sequence/processing

The cycle positions the tool to a starting point resulting from the thread run in the pitch direction. In the infeed direction, it is the maximum (at $DT < 0$) or minimum (at $DT > 0$) of all the thread coordinates, which is additionally displaced by a retract amount of 0,5 mm towards the infeed direction. It has to be possible to approach this point directly and without collision starting from the position reached prior to the cycle. Then, the tool travels to the initial cut and to the initial infeed depth on the workpiece. A thread cut is performed. If the retraction angle $ROF > 0$ has been programmed, the end point of the thread is not reached but a slope is inserted for radial retracting by retaining the axis coupling. The tool is retracted (whereby the retract coordinate corresponds to those of the starting point) and travels axis-parallelly to the infeed of the next cut. The individual thread cuts are performed cyclically until the end depth is reached. Glazed cuts can be optionally performed. The cycle ends with the tool traveling to the starting point of the cycle and then by returning to its position at the starting point before the cycle call.

Example

The thread is to be machined by edge infeed. The thread approach and run-out is to be 5 mm.

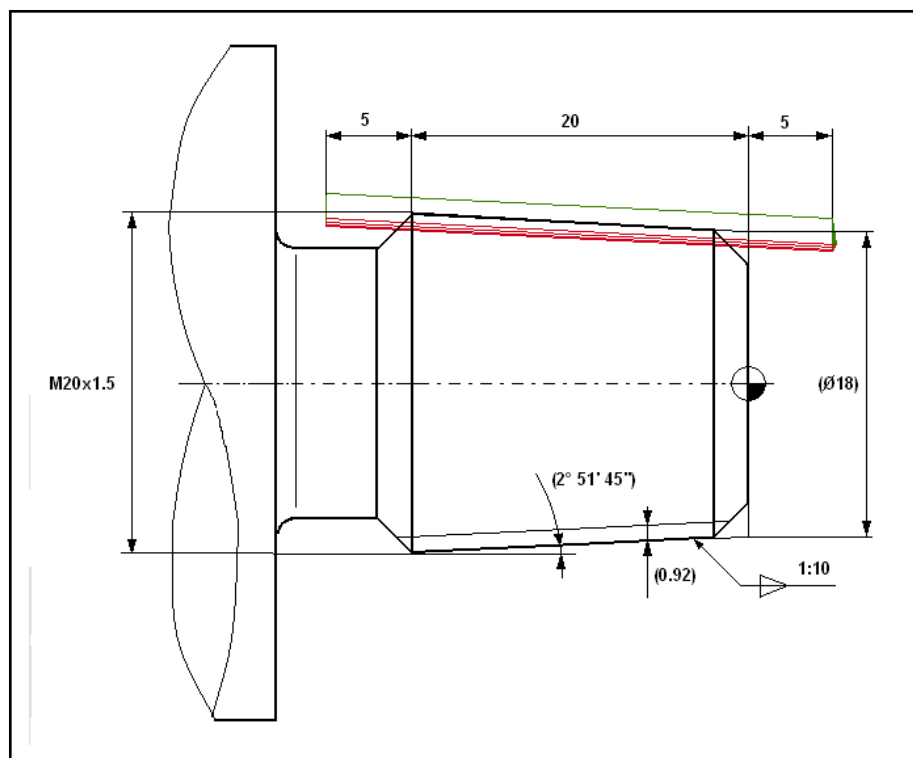


Fig. 10-21: Example G169 Thread turning

Program for G169 Tapered thread

```

N10 T6 M6                                ;Tool change
N20 G18 DIA G97 S500 M3                  ;Technology values
N30 G0 X25 Z5                             ;Prepositioning
N40 G169 (SR18, SA0, ER20, EA-20, TP1.5, DT-0.92, CT2, ATH5, RTH5)
                                           ;Cycle Tapered thread
N50 X200 Z100 M5                         ;Leave contour

```

10.9 G191 Rounded groove turning

Syntax G191(ORI,SR,SA,DT,W,TW,A1,A2,R1,CL1,RS,CL2,R3,CL3,R4,CL4,CV,CT,DW,FI,FF,RSL)

Short description The cycle "G191 Groove turning" machines grooves with trapezoidal and rectangular cross sections with a recessing tool.

Overview

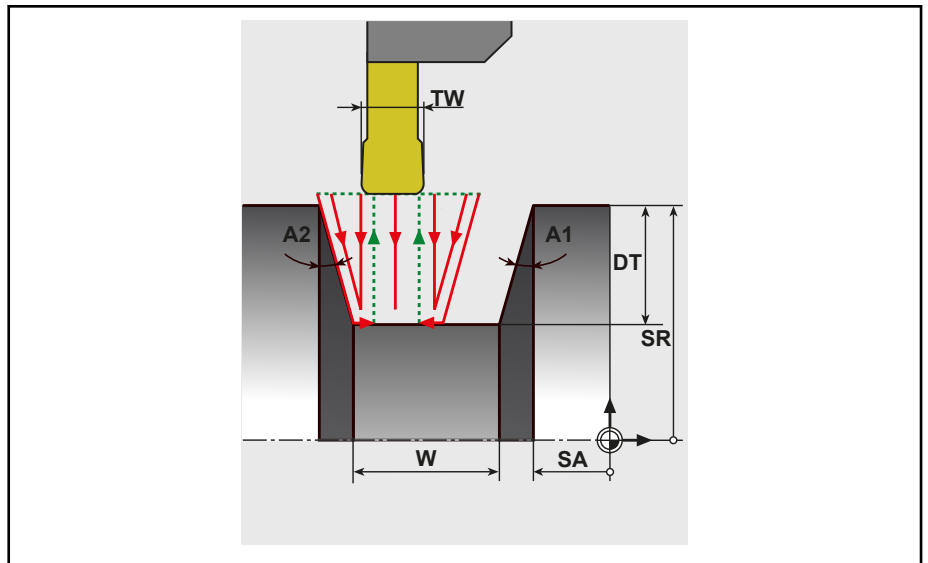


Fig. 10-22: Cycle G191 Groove turning

Parameter

Parameter	Name	Properties	Values	Pre-assignment
ORI	Position orientation	Optional		8
SR	Reference coordinate, radial	Mandatory	Absolute	
SA	Reference coordinate, axial	Mandatory	Absolute	
DT	Depth	Mandatory	Incremental, >0	
W	Width of basic straight line	Mandatory	Incremental, <>0	
TW	Tool width	Mandatory	>0	
A1	Angle Edge 1	Optional	0..88, Degree	0
A2	Angle Edge 2	Optional	0..88, Degree	0
R1	Radius Edge 1	As an option, optionally specify in addition to CL1	>=0	No rounding
CL1	Chamfer width Edge 1	As an option, optionally specify in addition to R1	>=0	No chamfer

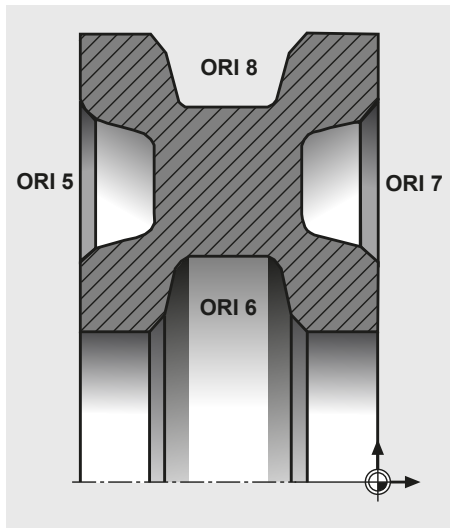
Turning cycles

R2	Radius Base/ Edge 1	As an option, optionally specify in addi- tion to CL2	≥ 0	No rounding or R1
CL2	Chamfer width Base/Edge 1	As an option, optionally specify in addi- tion to R2	≥ 0	No chamfer or CL1
R3	Radius Base/ Edge 2	As an option, optionally specify in addi- tion to CL3	≥ 0	No rounding or R1
CL3	Chamfer width Base/Edge 2	As an option, optionally specify in addi- tion to R3	≥ 0	No chamfer or CL1
R4	Radius Edge 2	As an option, optionally specify in addi- tion to CL4	≥ 0	No rounding or R1
CL4	Chamfer width Edge 2	As an option, optionally specify in addi- tion to R4	≥ 0	No chamfer or CL1
CV	Machining (cut- ting) variant	Optional	Roughing Complete (2) / Finishing (3)	2
CT	Cutting variant	Optional		1
CD	Maximum in- feed depth	Optional	> 0	Infeed once up to finishing al- lowance RSL
DW	Dwell time	Optional	≥ 0	2 dwell rota- tions
FI	Feed infeed	Optional	> 0	Current feed
FF	Finishing feed	Optional	> 0	Current feed
RSL	Finishing allow- ance	Optional	≥ 0	0

Tab. 10-25: Parameter call

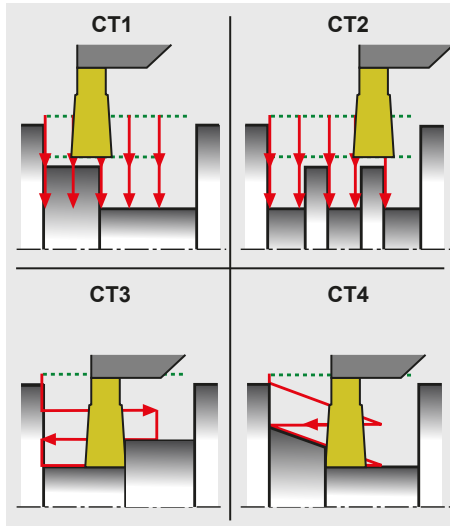
IT	Effect
5	Axial, grooving in positive axis direction
6	Radial, grooving in positive axis direction
7	Axial, grooving in negative axis direction

IT	Effect
8	Radial, grooving in negative axis direction



Tab. 10-26: Description of parameter ORI - Position orientation

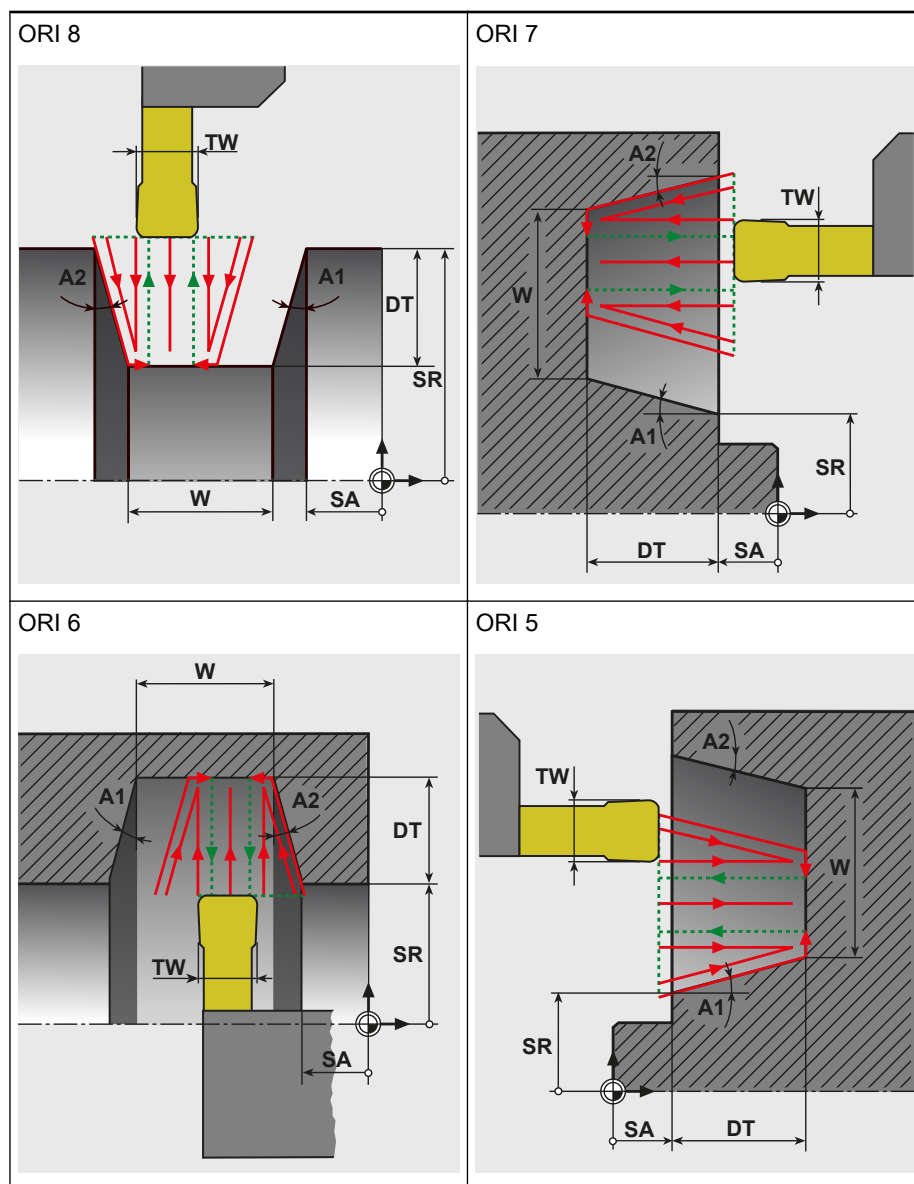
CT	Effect
1	Conventional grooving
2	Multiple grooving
3	Plunge-turning
4	Inclined grooving



Tab. 10-27: Description of parameter CT - Cutting variant

Description ORI Position orientation

Determines the direction of definition of the dimensions and the groove direction



Tab. 10-28: Main groove dimensions depending on the position orientation

SR Reference coordinate, radial

Defines the existing diameter for radial grooves and the start of the edge for axial grooves (without chamfer/rounding)

SA Reference coordinate, axial

Defines the existing plane surface for radial grooves and the start of the edge for radial grooves (without chamfer/rounding)

DT Depth

Groove depth

W Width of basic straight line

Width of groove base without chamfers/roundings

In case of a negative W, the reference point SR/SA changes to the opposite edge

TW Tool width

Width of the grooving tool. Corresponds to the distance of the two tool cutting edges

A1 Angle Edge 1

Angle of inclination of first edge

A2 Angle Edge 2

Angle of inclination of second edge

R1 Radius Edge 1

Rounding radius at the opening of the first edge

If R1 is the only parameter that is programmed out of parameters R1..R4 and CL1..CL4, a rounding is provided at all four transitions

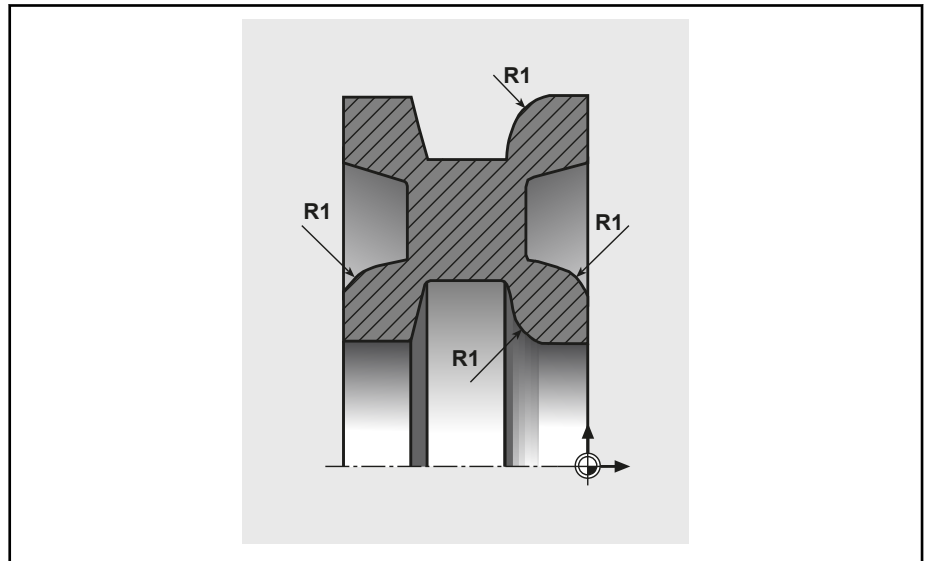


Fig. 10-23: Radius Edge 1 depending on the position orientation

CL1 Chamfer width Edge 1

Chamfer width at the opening of the first edge

If CL1 is the only parameter that is programmed out of parameters R1..R4 and CL1..CL4, a chamfer is provided at all four transitions

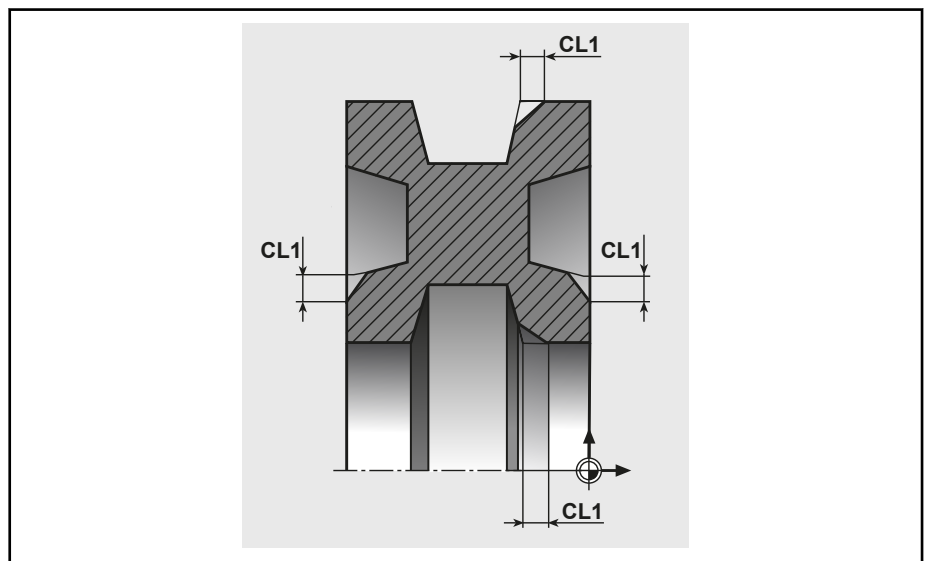


Fig. 10-24: Chamfer width Edge 1 depending on the position orientation

R1 Radius Edge 2

Rounding radius at the opening of the second edge

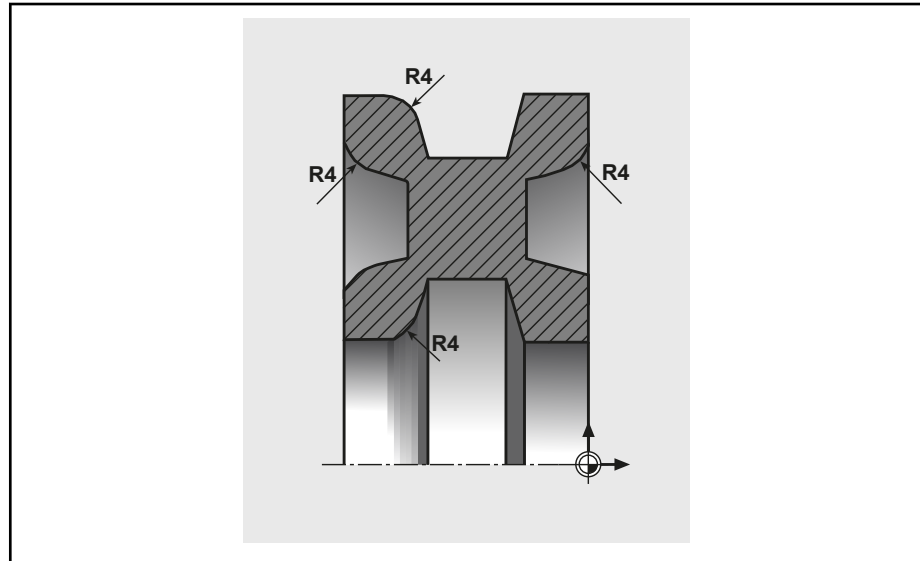


Fig. 10-25: Radius Edge 2 depending on the position orientation

CL2 Chamfer width Edge 2

Chamfer width at the opening of the second edge

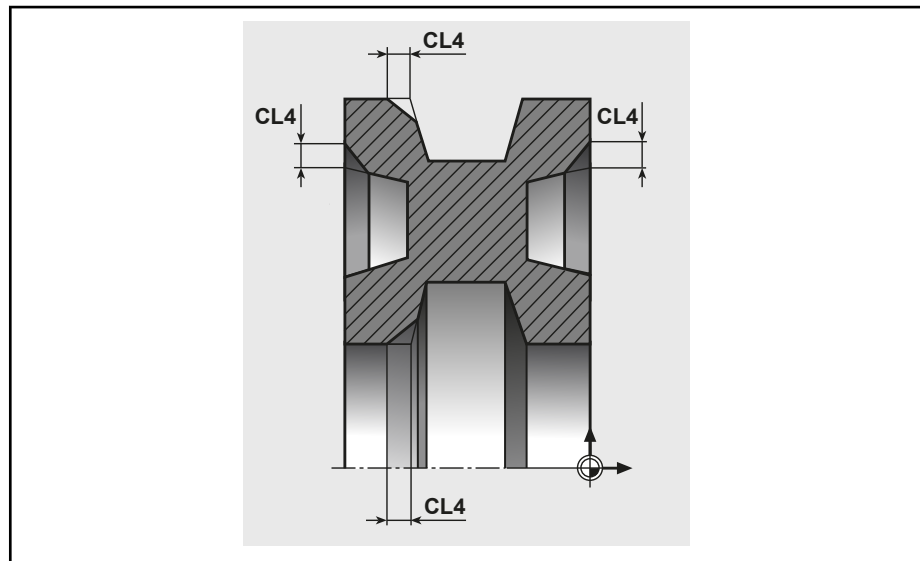


Fig. 10-26: Chamfer width Edge 2 depending on the position orientation

CV Machining (cutting) variant

Selecting premachining and finish-machining

CT Cutting variant

Selecting the cut guidance during rough grooving

CD Maximum infeed depth

Maximum depth during grooving in the material

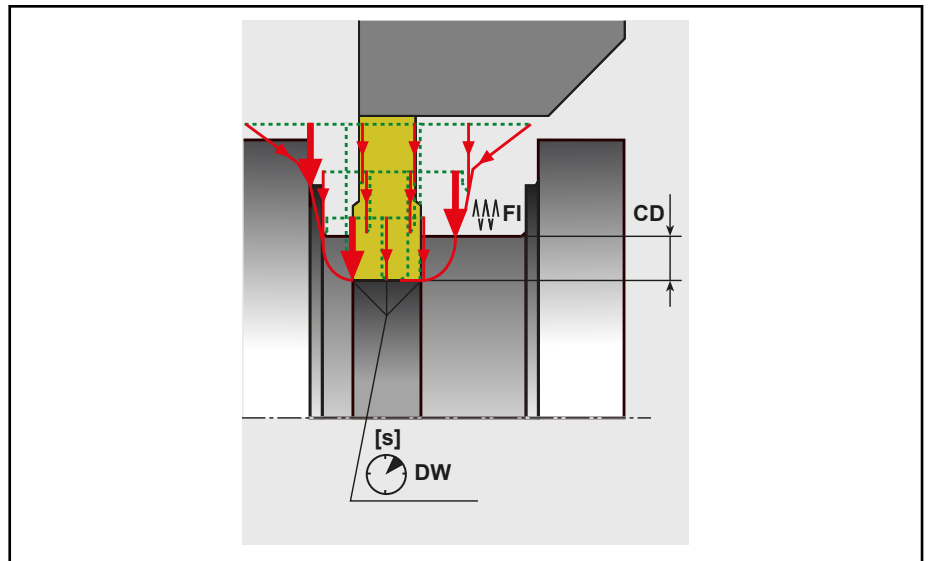


Fig. 10-27: Maximum infeed depth, feed during infeed and dwell time

DW Dwell time

This time interval is waiting when completing the finished dimension before changing direction to decrease the cutting pressure

FI Feed infeed

Feed during grooving in the material

FF Feed finishing

Feed during finish-machining

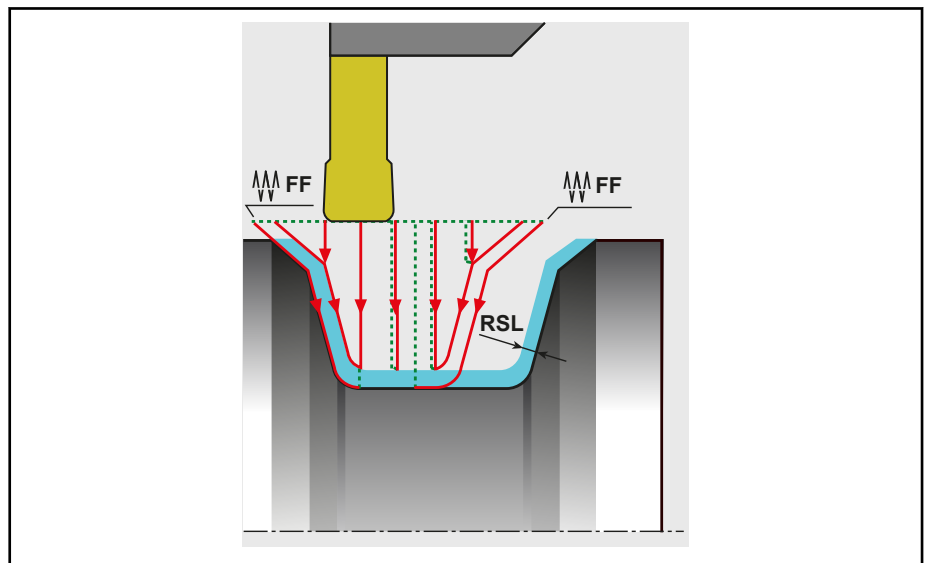


Fig. 10-28: Parameters for finishing

RSL Finishing allowance

Contour-parallel allowance during roughing that is retained as machining allowance for finishing

Special features

The cycle can be used in every interpolation plane, although turning is usually performed in plane G18. The main axis of the plane is the axial axis and the secondary axis is the radial axis. If, as an exception, the cycle is called in G19, the z-axis is the axial axis and the y-axis is the radial axis.

Turning cycles

Sequence/processing

The cycle calculates the motion sequence for the left edge. Prior to the cycle, always activate the left edge. The edge position has to be smaller than 5, matching the position orientation.

The initial position in grooving direction should be close to the tool surface. The distance of the initial position to the reference point SR or SA becomes active as run up distance in the feed.

The rounding R1 or the chamfer CL1 is applied to all transitions as long as no other chamfer/rounding has been parameterized. If required, another parameter of R2..CL4 has to be transmitted with zero.

The cycle can sporadically reduce the chamfers and roundings by reporting warnings.

Approaching the starting point of machining starts with positioning in cross direction. It is immediately travelled in feed in grooving direction.

Roughing:

Roughing consists of grooving and rough turning of the edges. During rough turning of the edges, steps that might have been left out during grooving are also cut.

In case of conventional grooving, the first groove is executed from the specified infeed depth in radial/axial direction. In case of radial grooves, it is started at the Z-negative side, in case of axial grooves, it is started at the outer side. Depending on the grooving width, more grooves with lateral overlapping and subsequently, pre-machining of the edges take place in steps. If the infeed depth has been limited, grooving can be continued in more infeed steps until reaching the finishing allowance. The direction changes in case of radial, external grooves.

Plunge-turning is similar to conventional grooving but without overlapping cuts. Ridges caused by grooving in the material with a width of 60% to 80% of the tool width are subsequently cut in reversed order before a new infeed can take place. Conventional grooving of the edges up to the current cutting depth is integrated in cutting the ridges. If the grooving width is insufficient for multiple grooving, conventional grooving takes place.

During plunge-grooving, cross turning (meander-shaped motion) takes place after each infeed. The edges are pre-machined in steps only after reaching the finishing allowance.

Inclined grooving is very similar to plunge-grooving, whereas the infeed is overlapped with cross turning. After each infeed, an axis-parallel cross turning follows.

Following the grooving, the edges including the transition element are turned to finishing allowance (contour turning), each edge starting at the groove opening. The basic straight line is not skimmed during roughing with RSL>0.

Finishing:

Finishing usually starts with grooved at the slot base to the final depth. Edges and the base are machining in the order specified for roughing. In case of radial external grooves, the edge sequence is determined by the minimum non-cutting tool path.

In case of finishing, steps can be omitted or can be limited to the upper roundings/chamfers if the groove has one/several rectangular edges and corresponds to tool width W.

At the end of the cycle, it is travelled to the starting position in grooving direction. Subsequently, it is traveled in traverse direction.

Example The following groove is to be conventionally machined with a grooving tool with a width of 6 mm. The maximum grooving depth is limited to 13 mm by the tool.

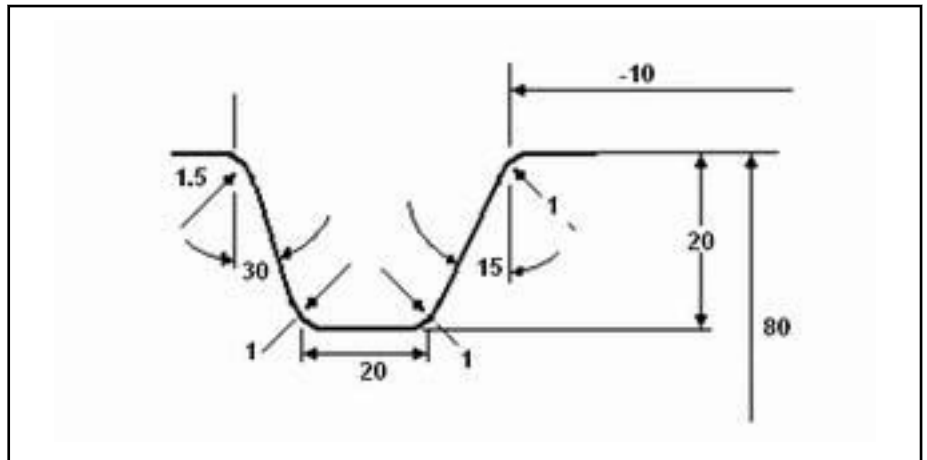


Fig. 10-29: Example G191 Groove turning

Program for G191 Groove turning

```

N10 T1 M6                                ;Radial grooving tool width 6
N20 G18 DIA G95 G96 F0.12 S160 M4        ;Technology values
N30 G0 G47 ED1 X82 Z-10                  ;left edge, approach position
N40 G191 (SR80, SA-10, DT20, W20, TW6, A1 15, A2 30, R1 1, R2 1,
        R3 1, R4 1.5, CD13, RSL0.2) ;Cycle Plunging turning
N50 X200 Z100 M5                          ;Leave contour

```

10.10 G192 Rounded groove turning

Syntax G192(ORI,SR,SA,DT,R,CV,CD,FI,FF,RSL)

Brief description The cycle "G192 Rounded groove turning" machines grooves with a round base and - in case of sufficient depth - orthogonal, straight edges with a circular tool.

Overview

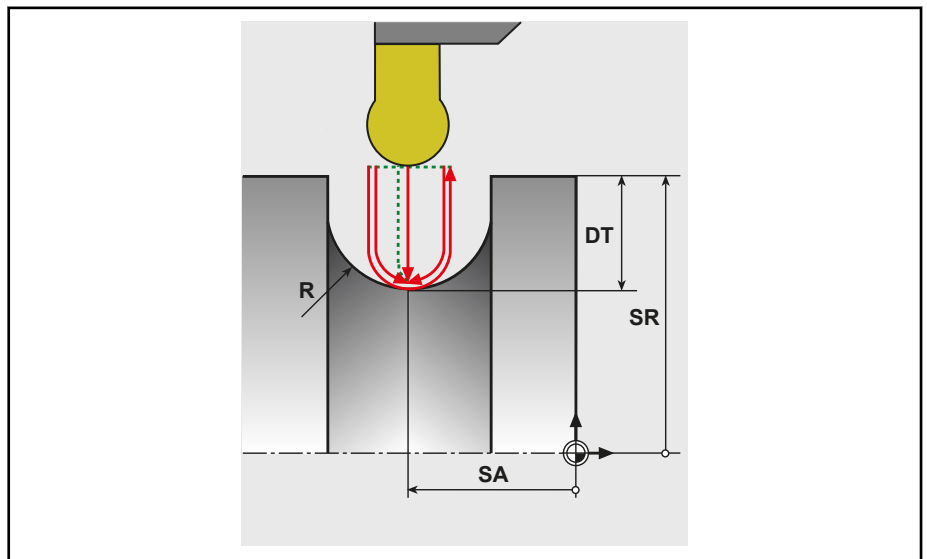


Fig. 10-30: Cycle G192 Rounded groove turning

Turning cycles

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
ORI	Position orientation	Optional		8
SR	Reference coordinate, radial	Mandatory	Absolute	
SA	Reference coordinate, axial	Mandatory	Absolute	
DT	Depth	Mandatory	Incremental >0	
R	Groove radius	Mandatory	Incremental >0	
CV	Machining (cutting) variant	Optional	Roughing (1) / Grooving and roughing (2) / Roughing and finishing (3) Complete (4) / Finishing (5)	4
CD	Maximum in-feed increment	Optional	0<CD≤edge radius	Edge radius
FI	Feed infeed	Optional	>0	Current feed
FF	Finishing feed	Optional	>0	Current feed
RSL	Finishing allowance	Optional	≥0	0

Tab. 10-29: Parameter call

IT	Effect
5	Axial, grooving in positive axis direction
6	Radial, grooving in positive axis direction
7	Axial, grooving in negative axis direction
8	Radial, grooving in negative axis direction
10 <u>START?</u> = FALSE  BOOLEAN variable	

Tab. 10-30: Description of the parameter ORI - Position orientation

Description

ORI Position orientation

Determines the definition direction of dimensions and the grooving direction

ORI 8 10 <u>START?</u> = FALSE BOOLEAN variable	ORI 7 10 <u>START?</u> = FALSE BOOLEAN variable
ORI 6 10 <u>START?</u> = FALSE BOOLEAN variable	ORI 5 10 <u>START?</u> = FALSE BOOLEAN variable

Tab. 10-31: Main dimensions of the groove depending on the position orientation

SR Reference coordinate, radial

Defines the available diameter for radial grooves and the start of an edge (without chamfer/rounding) for axial grooves

Reference coordinate, axial

Defines the available plane surfaces for axial grooves and the start of an edge (without chamfer/rounding) for radial grooves

DT Depth

Groove depth

R Groove radius

Radius at the groove base

CV Machining (cutting) variant

Selecting grooving operations, pre-machining and fine machining

CV1 and CV3 do not use center grooving and assume grooving with a previous tool. A minimum grooving width (equal to twice the current edge radius) has to be complied with

CD Maximum infeed increment

Maximum contour-parallel infeed between two cuts

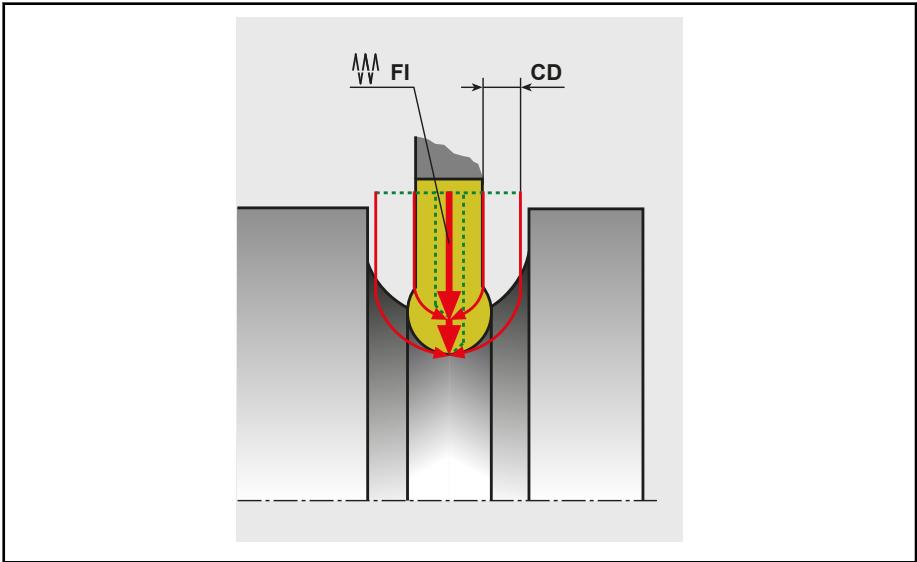


Fig. 10-31: Maximum infeed increment and infeed feed

FI Feed infeed

Feed during grooving

FF Finishing feed

Feed during fine machining

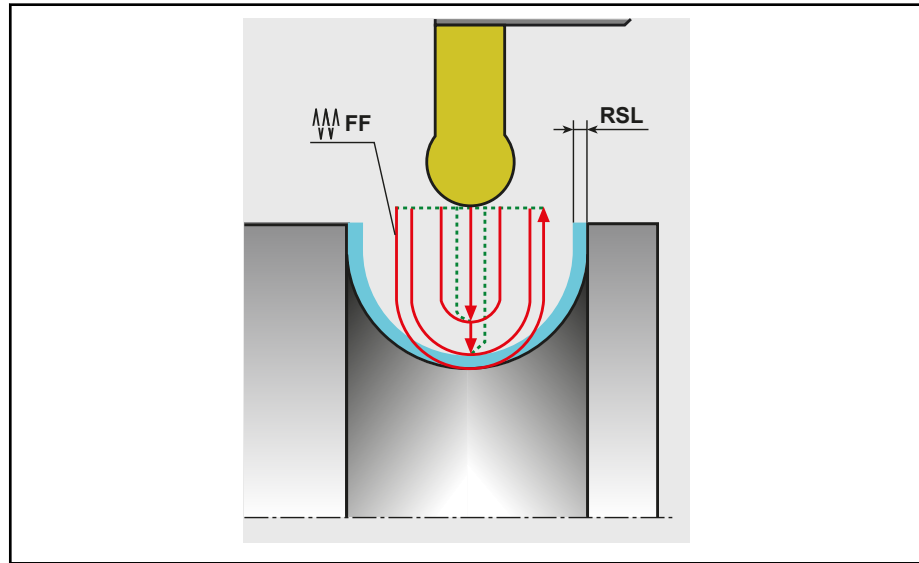


Fig. 10-32: Parameter for finishing

RSL Finishing allowance

Contour-parallel allowance during roughing, remaining as machining allowance for finishing

RSL also has to be specified in case of only finishing (CV5). Otherwise, or if RSL exceeds a sixth of the edge radius, the cut is not continuous but is executed from two sides

Special features

The cycle calculates the motion sequence for an edge corner. All edge positions 1..8 are valid as long as they match the position orientation (e.g., for ORI 8, edge positions 3,4 and 8).

The initial position in grooving direction should be close to the workpiece surface. The distance to the reference point SR or SA becomes active as approach distance in the feed.

As the cutting conditions are very poor when plunging into a material with a circular cutting tool, the machining variants CV2 and CV4 should only be used for selected machining cases and rather flat grooves. In general, this initial groove into the material is to be performed separately using a suitable groove turning tool.

Sequence/processing

Approaching the machining starting point starts with positioning in transverse direction. In the direction of grooving, it is immediately traveled in feed.

Grooving:

Grooving is executed from the center to the current cutting depth.

Roughing

During roughing, the contour is turned at each edge from the opening to the deepest point. If several cuts are required to reach the finishing allowance, a new infeed depth is rough grooved in the center. Subsequently, both edges are subsequently machined equidistantly.

Finishing

Finishing starts at one side of the groove and usually signifies turning of the entire profile in one move. If no finishing allowance RSL is specified or if the allowance exceeds a sixth of the edge radius, finishing is executed two sides.

At the end of the cycle, it is moved to the home position in the direction of grooving and subsequently in the transverse direction.

Example The following groove is to be machined using a circular tool with a width of 8 mm. The infeed is limited to 1.5 mm.

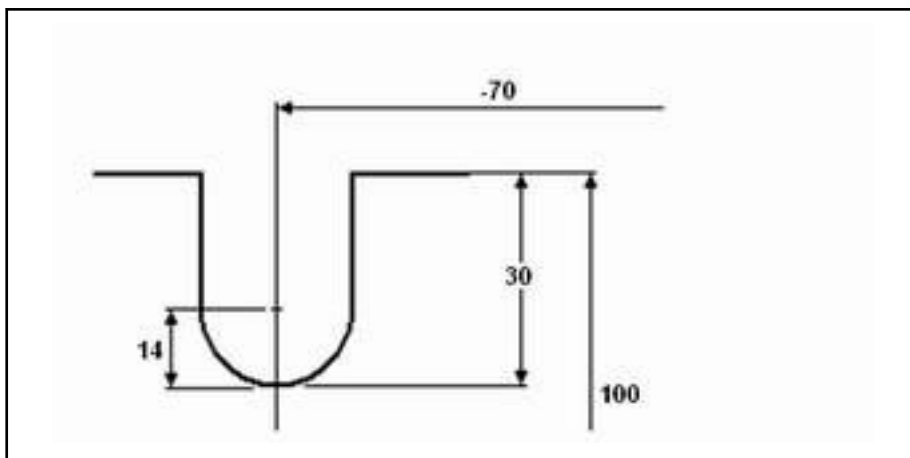


*Fig. 10-33: Example of G192 Round plunge turning
Program for G192 Rounded groove turning*

```

N10 T1 M6                                ;Radial circular tool width 8
N20 G18 DIA G95 G96 F0.1 S140 M4         ;Technology values
N30 G0 X102 Z-70                         ;Approach position
N40 G192 (SR100, SA-70, DT30, R14, CD1.5, RSL0.5)
N50 X200 Z100 M5                         ;Cycle Rounded groove turning
                                           ;Leave contour

```

10.11 G196 Thread undercut

The cycle "G196 Thread undercut" can machine thread undercuts according to DIN 76 Parts 1/2/3 or with user-defined dimensions. Thread undercuts can be implemented in a contour as a transitional element or turned separately (including approach and run-out from the starting point). Machining as such is restricted to the undercut contour (rather than cut segmentation) which is always oriented to extend from the axial plane towards the adjacent plane surface.

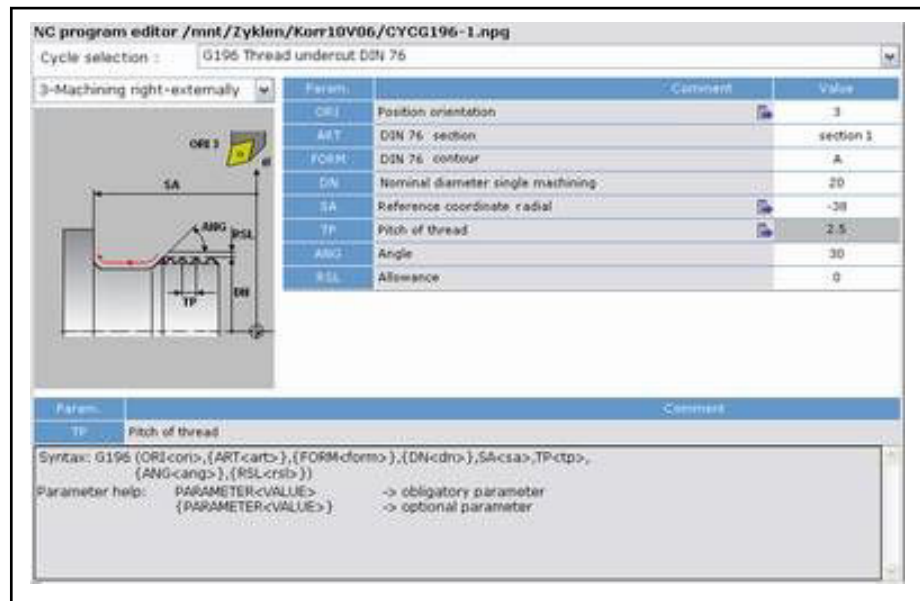


Fig. 10-34: G196 Thread undercut DIN 76

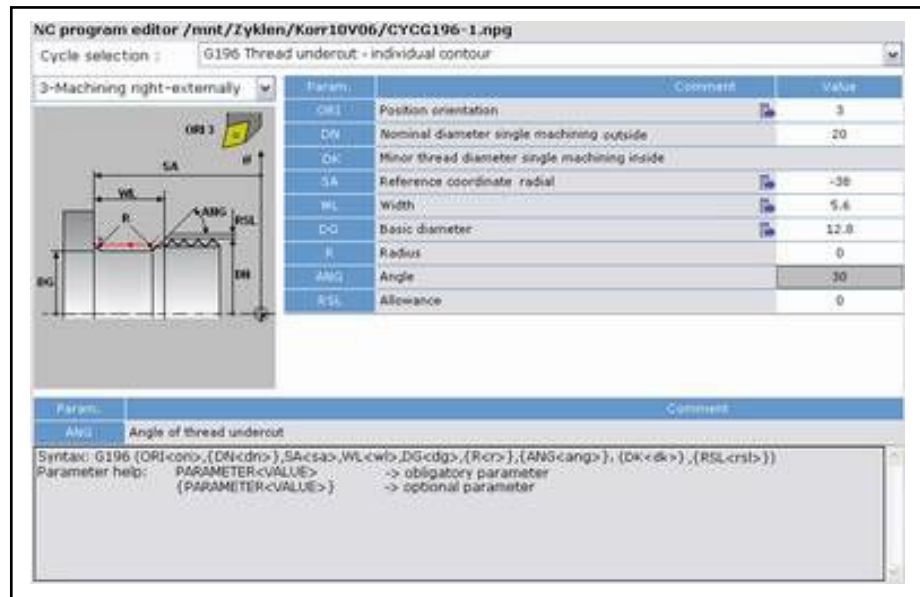


Fig. 10-35: G196 Thread undercut - free contour

G196		Thread undercut	G196(ART,ORI,DN,DK,SA,FORM,TP,DG,WL,R,ANG,RSL)			
TYPE	SD.SysCyc.ART	Type of thread undercut	Integer	optional	1<=ART<=3 0 (free form) is pre-assigned	
ORI	SD.SysCyc.ORI	Position orientation	Integer	Mandatory	1<=ORI<=4	

G196		Thread undercut	G196(ART,ORI,DN,DK,SA,FORM,TP,DG,WL,R,ANG,RSL)		
DN	SD.SysCyc.DN	Nominal diameter: single machining	Real	Optional; required for single machining according to DIN (ART>0) or for free contour (ART0) and external thread	DN<>0
DK	SD.SysCyc.DK	Core diameter: internal single machining	Real	Optional; only required for single machining, free contour (ART0) and internal thread	DK<>0
SA	SD.SysCyc.SA	Reference coordinate, axial	Real	Mandatory	Absolute
FORM	SD.SysCyc.FORM	Thread undercut DIN 76	String	Optional; can only be specified for DIN (ART > 0)	A,B,C,D,E,F The regular case is pre-assigned.
TP	SD.SysCyc.TP	Pitch of thread	Real	Mandatory for DIN (ART > 0)	TP>0
DG	SD.SysCyc.DG	Basic diameter of the thread undercut	Real	Mandatory for free contour (ART0)	DG>0
WL	SD.SysCyc.WL	Width	Real	Mandatory for free contour (ART0)	WL>0
R	SD.SysCyc.R	Radius	Real	Optional; can only be specified for free contour (ART0)	R>=0 0 is pre-assigned
ANG	SD.SysCyc.ANG	Angle	Real	optional	30<=ANG<=60 30 is pre-assigned
RSL	SD.SysCyc.RSL	Allowance	Real	optional	RSL>=0 0 is pre-assigned

Tab. 10-32:

Parameter features: The cycle can be used in **every interpolation plane**, although turning is usually performed in plane G18. The main axis of the plane is the axial axis and the secondary axis is the radial axis. If the cycle is for once called in G19, the z-axis is the axial axis and the y-axis is the radial axis.

With the active G71, all dimensions have to be programmed in mm, else in inches, with the radius dimension being applicable to all dimensions except for parameters DN, DK and DG.

If the thread undercut is called inside a contour, the radial reference coordinate is obtained from the radial position that was the last to be reached (coordinate of the secondary axis or y-coordinate in G19). This means that a cylinder surface is assumed as initial contour element. To allow turning the undercut separately from a contour, a diameter **DN** or **DK** has to be specified. DN is the **nominal thread diameter**. DN has to be defined for an undercut to be turned separately according to DIN 76 and for a separate outside undercut with user-defined dimensions. Optionally, the **kernel diameter DK** has to be specified in case of a separately rotating internal undercut with user-defined dimensions. Both diameters are always interpreted to have a diameter unit and have to have a negative sign if the undercut falls within negative radial coordinates.



If the SD data structure is used for programming, "0" has to be assigned to SD.SysCyc.DN and SD.SysCyc.DK, if necessary, to obtain a blank assignment.

The **reference coordinate axial SA** specifies the absolute position of the adjacent plane surface. The current axis scaling DIA/RAD has to be taken into account (particularly in G17).

The parameter **ART** refers to one of the three parts of DIN 76 for unit definition or the unit specification is activated via this parameter.

The following specifications can be made:

ART0	User-defined dimensions (default)
ART1	Dimensions according to DIN 76 Part 1 – for metrical ISO threads acc. to DIN 13-1
ART2	Dimensions according to DIN 76 Part 2 – for pipe threads acc. to DIN ISO 228 Part 1
ART3	Dimensions according to DIN 76 Part 3 – for trapezoidal, buttress, round and other threads with high pitch

The parameter **ORI** describes the **orientation of the undercut** which, along with the sign of the radial reference coordinate, allows a conclusion as to whether an undercut for an internal thread or an external thread is machined by calling this cycle.

The possible values correspond to the edge position code of the tool used to machine the undercut with the orientation of the same name:

ORI1	Contour pattern coming from the negative axial direction and continued in negative radial axis direction
ORI2	Contour pattern coming from the positive axial direction and continued in negative radial axis direction
ORI3	Contour pattern coming from the positive axial direction and continued in positive radial axis direction
ORI4	Contour pattern coming from the negative axial direction and continued in positive radial axis direction

The parameter **FORM** is used to obtain the dimensions (long, short, regular cases) from the part of DIN 76 that is addressed with the parameter ART in combination with **Pitch TP**. If the parameter FORM is not specified, the regular case set by ORI and the sign of the radial reference coordinate applies.

Possible specifications for the parameter FORM are:

FORM A	External thread, regular case acc. to Parts 1, 2 and 3
FORM B	External thread, short design acc. to Parts 1, 2 and 3
FORM C	Internal thread, regular case acc. to Parts 1 and 2. External thread, long design acc. to Part 3
FORM D	Internal thread, short design acc. to Parts 1 and 2. Internal thread, regular case acc. to Part 3
FORM E	Internal thread, short design acc. to Part 3
FORM F	Internal thread, long design acc. to Part 3

The **basic diameter DG** has to be specified for user-defined dimensioning (ART0). It defines the depth of the undercut. This value is always an average value without sign.

The **width WL** has to be specified as well for user-defined dimensioning (ART0) and defines the width of the undercut.

The **radius R** defines the internal curve and can be specified for user-defined dimensioning (ART0). By default, the current tool edge radius is applied as rounding radius.

The **angle ANG** defines the plunging angle and is 30° by default. It ranges from 30 to 60 for all ART parameter values.

The **allowance RSL** is used to optionally define a radial machining allowance on the previous cylinder surface (default 0). This parameter causes the plunging distance to be extended in tangential direction. If the undercut is integrated into a contour, the radial reference point is offset accordingly.

Sequence: If the undercut is called in a contour (neither DN nor DK is programmed), a linear interpolation in parallel to the main axis is executed with the active feed until the internally calculated undercut starting point is reached. The edge radius correction has to be activated prior to the cycle. The calculated undercut pattern is machined. The reference coordinate SA is reached. The adjacent plane surface has to be programmed outside the cycle.

If **separate machining** is selected, the undercut starting point is approached in rapid traverse, with the equidistant being opened and by complying with a short safety distance of 0.1 mm in radial direction. Thereafter, the undercut contour is machined with feed. Depending on the starting point selected prior to the cycle in relation to position and orientation, the plane surface is turned until the radial starting position is reached. Finally, the tool is returned to the starting position in rapid traverse, by deselecting the edge radius compensation. If separate undercut machining is selected, the tool edge radius correction – if active prior to the cycle – is stopped and is not activated again at the end of the cycle.



The cycle deactivates the global rounding (RND, RNE) and global chamfering (CHS, CHL) and does not activate these parameters again at the end of the cycle.

Example 1:

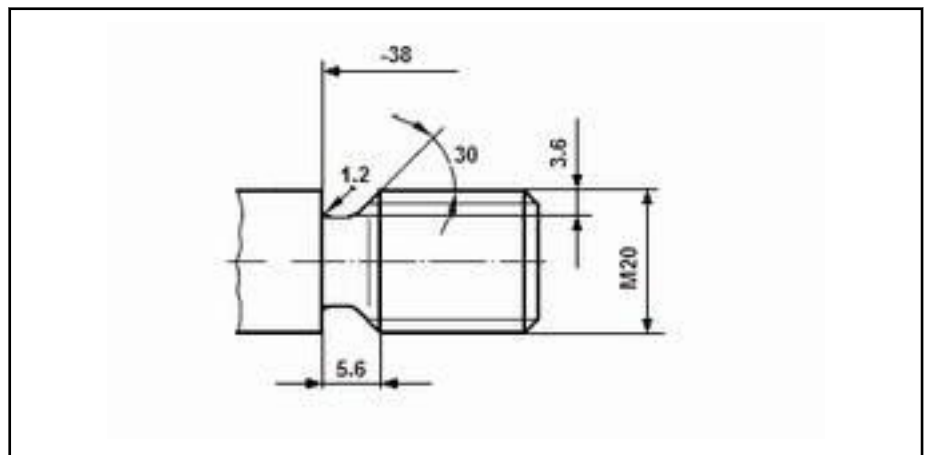


Fig. 10-36: Example of G196 Thread undercut – Free contour, separate machining

Program 1 for G196 Thread undercut turning

```

N10 T1 M6 ;Tool change (SR M6 is required)
N20 G18 DIA G0 X85 Y10 Z100 S250 M3 ;Prepositioning
N30 G94 G97 F100 ;Feed settings
N40 G0 Z-30 X30 ;Safety position prior to cycle
N50 G196(ORI3, DN20, SA-38, DG16.4, WL5.6, R1.2) ;Cycle
N60 G0 X200 Z10 ;Leave contour

```

Turning cycles

Example 2:

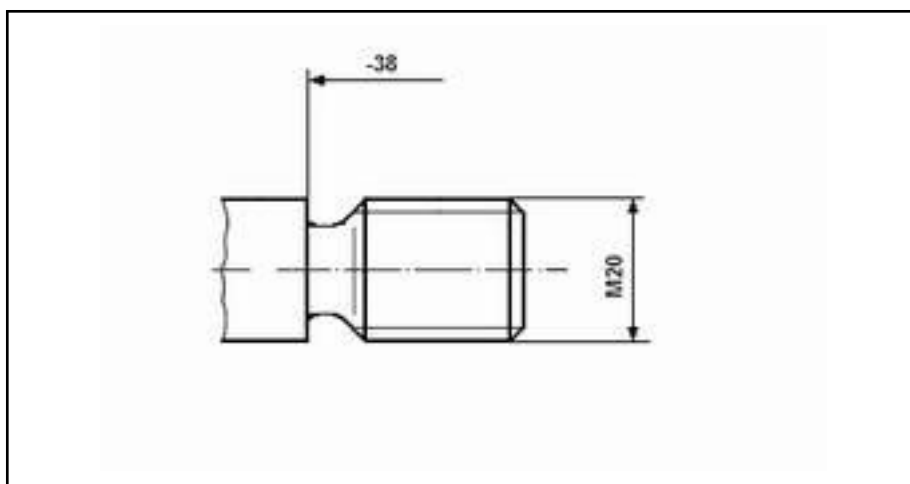


Fig. 10-37: Example of G196 Thread undercut DIN 76 Part 1, machining inside a contour

Program 2 for G196 Thread Undercut Turning

N10 T1 M6	;Tool change (SR M6 is required)
N20 G18 DIA G0 X85 Y10 Z100 S250 M3	;Prepositioning
N30 G94 G97 F100	;Feed settings
N40 G0 G42 Z2 X20	;Safety position in front of cycle
N50 G196(ORI3, ART1, FORM A, SA-38, TP2.5)	;Cycle
N60 G1 X30	;Plane surface
N60 G0 G40 X200 Z10	;Leave contour

10.12 G197 Undercut Turning

The cycle "G197 Undercut turning" can machine undercuts according to DIN 509 or with user-defined dimensions. Undercuts can be integrated in a contour as a transitional element or turned separately (including approach and run-out from the starting point). Machining is always oriented so that the contour extends from the axial surface towards the adjacent plane surface.

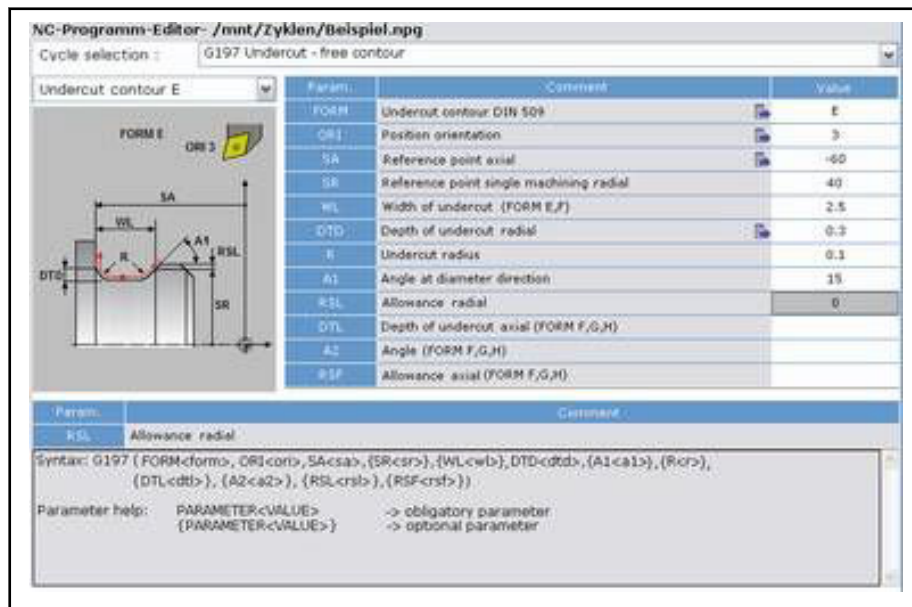


Fig. 10-38: G197 Undercut – Free contour

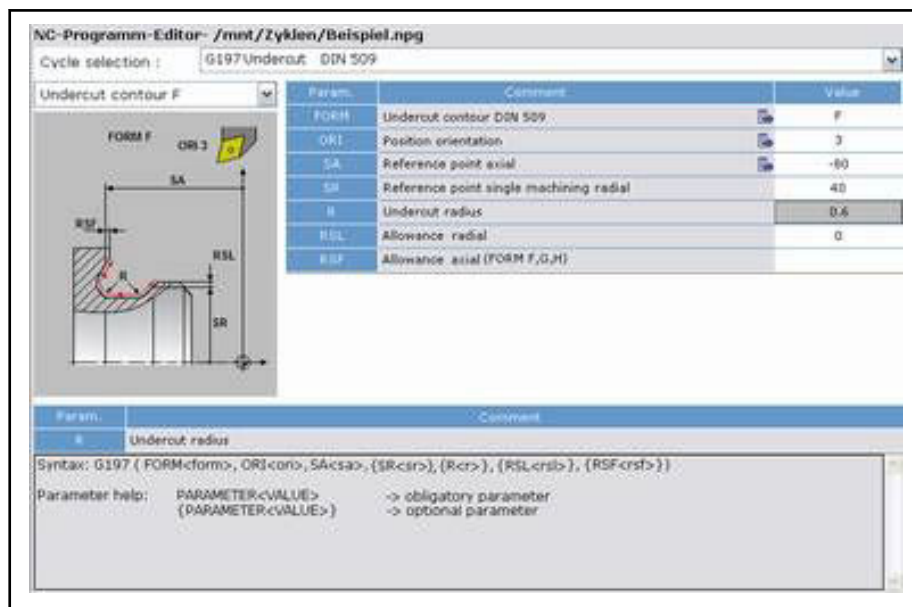


Fig. 10-39: G197 Undercuts DIN 509

G197		Undercuts	G197(ART,FORM,ORI,SR,SA,WL,DTD,R,A1,RSL,DTL,A2,RSF)		
TYPE	SD.SysCyc.ART	Type of undercut	Integer	optional	0<=ART<=1 0 (free form) is pre-assigned
FORM	SD.Sy-sCyc.FORM	Undercut contour acc. to DIN 509	String	Mandatory for free contour (ART0)	E,F,G,H

Turning cycles

G197		Undercuts	G197(ART,FORM,ORI,SR,SA,WL,DTD,R,A1,RSL,DTL,A2,RSF)		
ORI	SD.SysCyc.ORI	Position orientation	Integer	Mandatory	$1 \leq \text{ORI} \leq 4$
SR	SD.SysCyc.SR	Reference coordinate, single machining radial	Real	optional	Absolute
SA	SD.SysCyc.SA	Reference coordinate, axial	Real	Mandatory	Absolute
WL	SD.SysCyc.WL	Width, axial	Real	Mandatory for FORM E, F and ART0	$\text{WL} > 0$
DTD	SD.SysCyc.DTD	Depth, radial	Real	Mandatory for free contour (ART0)	$\text{DTD} > 0$
R	SD.SysCyc.R	Undercut radius	Real	optional	$R > 0$ Default in case of ART1 according to row 1 (assignment to diameter with usual load) 0 is pre-assigned in case of ART0.
A1	SD.SysCyc.A1	Plunging angle	Real	Optional; is only allowed for free contour (ART0)	$5 \leq A1 \leq 30$ Plunging angle pre-assigned according to DIN
RSL	SD.SysCyc.RSL	Allowance, radial	Real	optional	$\text{RSL} \geq 0$ 0 is pre-assigned
DTL	SD.SysCyc.DTL	Depth, axial	Real	Mandatory for free contour (ART0) and FORM F, G, H	$\text{DTL} > 0$
A2	SD.SysCyc.A2	Angle	Real	Mandatory for free contour (ART0) and FORM F, G, H	$5 \leq A2 \leq 30$
RSF	SD.SysCyc.RSF	Allowance, axial	Real	optional Only allowed for FORM F, G, H	$\text{RSF} \geq 0$ 0 is pre-assigned

Tab. 10-33:

Parameter features: The cycle can be used in **every interpolation plane**, although turning is usually performed in plane G18. The main axis of the plane is the axial axis and the secondary axis is the radial axis. If the cycle is for once called in G19, the z-axis is the axial axis and the y-axis is the radial axis.

With active G71, all dimensions have to be programmed in mm, else in inches, with the radius dimension being applicable to all dimensions except for the coordinates of reference points SA and SR.

If the thread undercut is called inside a contour, the radial reference coordinate is obtained from the radial position that was the last to be reached (coordinate of the secondary axis or y-coordinate in G19). This means that a cylinder surface is assumed as initial contour element. To allow turning the undercut separately from a contour, the **radial reference coordinate SR** has to be specified. This coordinate is interpreted as diameter or radius, depending on the scaling of the radial axis. If the undercut is in the range of negative radial coordinates, SR can have a negative sign.



If the SD data structure is used for programming, 0 has to be assigned to SD.SysCyc.SR, if necessary, to obtain a blank assignment.

The **reference coordinate axial SA** specifies the absolute position of the adjacent plane surface. The current axis scaling DIA/RAD has to be taken into account (particularly in G17).

The parameter **ART** refers to one of the three parts of DIN 76 for unit definition or the unit specification is activated via this parameter.

The following specifications can be made:

ART0 User-defined dimensions (default)

ART1 Dimensions according to DIN 509

The parameter **ORI** describes the **orientation of the undercut** which, along with the sign of the radial reference coordinate, allows a conclusion as to whether an internal thread or an external thread is machined by calling this cycle.

The possible values correspond to the edge position code of the tool used to machine the undercut with the orientation of the same name:

ORI1 Contour pattern coming from the negative axial direction and continued in negative radial axis direction

ORI2 Contour pattern coming from the positive axial direction and continued in negative radial axis direction

ORI3 Contour pattern coming from the positive axial direction and continued in positive radial axis direction

ORI4 Contour pattern coming from the negative axial direction and continued in positive radial axis direction

The parameter **FORM** defines **the basic contour of the undercut according to DIN 509**. This parameter is also required for user-defined dimensions (ART0).

Possible specifications for the parameter FORM are:

FORM E Undercut for a cylinder surface to be machined further

FORM F Undercut for a plane surface and cylinder surface to be machined further

FORM G Undercut for a small transition

FORM H Undercut for a transition with stronger rounding

Radius R defines the internal curve. If user-defined dimensioning (ART0) is selected, the current tool edge radius is applied as rounding radius by default. If dimensioning according to DIN is selected, the radius can be specified to address the tabular values of row 2 for increased change stability (except for FORM G).

The **width WL** and the **radial depth DTD** have to be specified for all FORM contours if user-defined dimensioning (ART0) is selected.

The **axial depth DTL** is also required for user-defined dimensioning (ART0), however, only for contours FORM E and F.

The **Plunging angle A1** can be programmed optionally if user-defined dimensioning (ART0) is selected. According to DIN 509, the default value is 15°; for FORM G, it is 17.5°.

The **angle A2** can be programmed optionally if user-defined dimensioning (ART0) is selected. It defines the recess angle in relation to the plane sur-

face. According to DIN 509, the default value is $F 8^\circ$ for FORM F; for FORM G and H, it is 17.5° . A2 is ignored for FORM E.

The **radial allowance RSL** is used to optionally define a machining allowance on the previous cylinder surface (default 0). This parameter causes the plunging distance to be extended in tangential direction. If the undercut is integrated into a contour, the radial reference point is offset accordingly.

The **axial allowance RSF** can be used to optionally define a machining allowance on the adjacent plane surface. It causes the plane surface to be offset in relation to reference coordinate SA, if the undercut is called in a contour. If separate undercut machining is selected, this parameter extends the run-out. RSF is ignored for FORM E.

Sequence: If the undercut is called in a contour (SR is not programmed), a linear interpolation in parallel to the main axis is executed with the active feed until the internally calculated undercut starting point is reached. The edge radius correction has to be activated prior to the cycle. The calculated undercut pattern is machined. The adjacent plane surface has to be programmed outside the cycle.

If **separate machining** is selected, the undercut starting point is approached in rapid traverse, with the edge radius compensation and by complying with a short safety distance of 0.1 mm in radial direction. Thereafter, the undercut contour is machined with feed. Depending on the starting point selected prior to the cycle in relation to position and orientation, the plane surface is now turned until the radial starting position is reached. This also applies to FORM E. Finally, the tool is returned to the starting position in rapid traverse, with the equidistant being closed. If separate undercut machining is selected, the tool edge radius correction – if active prior to the cycle – is stopped and is not activated again at the end of the cycle.



The cycle deactivates the global rounding (RND, RNE) and global chamfering (CHS, CHL) and does not activate these parameters again at the end of the cycle.

Example 1:

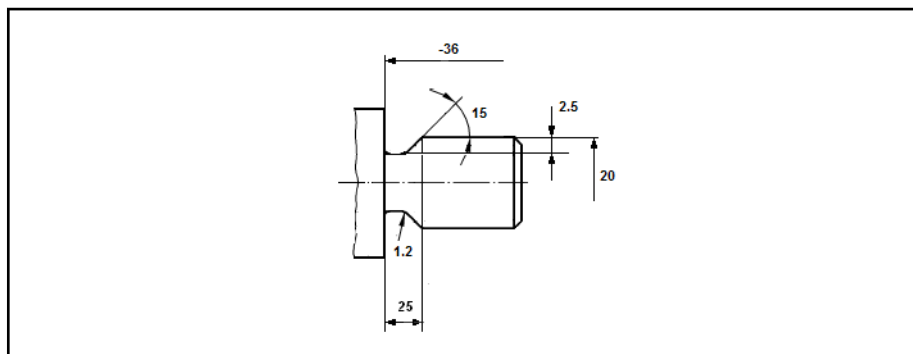


Fig. 10-40: Example of G197 Undercut – Free contour, separate machining
Program 1 for G197 Undercut Turning

```

N10 T1 M6                                ;Tool change (SR M6 is required)
N20 G18 DIA G0 X85 Z10 S250 M3           ;Prepositioning
N30 G94 G97 F100                         ;Feed settings
N40 G0 Z-33.5 X30                        ;Safety position prior to cycle
N50 G197(ART0, FORM E, ORI3,SA-36, SR20, ;Cycle
WL25, DTD2.5, R1.2, A1 15);
N60 G0 X50 Z100                          ;Leave contour

```

Example 2:

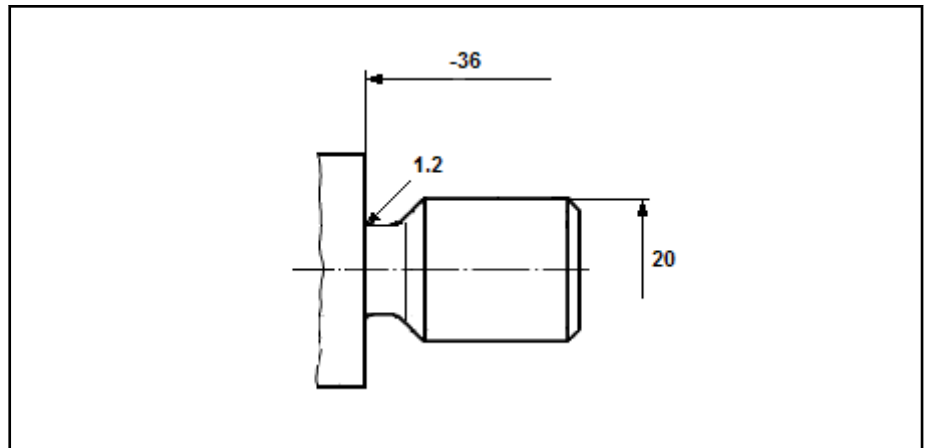


Fig. 10-41: Example of G197 Undercut DIN 509, machining inside the contour
 Program 2 for G197 Undercut turning

```

N10 T1 M6                                ;Tool change (SR M6 is required)
N20 G18 DIA G0 X85 Z10 S250 M3           ;Prepositioning
N30 G94 G97 F100                         ;Feed settings
N40 G0 G42 Z2 X20                        ;Safety position prior to cycle
N50 G197(ART1, FORM E, ORI3, SA-36, R1.2);Cycle
N60 G1 X30
N60 G0 G40 X50 Z100                      ;Leave contour
  
```

11 Turning cycles (contour-dependent)

11.1 General notes

NOTICE

If technology parameters are changed outside the program the cycle is programmed in, save the program again.

The contour-dependent cycles create a cut segmentation and use it again if the cycle parameters were not changed. When saving the program, all old cut segmentations are discarded and newly once created if required.

11.2 Defining the Contour

The contour is defined by the starting and end blocks, the starting and end labels or by a subroutine. Exactly one contour has to be defined in each of the contour turning cycles.

If the starting and end blocks are specified, the integers have to be entered exactly as given in the program. This means that the leading zeros may not be omitted.

Example: *Program:*

```
N0 G170 (S01, E005)
1 GOTO .END_OF_CONT1 ;skip contour in program sequence
; contour definition:
N01      ;start of contour
N02 .... ;enter contour here
N005     ;end of contour
10 .END_OF_CONT1
```

If the contour is defined by a subroutine, there has to be a space or an "=" between P and the program name (for a better readability). The program name may include a path. Neither the subroutine name nor the path may contain spaces, brackets or commas.

Example: *Program:*

```
N0 G170 (P Kontur1.npg)
N1 G170 (P=/mnt/Test/Kontur.npg)
```

The following applies to all contour turning cycles (G170-G174):

- The contour definition may not contain any internal contour cycle calls (G170-G174).
- Starting and end block or labels have to be in the same program as the cycle call.
- The contour definition may contain subroutines (e.g., cycles, such as undercuts, G168).
- The contour may not exceed 50 elements including an input tool (RND, CHL, CHS, straight moves).
- The number of pockets (undercuts) is limited to 10.
- If the contour is defined by starting and end block and if the contour is to be exclusively used as contour definition for a contour-dependent cycle, the contour has to be skipped in the parts program using a branch instruction to avoid that the blocks are processed as regular NC blocks within a program sequence. Optionally, the contour can be defined after the program return (M30).

Turning cycles (contour-dependent)

The following applies in particular to contour roughing cycles (G171-G174):

- Within the contour definition, only the geometry information (straight line, circular and starting and end points) is evaluated. Feeds, M-functions etc. are ignored.
- All motions outside the selected plane are ignored.



The contour is read in and processed at first run. It is then assumed as constant. Changing contours are not supported.



No rounding (e.g. RND) is to be enabled in the starting block of the contour. Roundings modify the path and thus the defined contour. Thus, undesired effects can result.



The number of blocks in the block preparation (machine parameter NCO/LookAh/Ch[]/NofBlkPrep) has to be at least greater by five than the number of blocks in the contour description. Roundings and chamfers create additional blocks.

11.3 G170 Contour finishing

The cycle "Contour Finishing" can machine a turning contour with one cut.

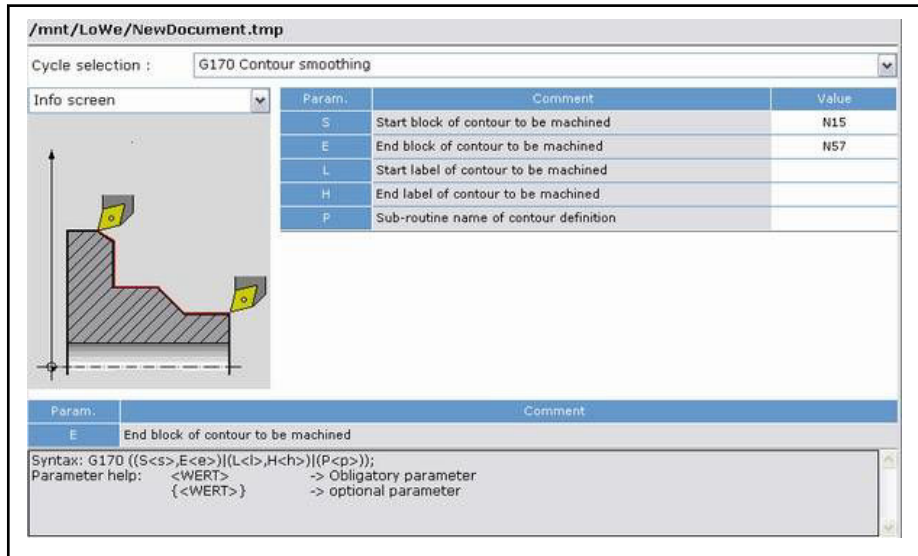


Fig. 11-1: G170 Contour finishing

G170	Contour finishing	G170(S,E,L,H,P)	
S	Starting block of the contour to be machined	STRING	optional
E	End block of the contour to be machined	STRING	optional
L	Starting label of the contour to be machined	STRING	optional
H	End label of the contour to be machined	STRING	optional
P	Subroutine name of the contour definition	STRING	optional

The contour definition is described in [chapter 11.2 Defining Contour, page 251](#).

Sequence: The cycle uses all axes programmed in the contour definition irrespective of the level. The contour to be machined can either be specified via the starting

block/end block or via the starting label/end label or via the subroutine name of the subroutine specifying the contour. One of these entries is required. It has to be possible to travel to the starting point of the contour straightforwardly from the last programmed position.

The cycle acts as a GOTO with return jump or as a subroutine call.

NOTICE

A contour description can be used for roughing cycles and for contour finishing G170 at the same time. Compared to other contour cycles, all blocks of the contour are normally processed at contour finishing G170. This means that auxiliary functions become active, modal states are switched and a programmed feed or speed is applied. Additionally, the cycle (in contrast to other contour cycles) does not return to the starting point but is only left at the end of the contour.

If the contour description is used for roughing and contour finishing G170, it has to be ensured that all feeds, speeds and auxiliary functions are correct for contour finishing. These specifications are not applicable for the other contour cycles since these read only geometry information.

The process is performed in one cut along the contour.

Example:

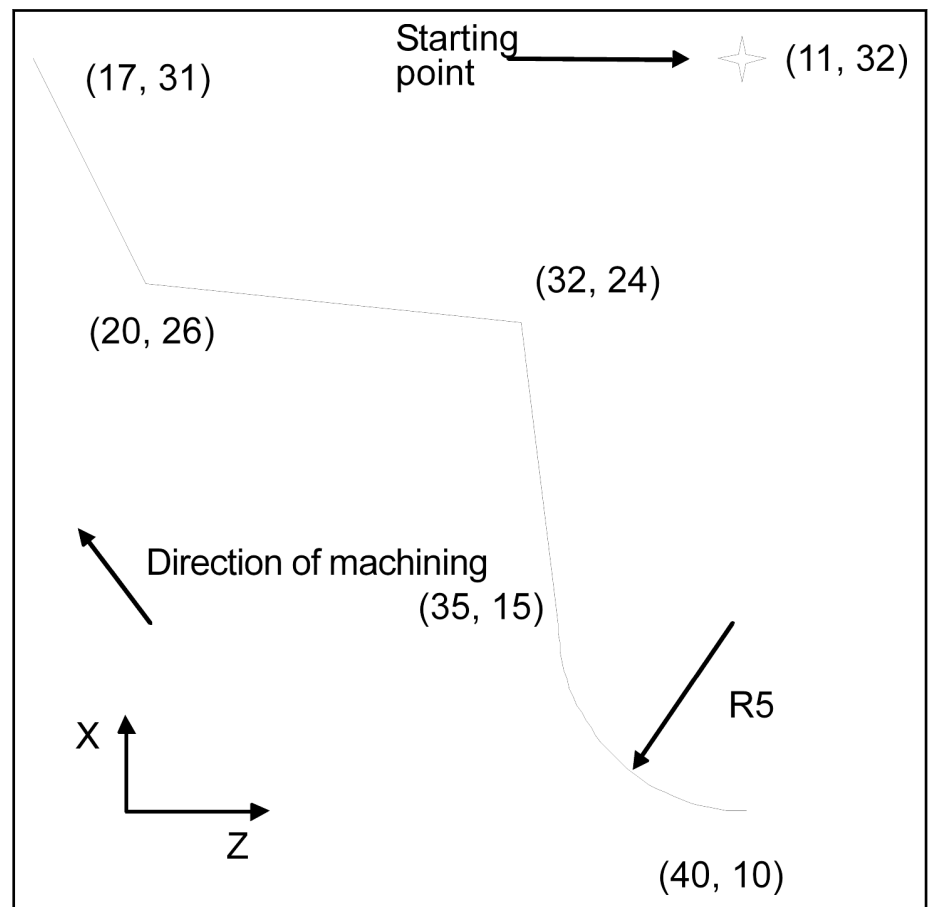


Fig. 11-2: G170 Contour finishing

Turning cycles (contour-dependent)

Program for G170 Contour finishing

```

N10 G18 G94 F650 ;Initialization
N20 G0 Z50 X32 ;Approach starting position
N30 G170 (S100, E140) ;Call cycle
N40 M30 ;Program end

;*****
;*** Contour description ***
;*****
N100 Z40 X10
N110 G2 Z35 X15 R5
N120 G1 Z32 X24
N130 Z20 X26
N140 Z17 X31

```

11.4 G171 Contour roughing, longitudinal

The cycle "Contour roughing, longitudinal" can machine a turning contour with multiple longitudinal axis-parallel cuts.

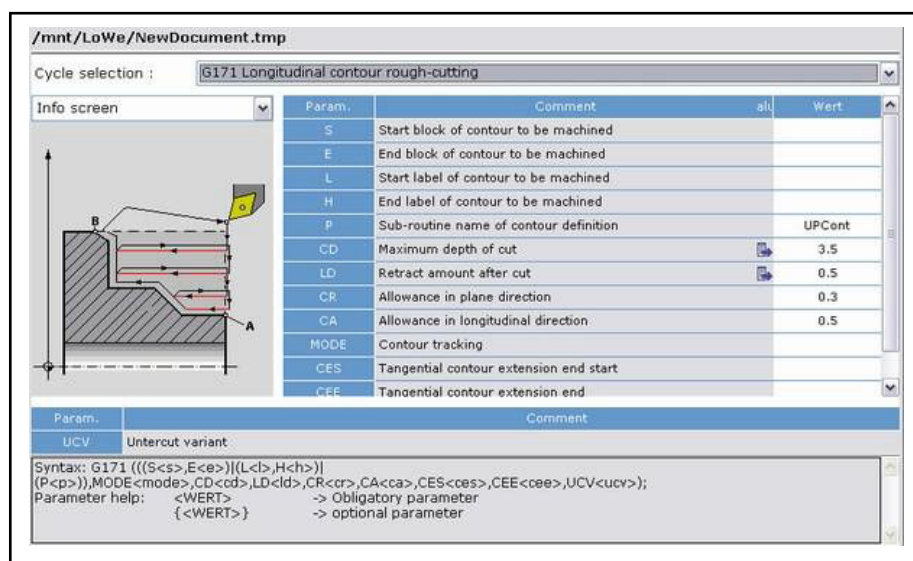


Fig. 11-3: G171 Contour roughing, longitudinal

G171	Contour Roughing, Longitudinal	G171(S,E,L,H,P,CD,LD,CR,CA,RSL,MODE,CES,CEE,UCV)	
S	Starting block of the contour to be machined	STRING	optional
E	End block of the contour to be machined	STRING	optional
L	Starting label of the contour to be machined	STRING	optional
H	End label of the contour to be machined	STRING	optional
P	Subroutine name of the contour definition	STRING	optional
CD	Maximum depth of cut	REAL	Mandatory
LD	Retract amount after cut	REAL	optional
CR	Allowance in plane direction	REAL	optional
CA	Allowance in longitudinal direction	REAL	optional
RSL	Equidistant allowance	REAL	optional
MOD	Contour tracking	INTEGER	optional
CES	Tangential approach	REAL	optional

G171	Contour Roughing, Longitudinal	G171(S,E,L,H,P,CD,LD,CR,CA,RSL,MODE,CES,CEE,UCV)	
CEE	Tangential run-out	REAL	optional
UCV	Undercut and recess variant	INTEGER	optional

The contour definition is described in [chapter 11.2 Defining Contour, page 251](#).

Sequence: The cycle operates in the selected plane with main axis and secondary axis. It has to be possible to travel to the starting point of the cycle (refer to the figure) straightforwardly from the last programmed position.

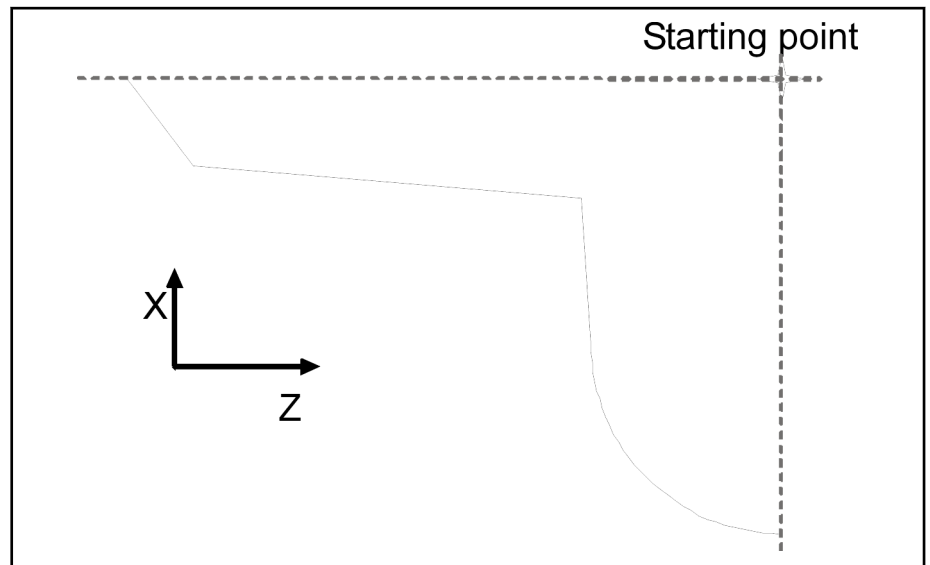


Fig. 11-4: Sequence of G171 basic



If the contour starts with a vertical straight line in X+, the first contour element is ignored. Machining starts from the first non-vertical element. A programmed contour extension at the start of the contour CES starts from this first non-vertical element.

If no contour extension is specified, the starting point is moved by 2mm to Z+ in order to avoid collisions during the infeed.

The machining is performed in multiple, axis-parallel cuts in longitudinal direction. The maximum depth of cut of the individual cuts is specified via CD. Due to the cycle, all cuts are of the same depth.

A specified allowance remains on the contour following the machining. MOD contour tracking is used to define whether - deviating from axis-parallel machining - the contour is machined with an additional contour cut at the end of each cut or only at the end of the machining process.

Turning cycles (contour-dependent)

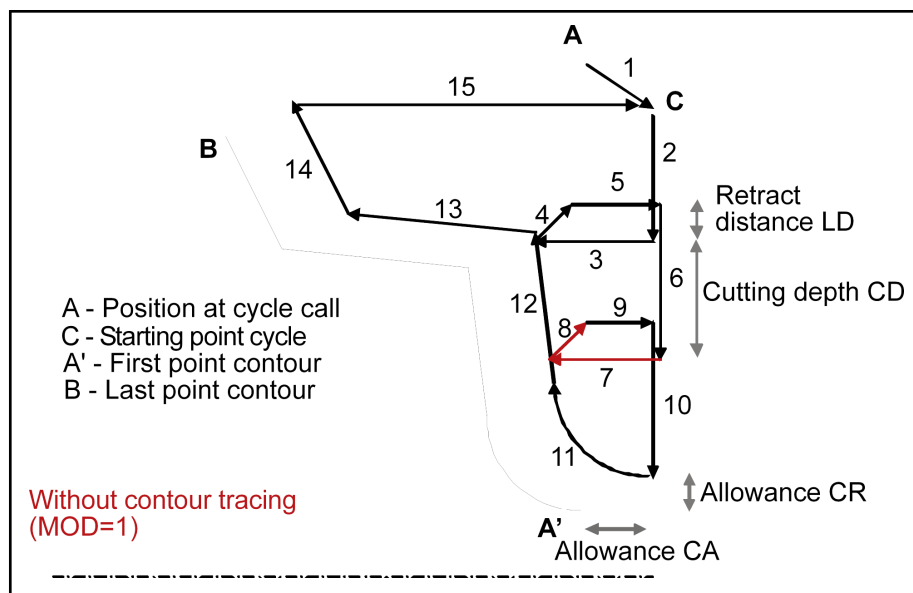


Fig. 11-5: Sequence of G171 without contour tracking

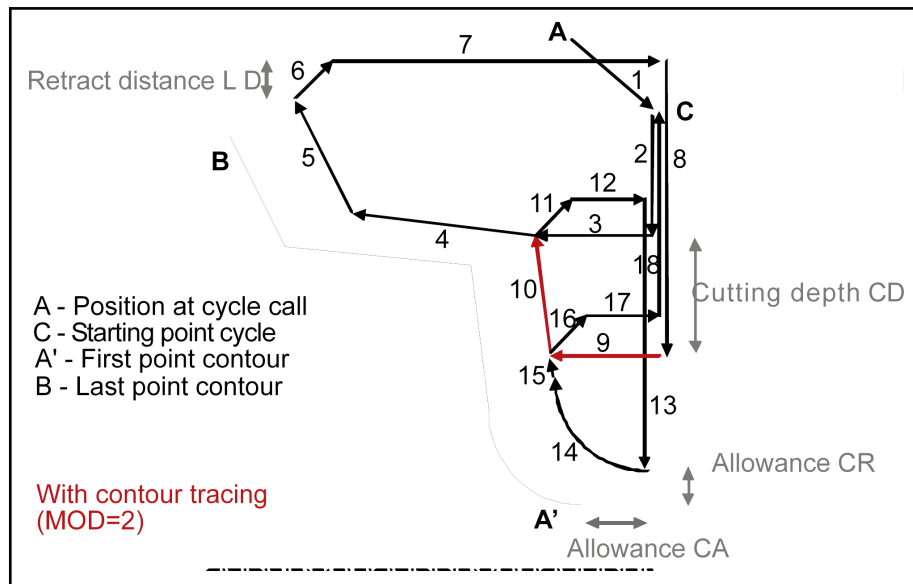


Fig. 11-6: Sequence of G171 without contour tracking

If no contour tracking mode MOD is specified, the cycle operates without contour tracking on a contour without undercuts and with contour tracking on a contour with undercuts.

The undercut variant and the recess variant determine whether these (e.g., undercuts) are to be machined.

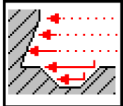
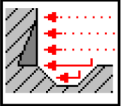
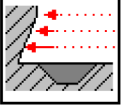
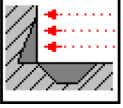
UCV0	Machining Undercut and recess (default)	
UCV1	Hide recess	
UCV2	Hide undercut	
UCV3	Hide undercut and recess	

Fig. 11-7: Sequence of G171 cutting variants

For undercuts, individual pockets are roughed up to the defined allowance and then the remaining contour is machined.

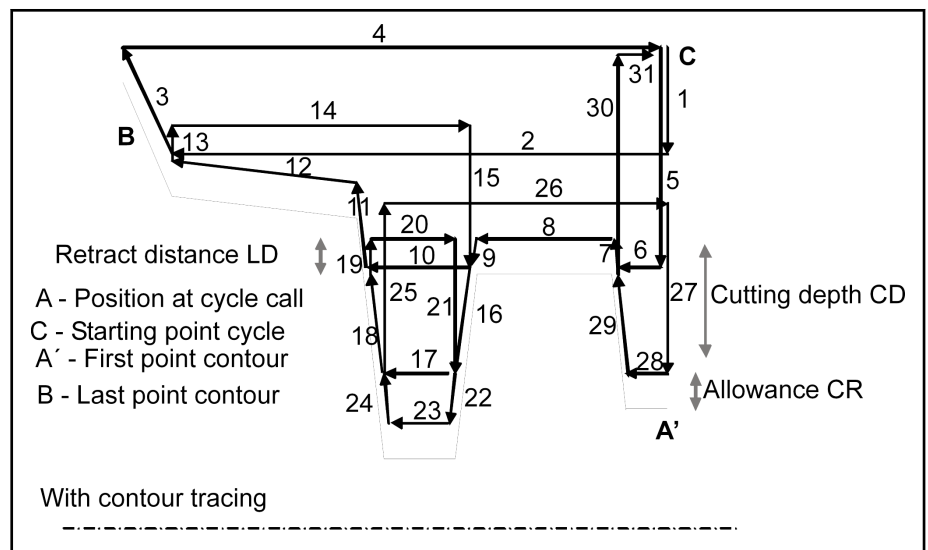


Fig. 11-8: Sequence of G171 pockets

Machining can be appropriately lengthened by means of the tangential approach and run-out.

Turning cycles (contour-dependent)

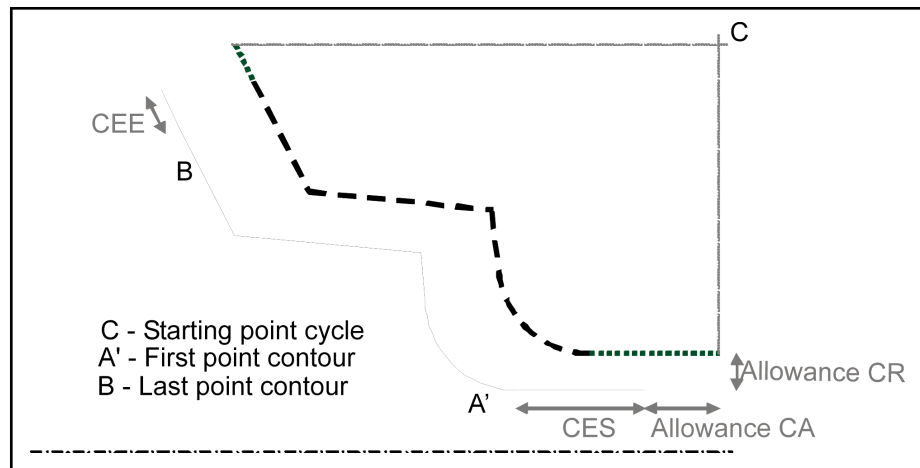


Fig. 11-9: Sequence of G171 tangential approach and run-out

The angle between a rear panel of a recess and the main axis may not be lower than 45° (see figure).

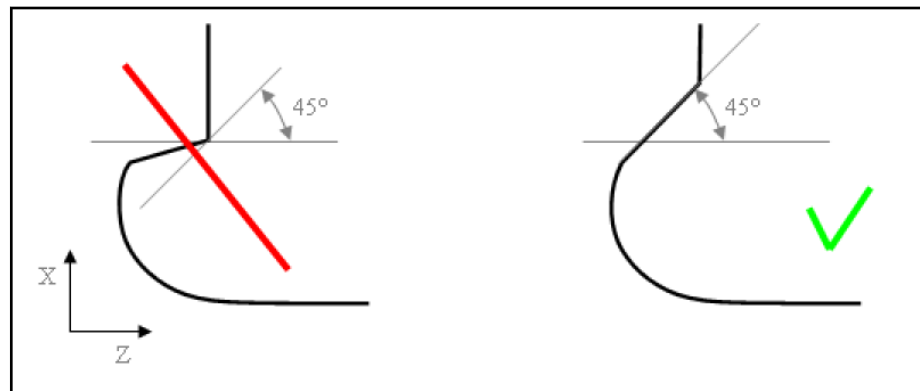


Fig. 11-10: Restriction of the recess

If this condition of the contour specified is not met, the MTX generates a run-time error.

Example:

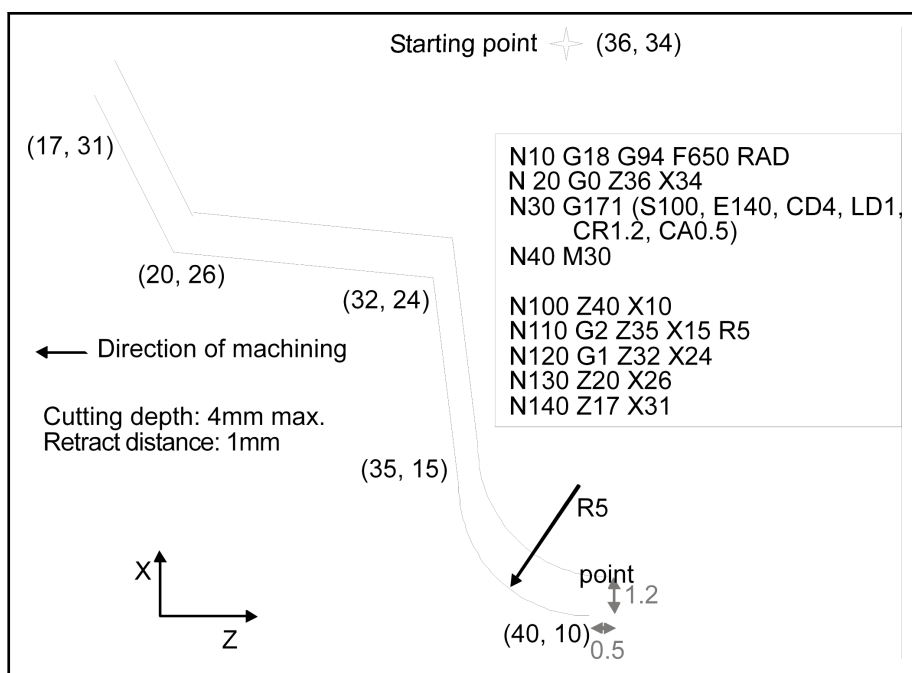


Fig. 11-11: Example of G171 Contour roughing, longitudinal

Program for G171 Contour roughing, longitudinal

```

N10 G18 G90 G94 F650 RAD ;Initialization
N20 G0 Z36 X34 ;Approach starting point
N30 G171 (S100,E140,CD4,LD1,CR1.2,CA0.5) ;Call cycle
N40 M30 ;Program end

;*****
;*** Contour description ***
;*****
N100 Z40 X10
N110 G2 Z35 X15 R5
N120 G1 Z32 X24
N130 Z20 X26
N140 Z17 X31

```

11.5 G172 Contour roughing, plane

The cycle "Contour Roughing, Plane" can machine a turning contour with multiple, axis-parallel plane cuts.

Turning cycles (contour-dependent)

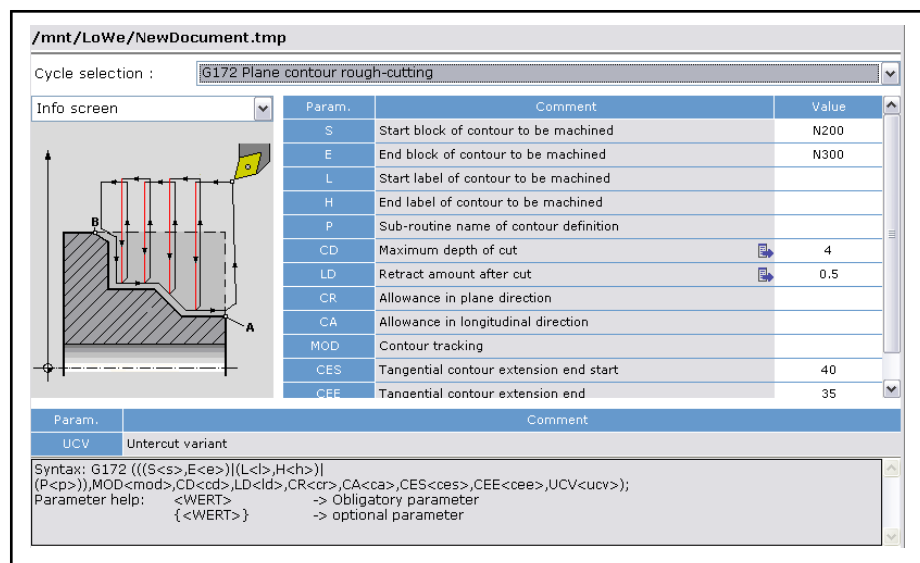


Fig. 11-12: G172 Contour roughing, plane

G172	Contour Roughing, plane	G172(S,E,L,H,P,CD,LD,CR,CA,RSL,MOD,CES,CEE,UCV)		
S	Starting block of the contour to be machined	STRING	optional	
E	End block of the contour to be machined	STRING	optional	
L	Starting label of the contour to be machined	STRING	optional	
H	End label of the contour to be machined	STRING	optional	
P	Subroutine name of the contour definition	STRING	optional	
CD	Maximum depth of cut	REAL	Mandatory	
LD	Retract amount after cut	REAL	optional	Pre-assigned - 0
CR	Allowance in plane direction	REAL	optional	Pre-assigned - 0
CA	Allowance in longitudinal direction	REAL	optional	Pre-assigned - 0
RSL	Equidistant allowance	REAL	optional	Pre-assigned - 0
MOD	Contour tracking	INTEGER	optional	Pre-assigned: contour tracking only for undercuts
CES	Tangential approach	REAL	optional	Pre-assigned - 0
CEE	Tangential run-out	REAL	optional	Pre-assigned - 0
UCV	Undercut and recess variant	INTEGER	optional	Pre-assigned - 0

The definition of the contour is described in [chapter 11.2 "Defining the Contour"](#) on page 251.

Sequence: The cycle operates in the selected plane with main axis and secondary axis. It has to be possible to travel to the starting point of the cycle (refer to the figure) straightforwardly from the last programmed position.

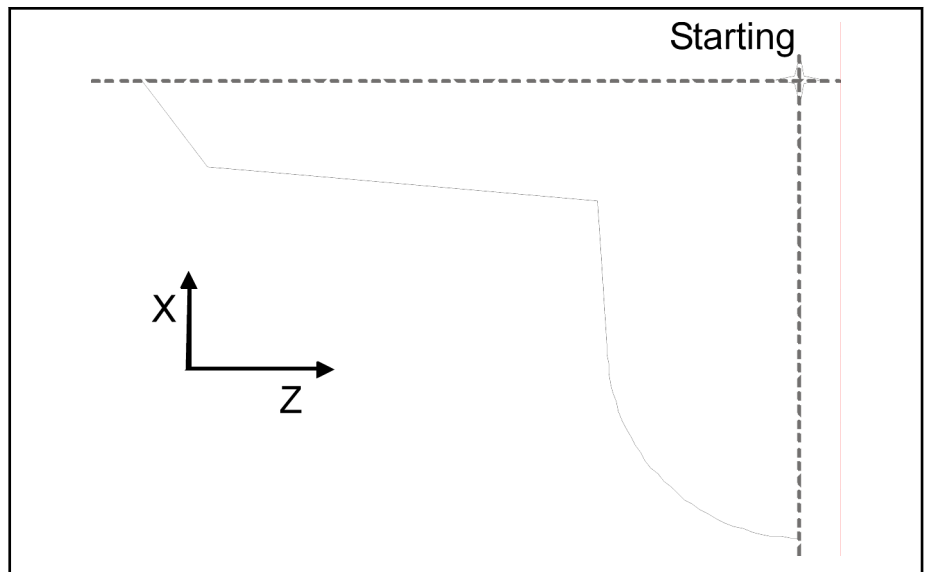


Fig. 11-13: G172 Defining starting point

Machining is performed in multiple, axis-parallel cuts in a plane direction. The maximum depth of cut of the individual cuts is specified via CD. Due to the cycle, all cuts are of the same depth. A specified allowance remains on the contour following the machining.

MOD contour tracking is used to define whether - deviating from axis-parallel machining - the contour is machined with an additional contour cut at the end of each cut or only at the end of the machining process. Thus, the cycle acts like a G171 rotated around 90°.

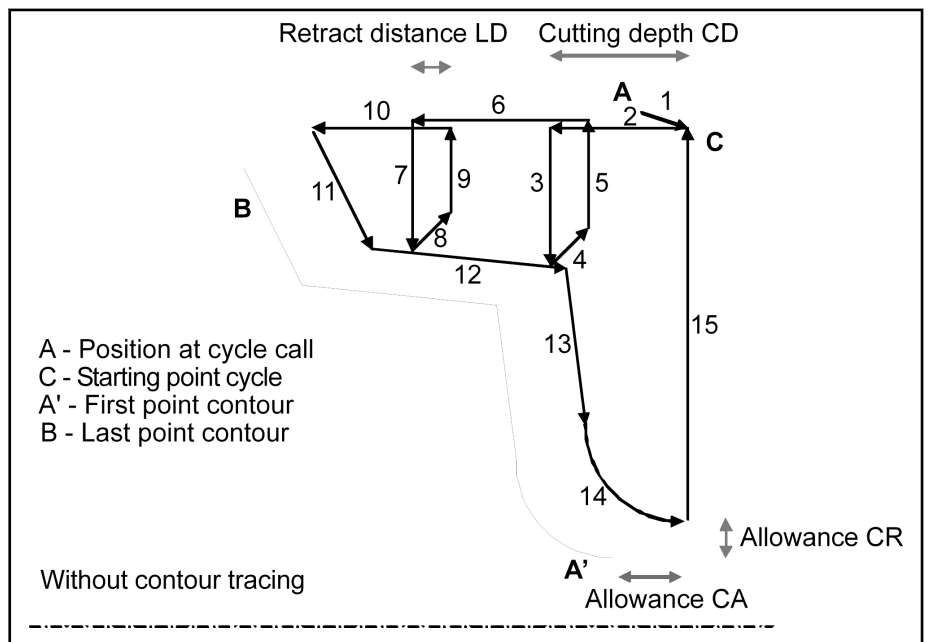


Fig. 11-14: G172 Cut segmentation without contour tracking

If no contour tracking mode MOD is specified, the cycle operates without contour tracking.

The undercut variant and the recess variant determine whether they (e.g., undercuts) are to be machined.

Turning cycles (contour-dependent)

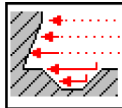
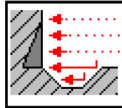
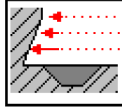
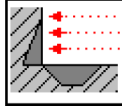
UCV0	Machining undercut and recess (default)	
UCV1	Hide recess	
UCV2	Hide undercut	
UCV3	Hide undercut and recess	

Fig. 11-15: G172 Recess cut and undercut variants

Machining can be appropriately lengthened by means of the tangential approach and run-out.

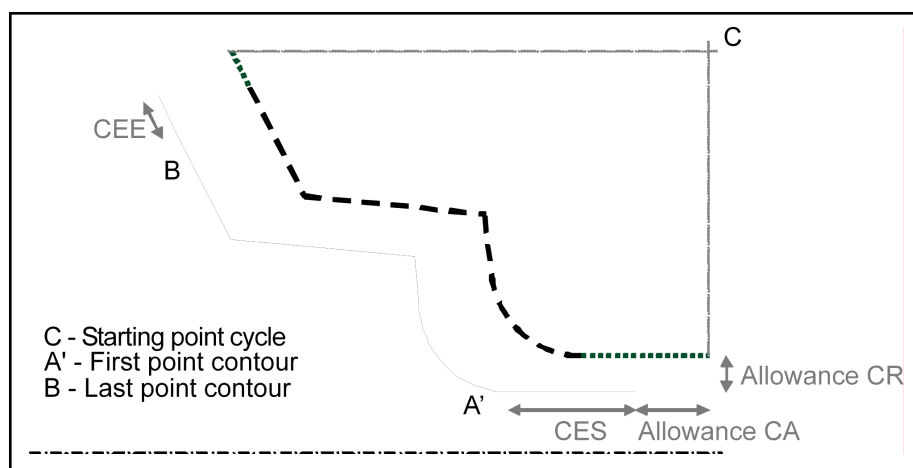


Fig. 11-16: G172 Tangential approach or run-out

Example:

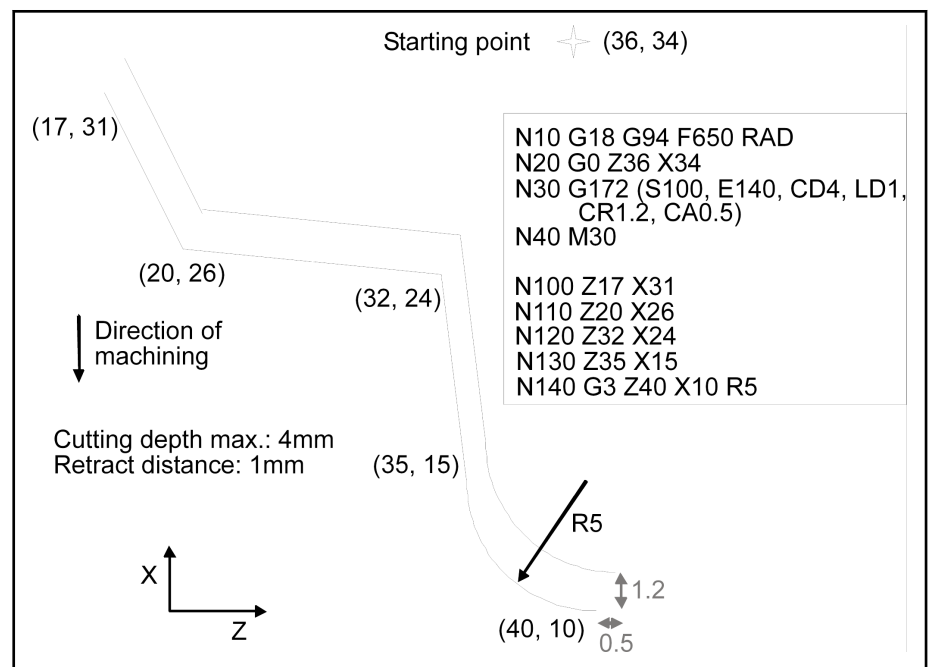


Fig. 11-17: Example of G172 Contour roughing, plane
Program for G172 Contour roughing, plane

```

N10 G18 G90 G94 F650 RAD           ;Initialization
N20 G0 Z36 X34                     ;Approach starting position
N30 G172 (S100, E140, CD4, LD1,    ;Call cycle
CR1.2,CA0.5)
N40 M30                             ;Program end

;*****
;*** Contour description ***
;*****
N100 Z40 X10
N110 G2 Z35 X15 R5
N120 G1 Z32 X24
N130 Z20 X26
N140 Z17 X31

```



The contour in case of G172 may not have recesses. If required, these have to be hidden with the undercut variant (UCV=1).

11.6 G173 Contour roughing, parallel to contour

The cycle "Contour roughing, parallel to contour" can machine a turning contour with multiple cuts parallel to contour.

Turning cycles (contour-dependent)

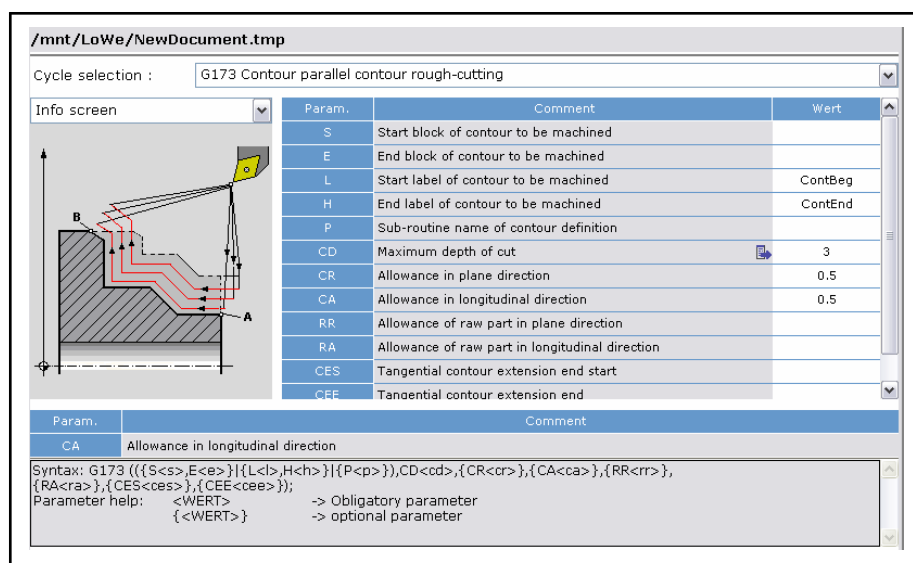


Fig. 11-18: G173 Contour roughing, parallel to contour

G173	Contour Roughing, Parallel to Contour	G173(S,E,L,H,P,CD,CR,CA,RSL,RR,RA,CES,CEE)		
S	Starting block of the contour to be machined	STRING	optional	
E	End block of the contour to be machined	STRING	optional	
L	Starting label of the contour to be machined	STRING	optional	
H	End label of the contour to be machined	STRING	optional	
P	Subroutine name of the contour definition	STRING	optional	
CD	Maximum depth of cut	REAL	Mandatory	
CR	Allowance in plane direction	REAL	optional	Pre-assigned - 0
CA	Allowance in longitudinal direction	REAL	optional	Pre-assigned - 0
RSL	Equidistant allowance	REAL	optional	Pre-assigned - 0
RR	Raw piece: allowance, plane	REAL	optional	Pre-assigned - 0
RA	Raw piece: allowance, longitudinal	REAL	optional	Pre-assigned - 0
CES	Tangential approach	REAL	optional	Pre-assigned - 0
CEE	Tangential run-out	REAL	optional	Pre-assigned - 0

The definition of the contour is described in [chapter 11.2 "Defining the Contour"](#) on page 251.

Sequence: The cycle operates in the selected plane with main axis and secondary axis.

The contour to be machined can either be specified via the starting block/end block or via the starting label/end label or via the subroutine name of the subroutine specifying the contour. One of these entries is required. It has to be possible to travel to the starting point of the contour straightforwardly from the last programmed position.

At least one raw piece allowance (RR, RA) has to be programmed. The sign of the raw piece allowance can be specified whether it is an outer or inner cut or whether it is turned from right or left. If RR and RA are greater than 0, an (as in [fig. 11-19 "G173 Cut segmentation"](#) on page 265) outer cut is ma-

chined from the right. If RR is smaller and RA greater than 0, it is an inner cut from right. Analogously, if RA is smaller than 0, it is machined from left.

Machining takes place in several steps, in parallel to the contour. The maximum depth of cut of the individual cuts is specified via CD. A specified allowance remains on the contour following the machining.

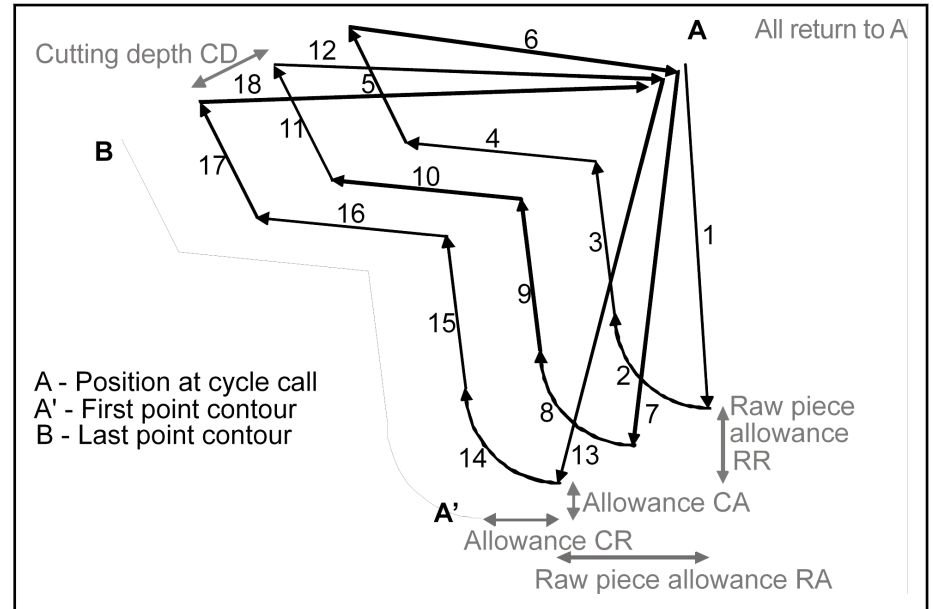


Fig. 11-19: G173 Cut segmentation

Machining can be appropriately lengthened by means of the tangential approach and run-out. The area to be machined can be defined via the raw piece allowances.

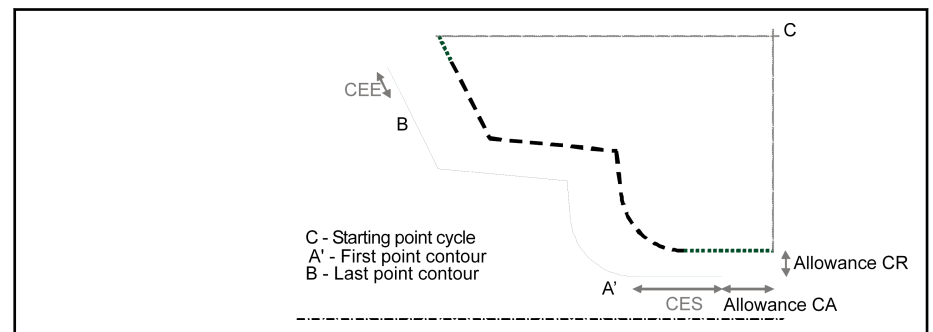


Fig. 11-20: G173 Tangential approach or run-out

Turning cycles (contour-dependent)

Example:

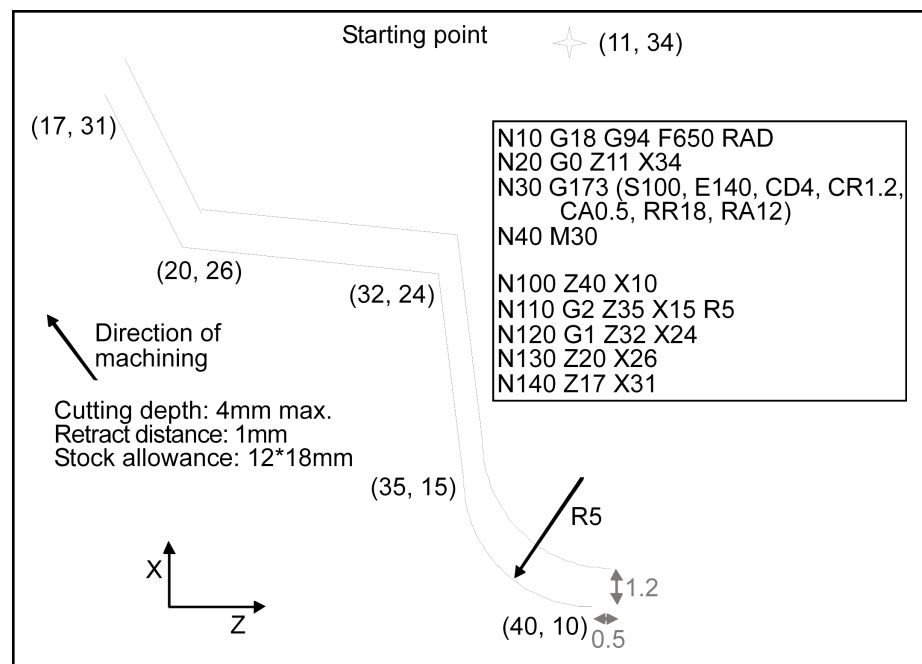


Fig. 11-21: G173 Contour roughing, parallel to contour

Program for G173 Contour roughing, parallel to contour

```

N10 G18 G90 G94 F650 RAD          ;Initialization
N20 G0 Z50 X34                    ;Approach starting position
N30 G173 (S100, E140, CD4, CR1.2, CA0.5, RR18) ;Call cycle
N40 M30                            ;Program end
  
```

```

;*****
;*** Contour description ***
;*****
N100 Z40 X10
N110 G2 Z35 X15 R5
N120 G1 Z32 X24
N130 Z20 X26
N140 Z17 X31
  
```



The modal motion function (G0/G1/G2/G3) following the cycle is not defined and is to be set by the NC program.

11.7 G174 Contour grooving

The cycle "Contour Grooving" can machine a turning contour with multiple, axis-parallel grooves.

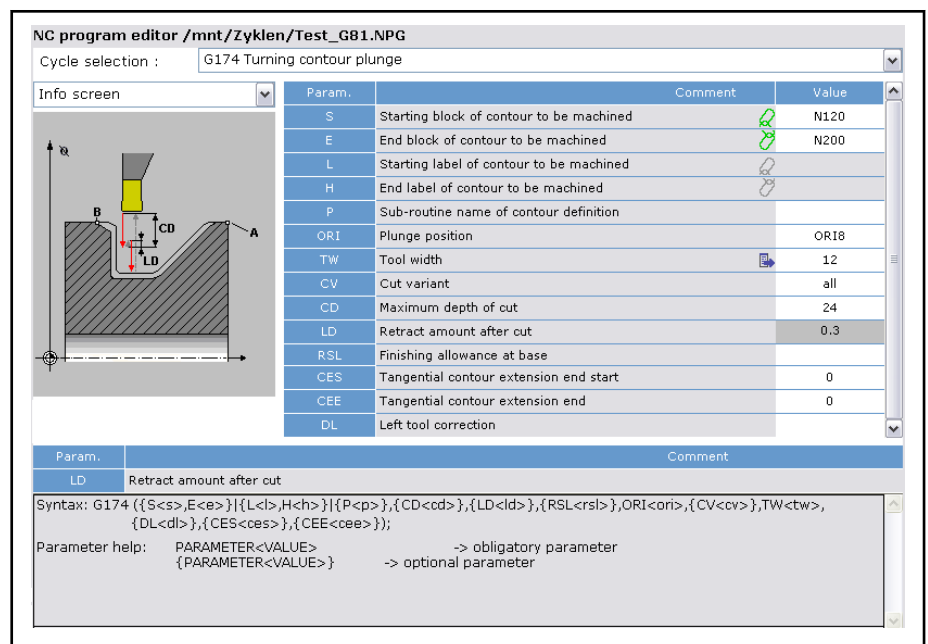


Fig. 11-22: G174 Contour plunging

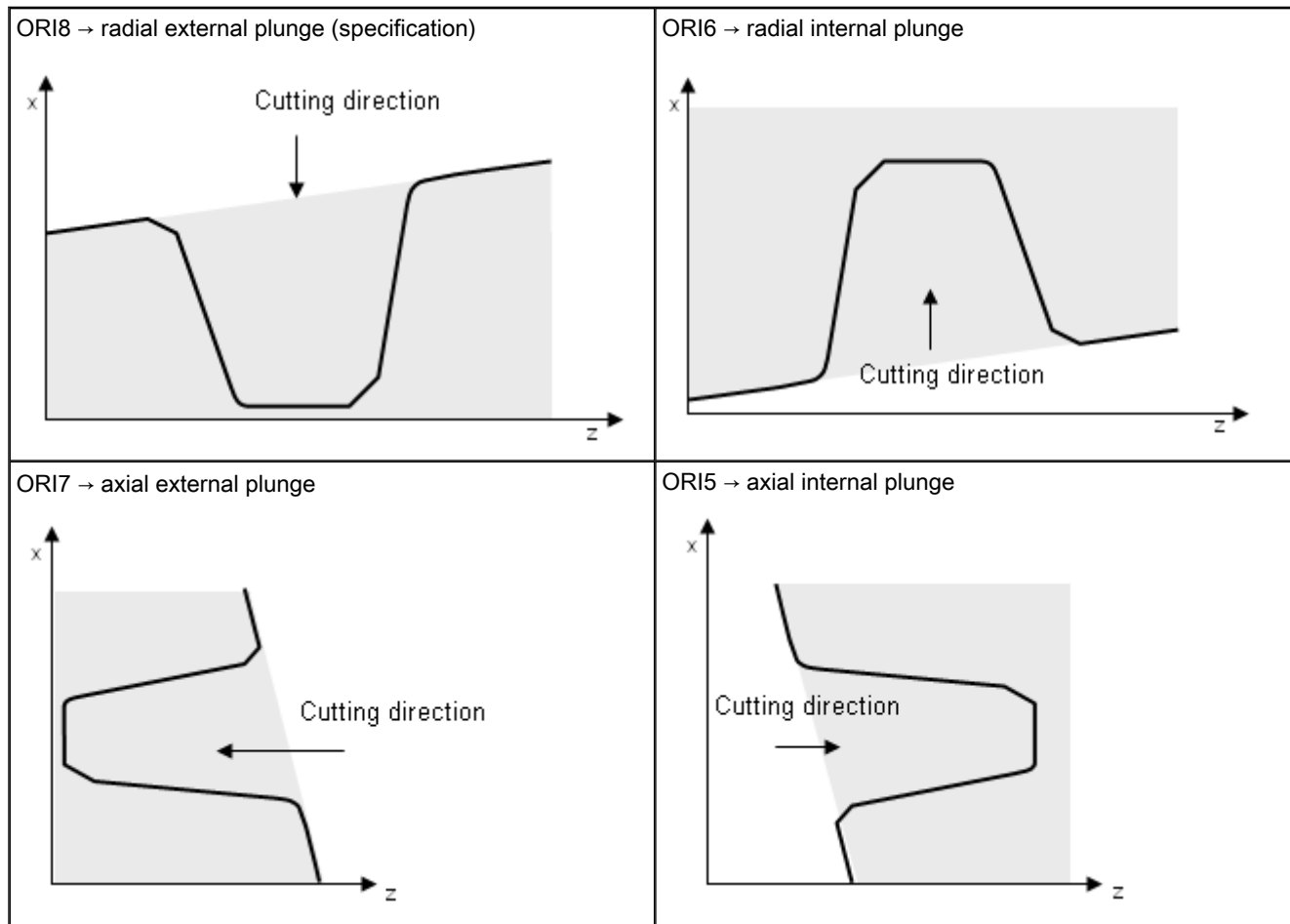
G174	Contour grooving	G174(S,E,L,H,P,TW,CD,LD,RSL,ORI,CES,CEE,CV,DL)	
S	Starting block of the contour to be machined	STRING	optional
E	End block of the contour to be machined	STRING	optional
L	Starting label of the contour to be machined	STRING	optional
H	End label of the contour to be machined	STRING	optional
P	Subroutine name of the contour definition	STRING	optional
TW	Tool width	REAL	Mandatory
CD	Maximum depth of cut	REAL	optional
LD	Retract amount after cut and safety distance	REAL	optional
RSL	Equidistant allowance	REAL	optional
ORI	Orientation of cuts See tab. 11-2 "Plunge orientation ORI" on page 268	INTEGER	optional
CES	Tangential approach	REAL	optional
CEE	Tangential run-out	REAL	optional
CV	Cutting variant	INTEGER	optional
DL	Tool correction for the left side of the plunge	STRING	optional

Tab. 11-1: G174 Contour plunging

The definition of the contour is described in [chapter 11.2 "Defining the Contour" on page 251](#).

Turning cycles (contour-dependent)

Plunge orientation ORI

Tab. 11-2: Plunge orientation **ORI**

Before calling the cycle, activate the tool correction (G41/G42 and G47).

An edge is selected used for the contour cut of the right side. The cycle internally deselects the contour when creating the step profile. It is activated again for the right contour cut.

Sequence: The cycle operates in the selected plane with main axis and secondary axis.

Machining is executed in five steps:

1. Calculate the equidistant contour.
2. Determine the retract plane.
3. Create the step profile
4. Contour cut right
5. Contour cut left

The equidistant contour is determined in the first step. The distance to the original contour is specified by the parameter "RSL".

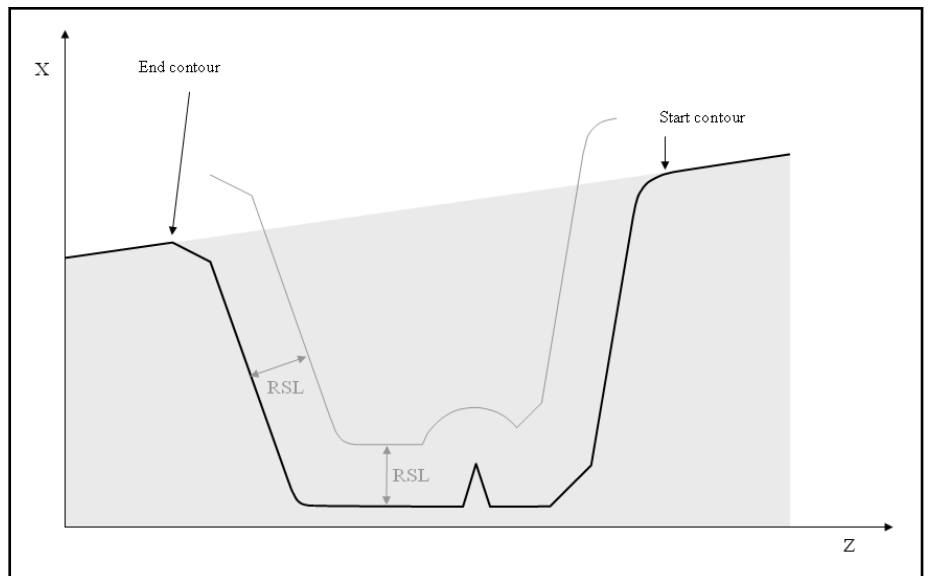


Fig. 11-23: Step 1: Determine equidistant contour

Determine the reference plane. Thus, specify the highest x-value of starting and end point of the contour. The safety distance "LD" is added to maintain the reference plane.

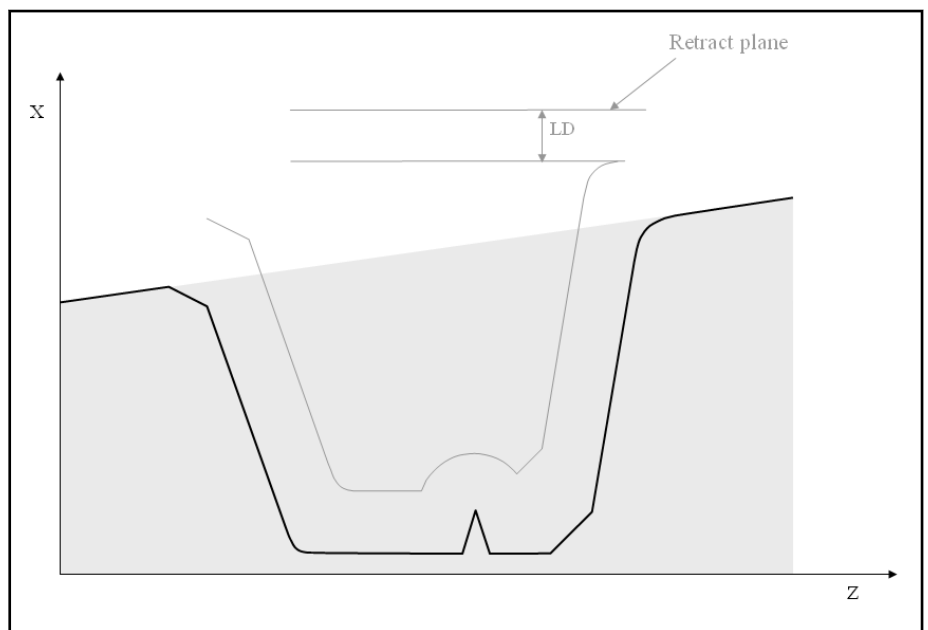


Fig. 11-24: Step 2: Determine reference plane

A step profile is created by axis-parallel grooves. The infeed width corresponds to the parameter "TW" (tool width).

The maximum depth of cut of the individual cuts is specified via "CD". Due to the cycle, all cuts are of the same depth. The tool is retracted after each step by "LD" for chip breaking. If "CD" is not programmed, only one cut per infeed (without retracting) is executed. All cuts are executed without tool radius correction. This is internally switched off.

Turning cycles (contour-dependent)

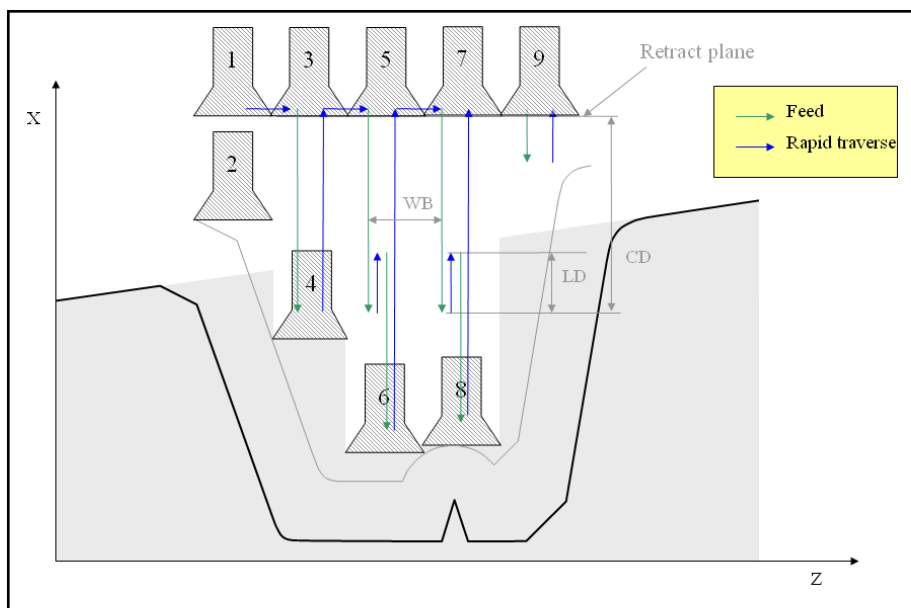


Fig. 11-25: Step 3: Create step profile

Step 4: The right side of the plunge up to the equidistant contour is machined with active tool correction. The correction number (D or ED) active before the cycle call is used.

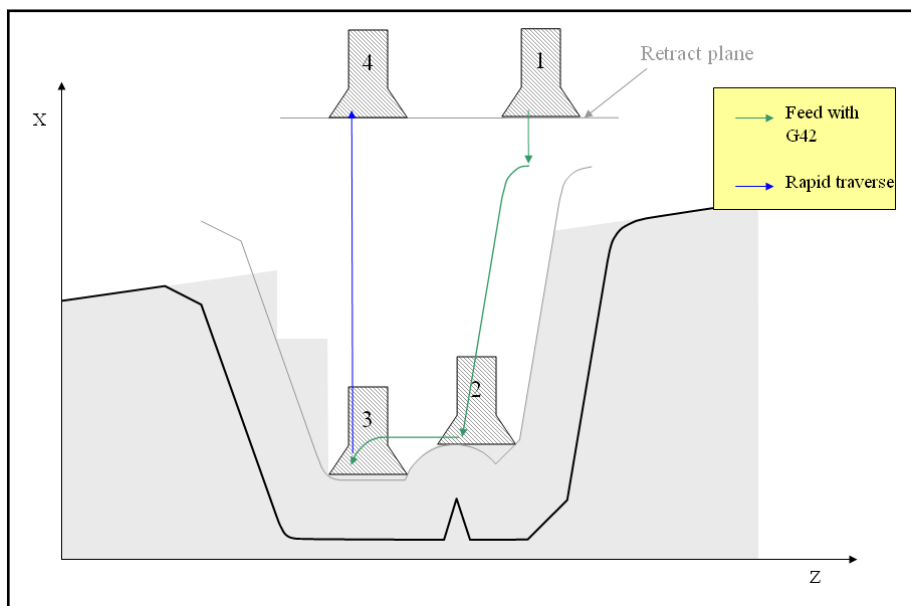


Fig. 11-26: Step 4: Edit right side

The left sides are machined in the last step. Thus, the tool correction specified by the parameter "DL" is activated.

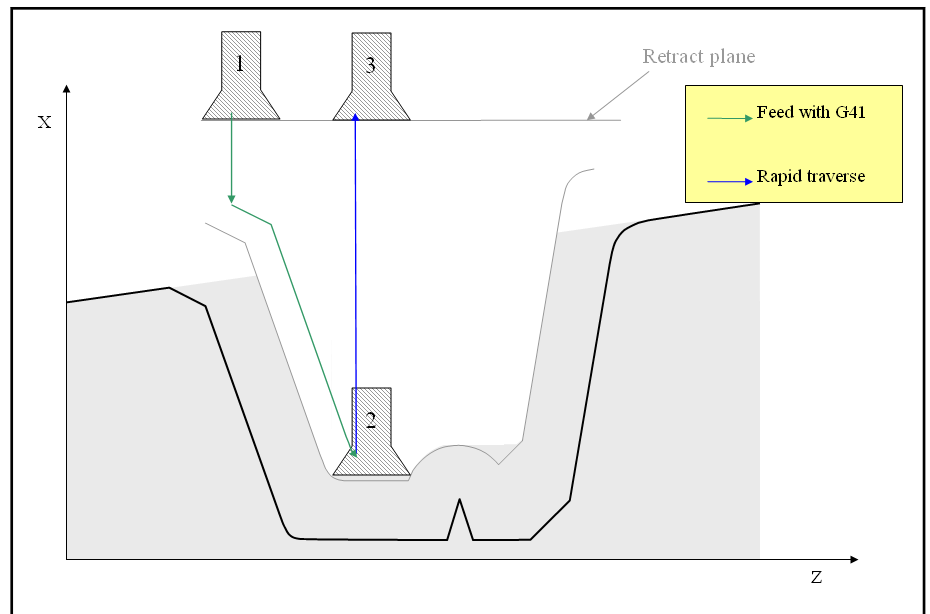


Fig. 11-27: Step 5: Edit left side

If "DL" is not programmed, the cycle uses the tool correction following the tool correction that was active when the cycle was called for machining the left sides. This means that if cycle call D4 was active (and thus used for the right side), the cycle uses D5 for the left side. The same applies to ED.



If $\overline{DL}$ is not programmed and D as well as ED were active during the cycle call, an error message is output.

If no tool correction was active during the cycle call and no $\overline{DL}$ was programmed, the right side is created without tool correction. The left side is displaced by the tool width "TW".

Machining can be appropriately lengthened by means of the tangential approach and run-out.

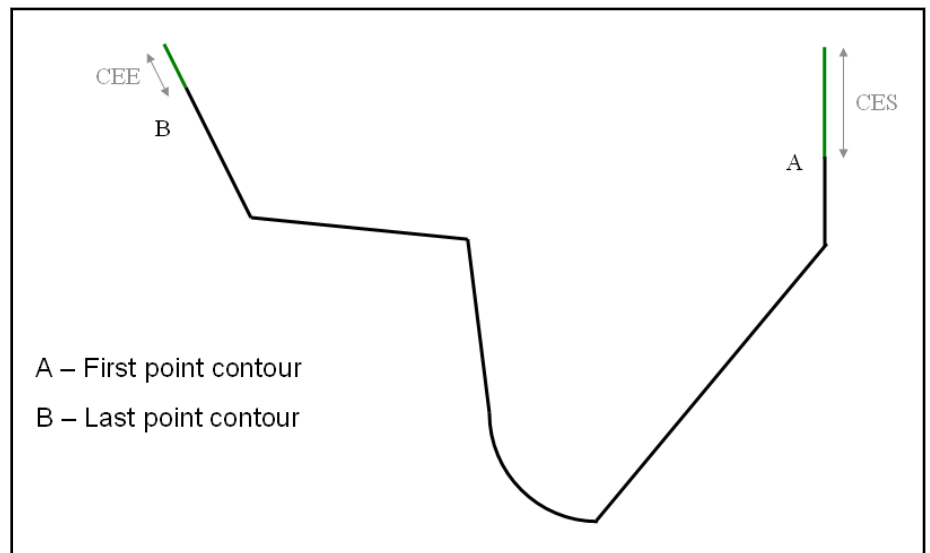


Fig. 11-28: Tangential approach or run-out

Turning cycles (contour-dependent)

The cutting variant "CV" indicates whether only parts of the cycle algorithm are to be executed. If "CV" is not programmed, the complete algorithm is machined.

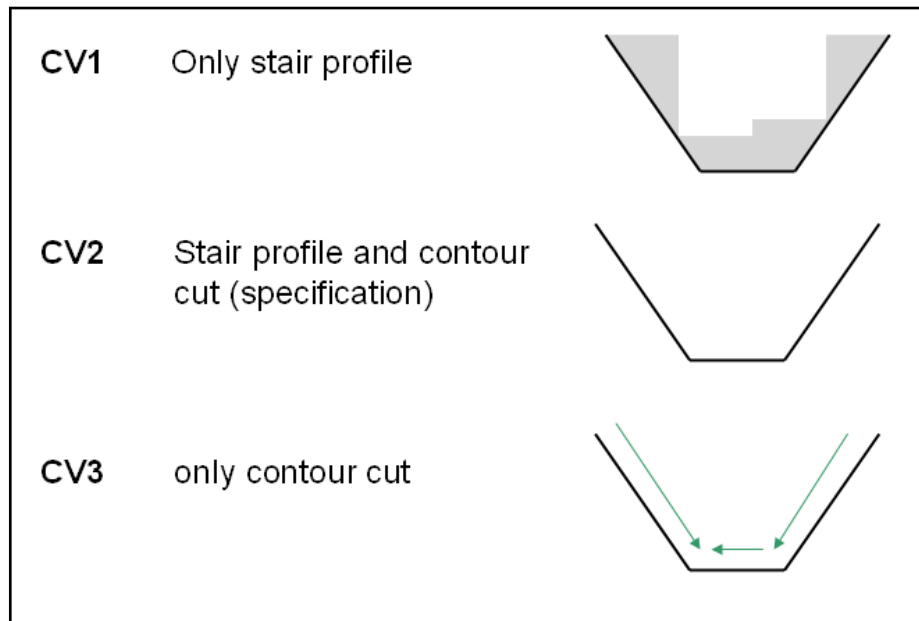


Fig. 11-29: Definition cutting variant CV

Example:

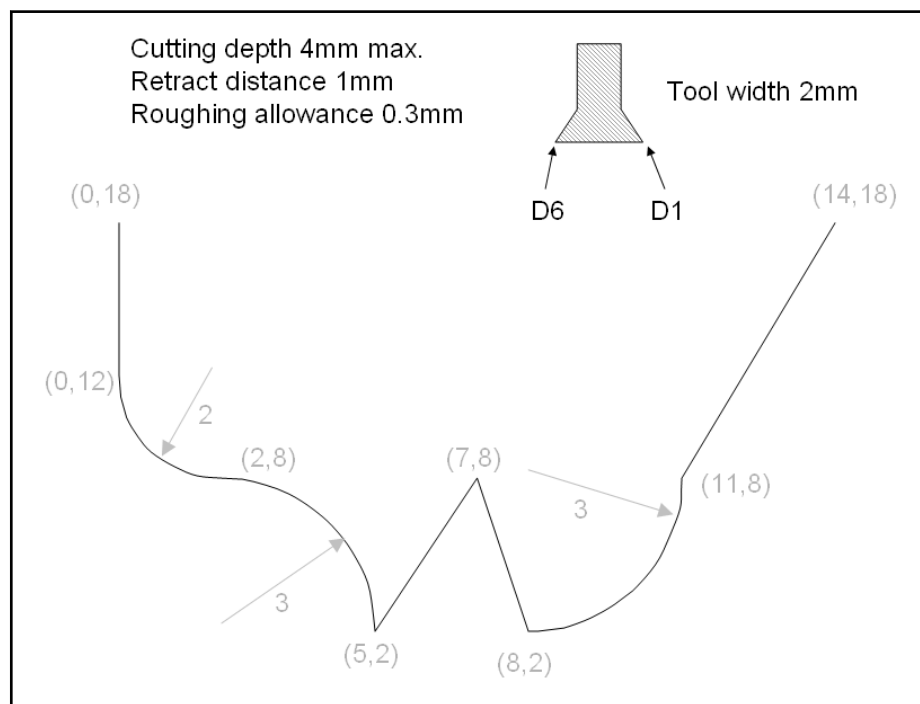


Fig. 11-30: Example G174 Contour plunging
Program for G174 Contour grooving

```

N10 G40 G0 F2000 G94 DIA(X) ;Initialization
N20 X25 Z10 G42 G47 D1
N30 X20 Z10 ;Approach starting point
N40 G174(CD4, LD1, ORI8, TW2, DL D6, S100, E170, RSL0.3) ;Call cycle
N50 M30 ;Program end

```

```

;*****
;*** Contour description ***

```

```
;*****  
N100 Z14 X18  
N110 Z11 X8  
N120 G2 Z8 X2 R3  
N130 G1 Z7 X8  
N140 Z5 X2  
N150 G3 Z2 X8 R3  
N160 G2 Z0 X12 R2  
N170 G1 X18
```



The programmed contour must neither start nor end in a straight line that extends in parallel to the main axis. A respective contour can be supplemented by a distance in secondary axis direction.

12 Contour drafts

12.1 Contour programming G31

12.1.1 General information

Definition Open and closed contours can be written with the contour path programming. The contour can consist of straight line and circle segments.

The contour move programming supports for example:

- Programming straight lines (2, 3, and 4 point moves)
- Tangential transitions of circles and straight lines
- Calculating the intersection for circles and straight lines

Additionally, the intersections of straight lines and circles can be extended by chamfers and roundings.

The following example shows the possible basic forms that can be combined.

The following program syntax is used:

Program address

X, Y, Z, ...	Coordinate names (as configured)
A	Angles (syntax as for polar angle 1 of the polar coordinate programming; can be set via machine parameters)
I, J, K	Interpolation parameters for the circle center point (the circle center point is always specified as absolute for G31)
R	Circle radius
CW, CCW	Direction of rotation of the circle: Clockwise, counterclockwise
TG	Tangential transition or tangential connection
CS, CSL	Chamfer segment of a connecting chamfer
CL, CLL	Chamfer length of a connecting chamfer
RND, RNDL	Rounding radius

Tab. 12-1: Program syntax

CS, CSL always refers to the positive main axis of the active plane in case of G31.

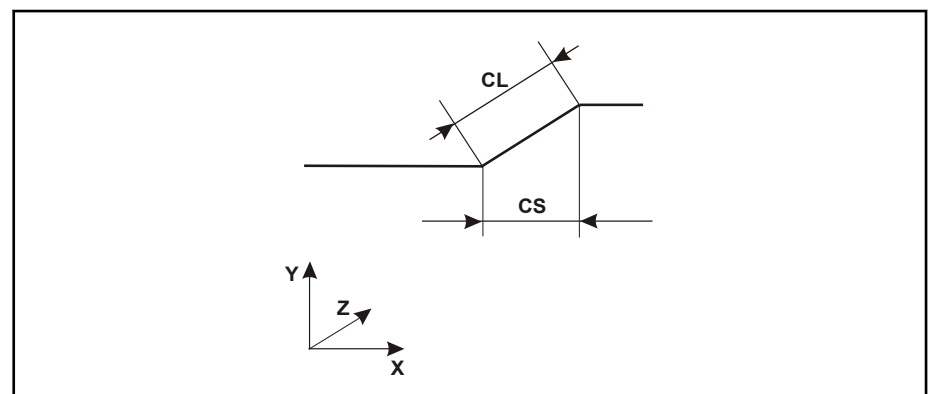


Fig. 12-1: Chamfer length and chamfer segment



The polar coordinate angle must not be assigned an axis name. In this case, change the syntax of the polar coordinate angle in the machine parameters, e.g. for axis name "A", the syntax for polar coordinate angle 1 should be "AW" in machine parameter "NCP/PolarCoord/PolarCoordAngle1".

12.1.2 Contour programming G31 for milling

Point contours

2-point contour

Definition A 2-point contour is programmed in an NC block.
This NC block always contains an angle and an end point coordinate.
The position of the previous NC block is assumed as starting position.
The second coordinate of the end position X2 or Y2 is calculated.

Description The starting position of the chamfer in X1 and Y1 is known.
The angle of the chamfer (A1) and a coordinate of the end position is known (X2 or Y2).
The second coordinate of the end position is calculated (X2 or Y2).

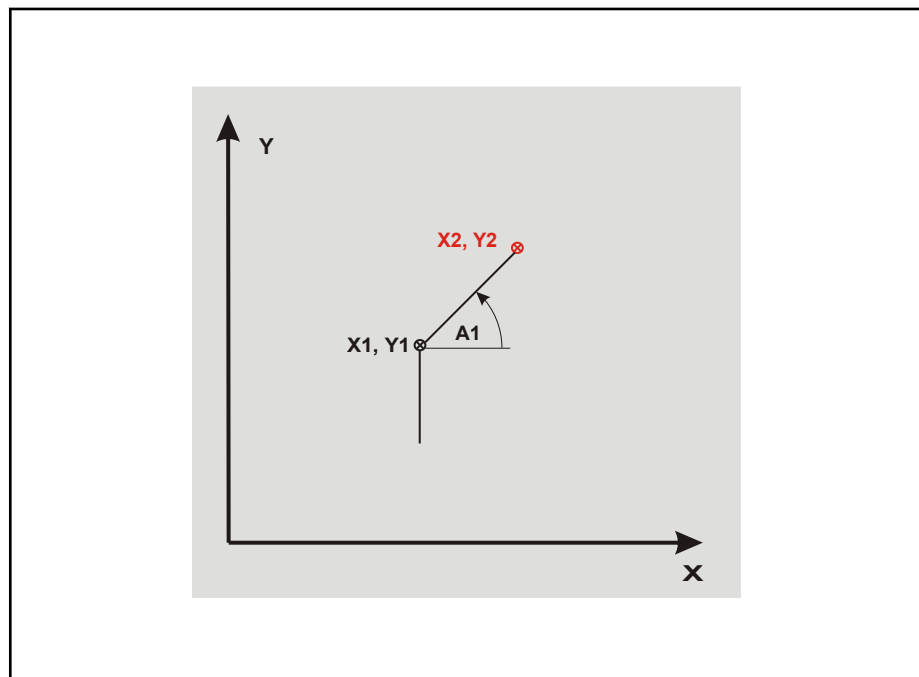


Fig. 12-2: 2-point contour

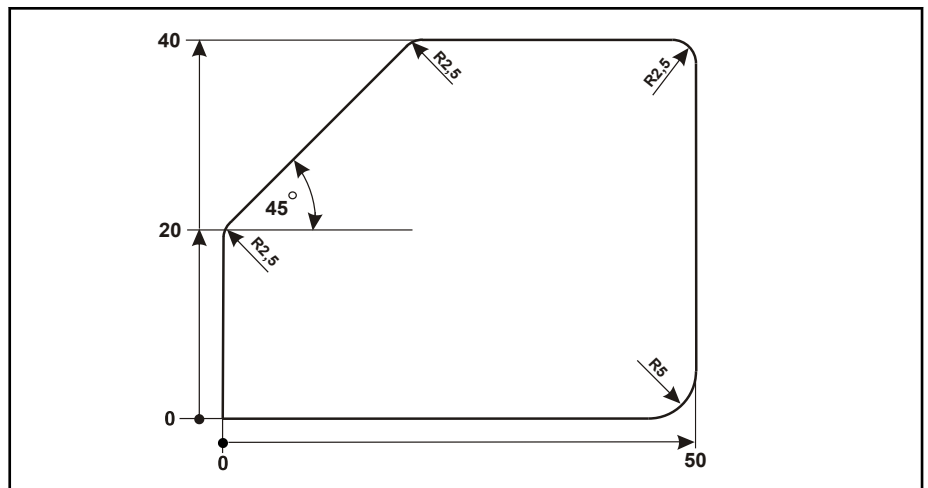


Fig. 12-3: Sketch: 2-point contour, straight line, chamfer

Program excerpt: 2-point contour, straight line, chamfer

N10 G17 G0 X0 Y0	;approaching the start position
N20 G1 Y20 RNDL(2.5) F1000	;straight line with rounding
N30 G31 A45 Y40 RNDL(2.5)	;contour draft on, inclined straight line, with angle specification, end point in Y, rounding
N40 G1 X50 RNDL(2.5)	;contour draft off, horizontal straight line with rounding
N50 Y0 RNDL(5)	;vertical straight line with rounding
N60 X0	;horizontal straight line

3-point contour

Definition A 3-point contour is programmed in 2 NC blocks.

The first NC block always contains an angle.

The second NC block always contains an angle and the end position.

The position of the previous NC block is assumed as starting position.

The starting position of the chamfer in X1 and Y1 is known.

The coordinates of the point of intersection X2, Y2 are calculated.

Description The starting position X1, Y1 and the angle (A1) of the first chamfer are known.

The end positions X3, Y3 and the angle of the second chamfer (A2) are known.

The end position of the first chamfer X2, Y2 is calculated.

Contour drafts

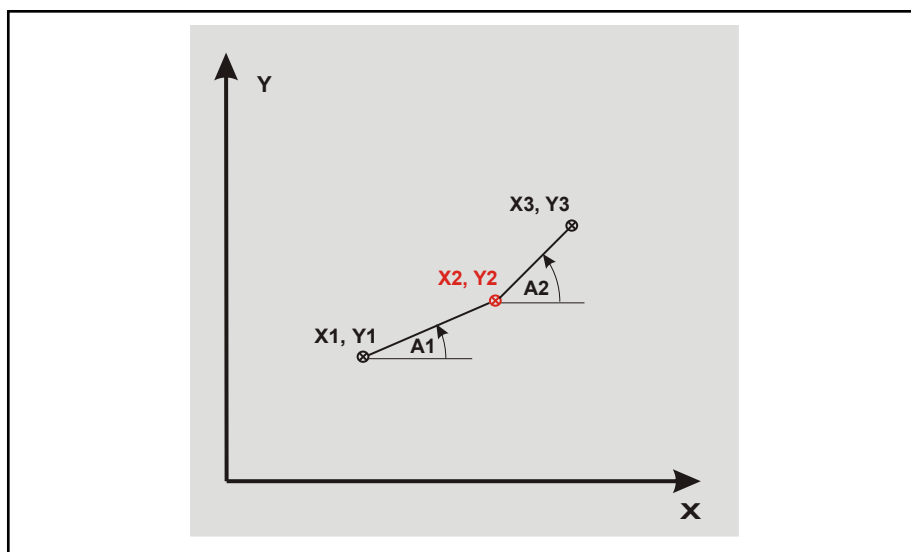


Fig. 12-4: 3-point contour

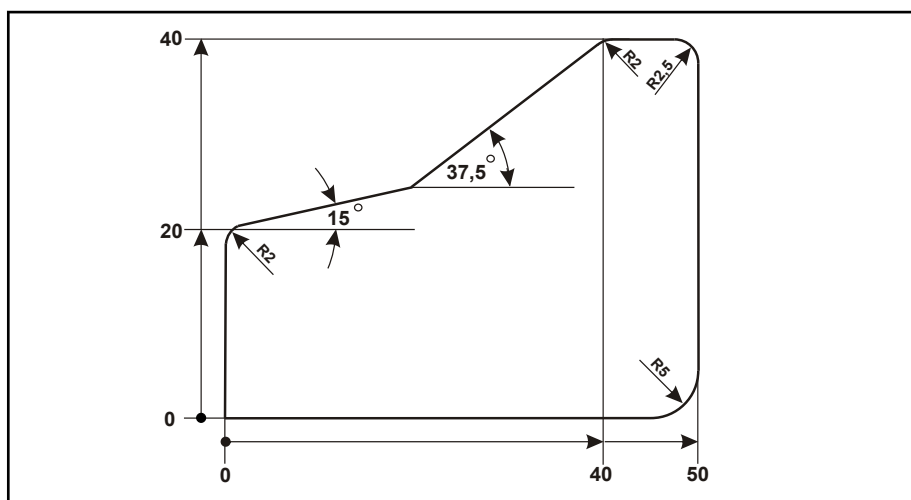


Fig. 12-5: Sketch: 3-point contour, vertical straight line, chamfer, chamfer

Program excerpt: 3-point contour, vertical straight line, chamfer, chamfer

```

N10 G17 G0 X0 Y0      ;approaching the start position
N20 G1 Y20 RNDL(2) F1000 ;straight line with rounding
N30 G31 A15           ;contour draft on, inclined straight line, with angle specification
N40 X40 Y40 A37.5 RNDL(2) ;inclined straight line, with angle specification
N50 G1 X50 RNDL(2.5)   ;contour draft off, horizontal straight line with rounding
N60 Y0 RNDL(5)         ;vertical straight line with rounding
N70 X0                 ;horizontal straight line

```

4-point contour

Definition A 4-point contour is programmed in 3 NC blocks.

The first NC block always contains an angle.

The second NC block always contains an angle.

The third NC block always contains an angle, the end position and the chamfer length or chamfer section.

The position of the previous NC block is assumed as starting position.

The coordinates of the intermediate points X2, Y2 and X3, Y3 are calculated.

Description The starting position X_1, Y_1 and the angle (A_1) of the first chamfer is known.
 The angle (A_2) of the second chamfer is known.
 The angle (A_3) of the third chamfer and the end positions X_4, Y_4 and the chamfer lengths/sections are known.
 The intermediate position of the first chamfer X_2, Y_2 is calculated.
 The intermediate position of the second chamfer X_3, Y_3 is calculated.

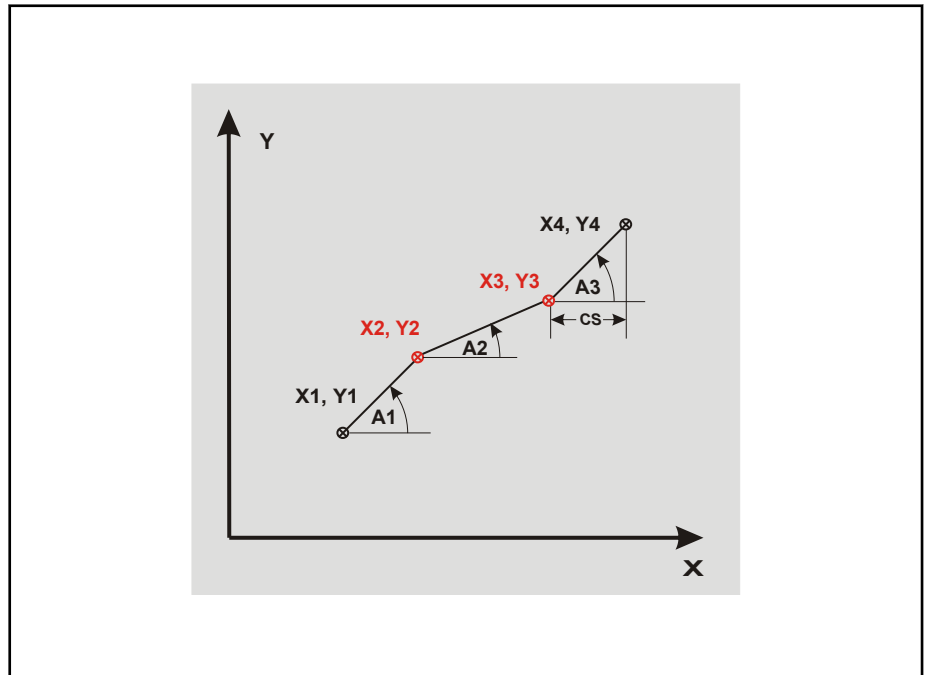


Fig. 12-6: 4-point contour

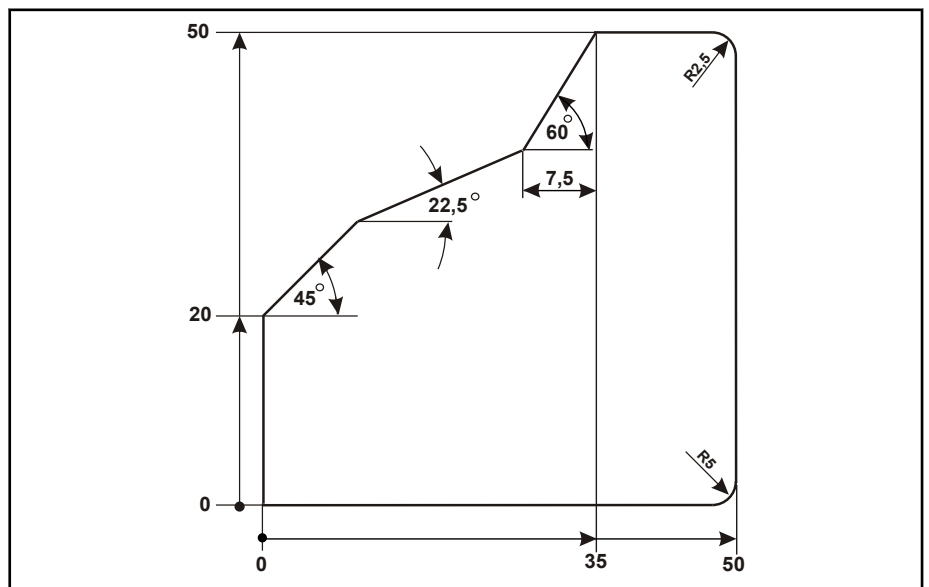


Fig. 12-7: Sketch: 4-point contour, vertical straight line, chamfer, chamfer, chamfer, 4-point contour, horizontal straight line

Program excerpt: 4-point contour, vertical straight line, chamfer, chamfer, chamfer, horizontal straight line

```

N10 G17 G0 X0 Y0      ;approaching the start position
N20 G1 Y20 F1000      ;straight line
N30 G31 A45           ;contour draft on, inclined straight line with
                        angle specification

```

Contour drafts

```

N40 A22.5 ;inclined straight line with angle specification
N50 X35 Y50 A60 CS7.5 ;inclined straight line with angle specification and
                        both end position as well as chamfer section
N60 G1 X50 RNDL(2.5) ;contour draft off, horizontal straight line with rounding
N70 Y0 RNDL(5) ;vertical straight line with rounding
N80 X0 ;horizontal straight line

```

Tangential transitions

Tangential transition of straight line-circle

Definition A straight line tangentially intersects a circular arc.
 The tangential intersection point is calculated.
 The circle is programmed as follows in the following examples:

- Case 1: with radius specification
- Case 2: with center point specification

Description Case 1 The starting position X_1, Z_1 of the straight line is known.
 The end point X_3, Y_3 and the radius of the circular arc are known.
 The tangential intersection point of the straight lines in the circular arc X_2, Y_2 is calculated.

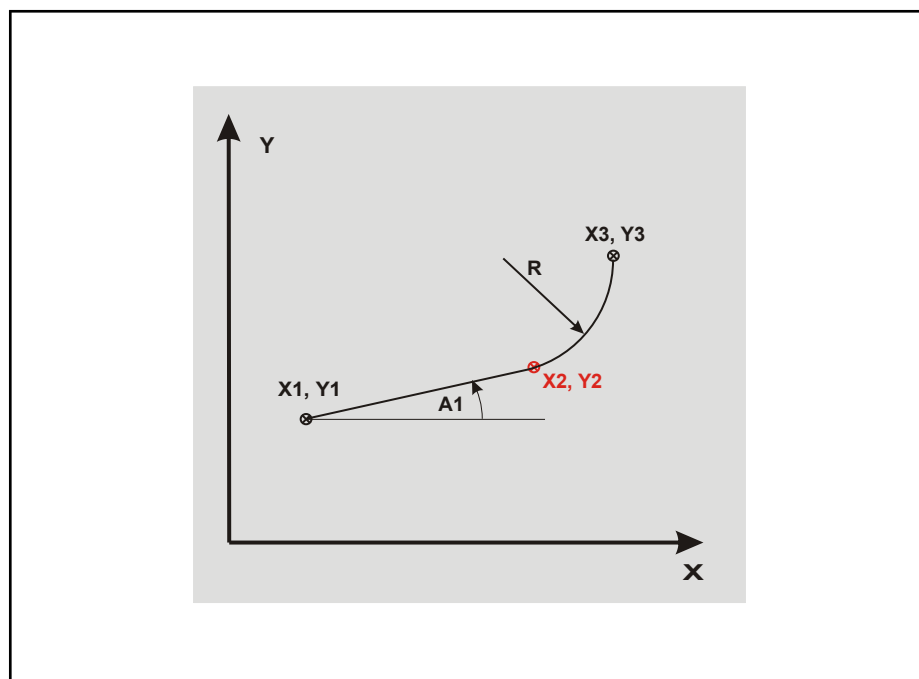


Fig. 12-8: Tangential transition straight line-circle, case 1

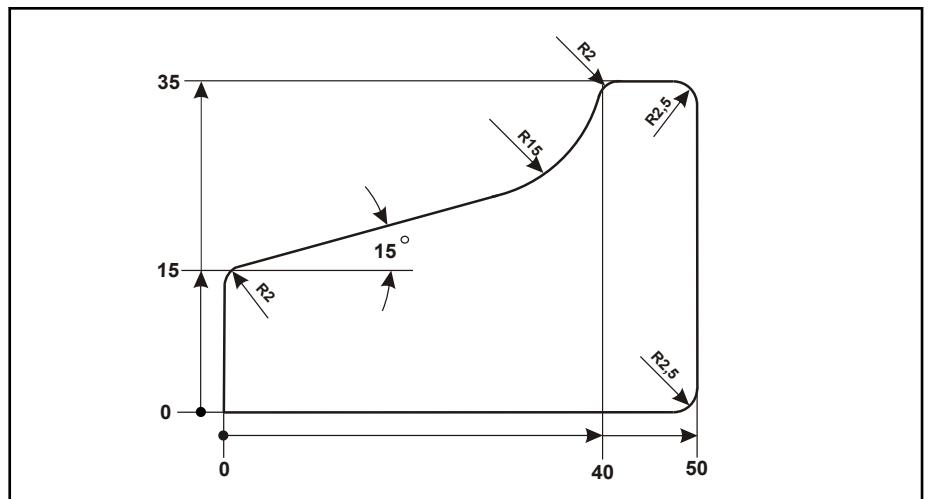


Fig. 12-9: Sketch: Tangential transition straight line-circle, (1)

Program excerpt: Tangential transition: Straight line, circle (case 1)

```

N10 G17 G0 X0 Y0
N20 G1 Y15 RNDL(2) F1000
N30 G31 A15
N40 CCW TG X40 Y35 R15 RNDL(2)
N50 G1 X50 RNDL(2.5)
N60 Y0 RNDL(2.5)
N70 X0

```

;approaching the start position
 ;straight line with rounding
 ;contour draft on, inclined straight line with angle
 ;arc (CCW) tangential connection with end position
 circle radius and rounding
 ;contour draft off, horizontal straight line with rounding
 ;vertical straight line with rounding
 ;horizontal straight line

Description Case 2

The starting position X_1, Z_1 and the angle of the straight line is known.

The end point X_3, Y_3 and the center point coordinates of the circular arc are known.

The tangential intersection point of the straight lines in the circular arc X_2, Y_2 is calculated.

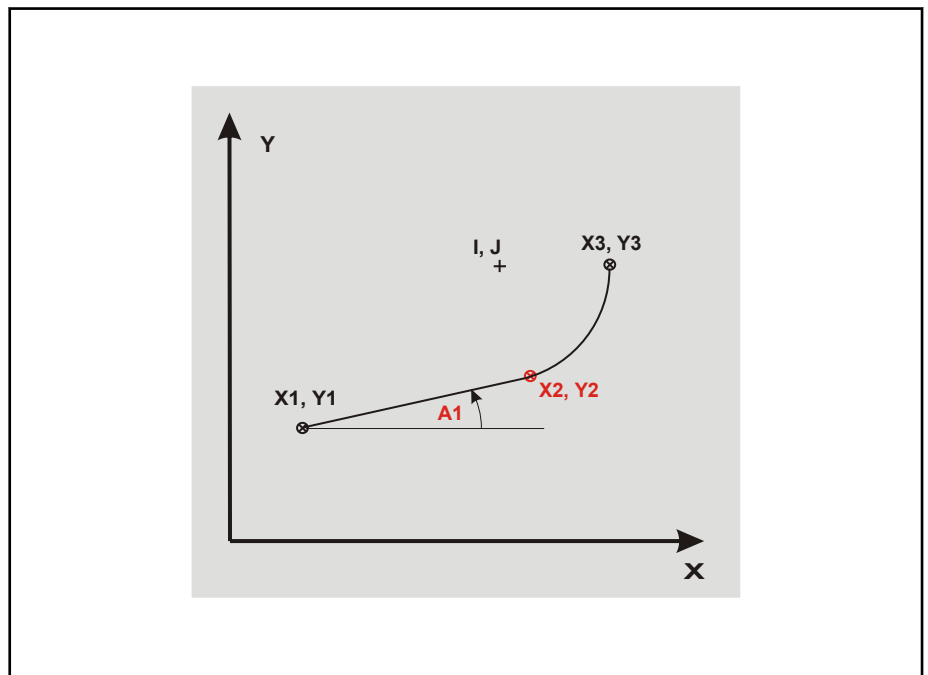


Fig. 12-10: Tangential transition straight line-circle, case 2

Contour drafts

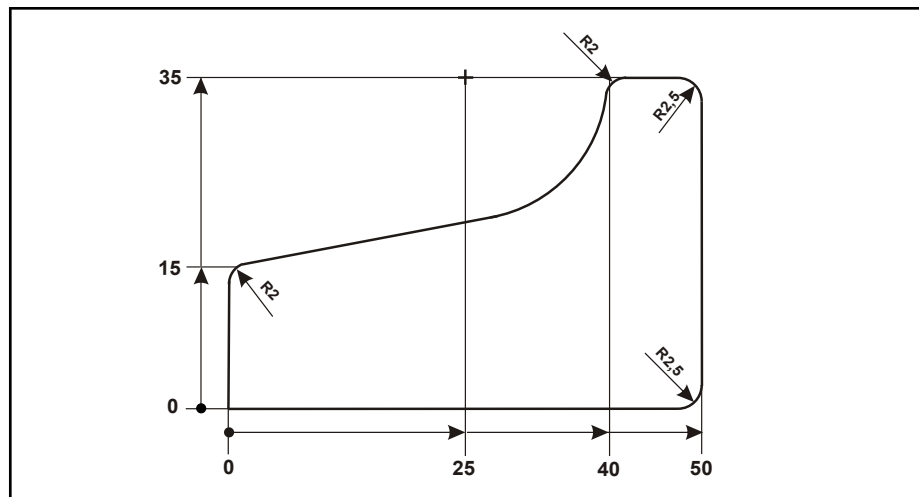


Fig. 12-11: Sketch: Tangential transition straight line-circle, (2)

Program excerpt: Tangential transition: Straight line, circle (case 2)

```

N10 G17 G0 X0 Y0 ;approaching the start position
N20 G1 Y15 RNDL(2) F1000 ;chamfer with rounding
N30 G31 TG ;contour draft on, inclined straight line with angle
N40 CCW X40 Y35 I25 J35 RNDL(2) ;arc (CCW) tangential connection with end position and circle center
N50 G1 X50 RNDL(2.5) ;contour draft off, horizontal straight line with rounding
N60 Y0 RNDL(2.5) ;vertical straight line with rounding
N70 X0 ;horizontal straight line

```

Tangential transition circle, straight line

Definition

A circular arc intersects a straight line.

The tangential intersection point is calculated.

Description

The starting position $X1, Y1$ with the circle center point coordinates K, I of the circular arc are known.

The end position $X3, Y3$ are known.

The tangential transition point from the circular arc to the straight line $Z2, X2$ is calculated.

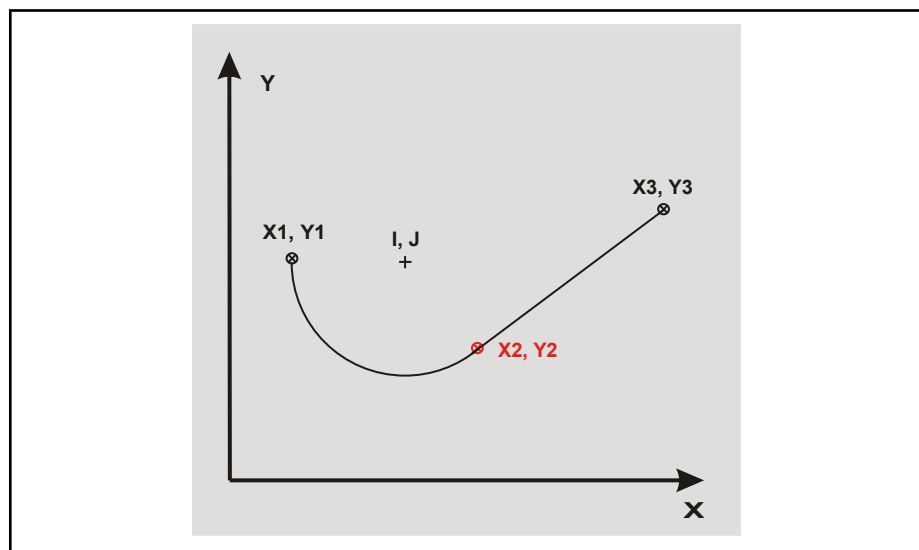


Fig. 12-12: Tangential transition circle, straight line

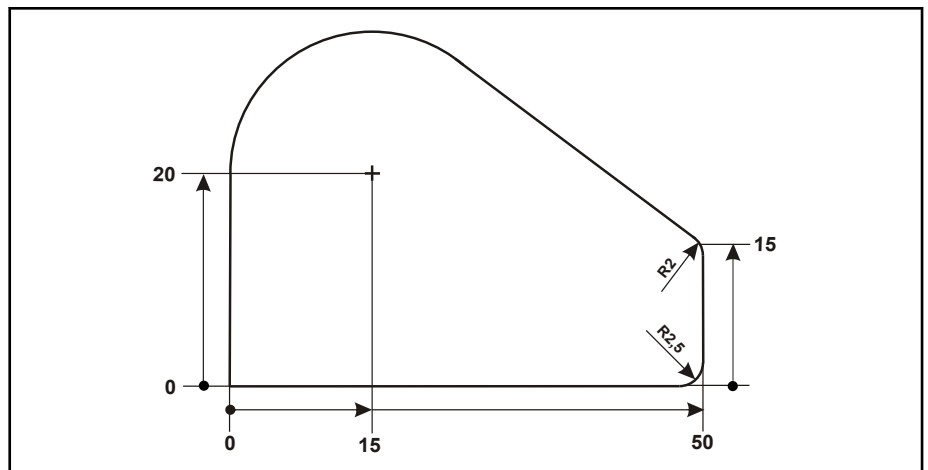


Fig. 12-13: Sketch: Tangential transition circle, straight line

Program excerpt: Tangential transition: Circle, straight line

```

N10 G17 G0 X0 Y0      ;approaching the starting position
N20 G1 Y20 F1000      ;vertical straight line
N30 G31 CW I15 J20    ;Contour draft on, arc (CW) with circle center point
N40 TG X50 Y15 RNDL(2) ;tangential connection with end position and rounding
N50 G1 Y0 RNDL(2.5)   ;contour draft off, straight line vertical with rounding
N60 X0                ;horizontal straight line

```

Tangential transitions from circle-straight line-circle

Definition

Two circular arcs are connected to a straight line tangentially.

The contact points of the straight lines at the circular arcs are calculated.

Description

The first circular arcs is determined by the circle center point coordinates I, J.

The second circular arcs is determined by the end points X4, Y4 and the circle center point coordinates I, J.

The starting position of the straight lines X2, Y2 is calculated.

The end position of the straight lines X3, Y3 is calculated.

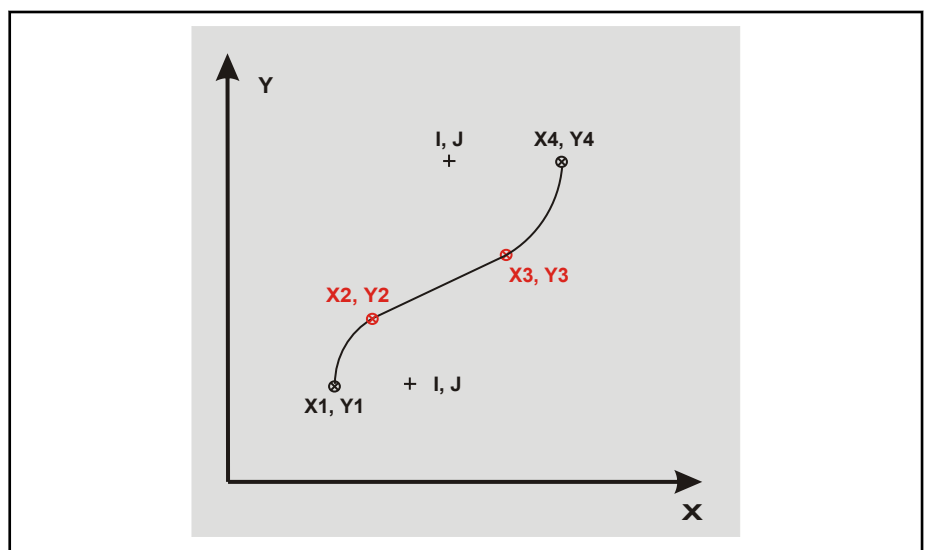


Fig. 12-14: Tangential transitions from circle-straight line-circle

Contour drafts

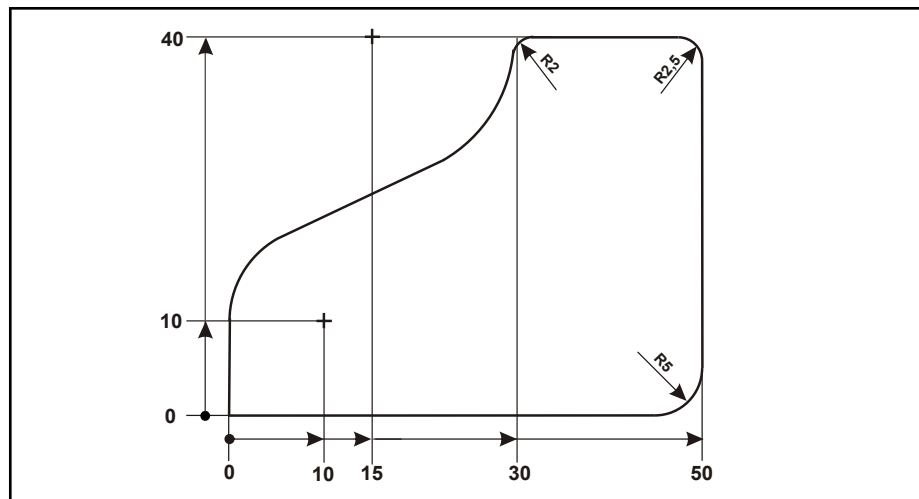


Fig. 12-15: Sketch: Tangential transition circle, straight line, circle

Program excerpt: Tangential transition circle, straight line, circle

N10 G17 G0 X0 Y0	;approaching the start position
N20 G1 Y10 F1000	;vertical straight line
N30 G31 CW I10 J10	;contour draft on, arc clockwise with circle center
N40 TG	;tangential connection (straight line)
N50 CCW X30 Y40 I15 J40 RNDL(2)	;arc (CCW) with end position, circle center with rounding
N60 G1 X50 RNDL(2.5)	;contour draft off, horizontal straight line with rounding
N70 Y0 RNDL(5)	;vertical straight line with rounding
N80 X0	;horizontal straight line

Tangential transitions from circle-circle-circle

Definition Two circular arcs are tangentially connected to a straight line.
The contact points of the three circle are calculated.

Description The first circular arcs is determined by the circle center point coordinates I, J.
The second circular arc is only determined by the radius.
The third circular arc is determined by the end points Z4, X4 and the circle center point coordinates I, J.
The contact points of the second circular arcs X2, Y2 and X3, Y3 are calculated.

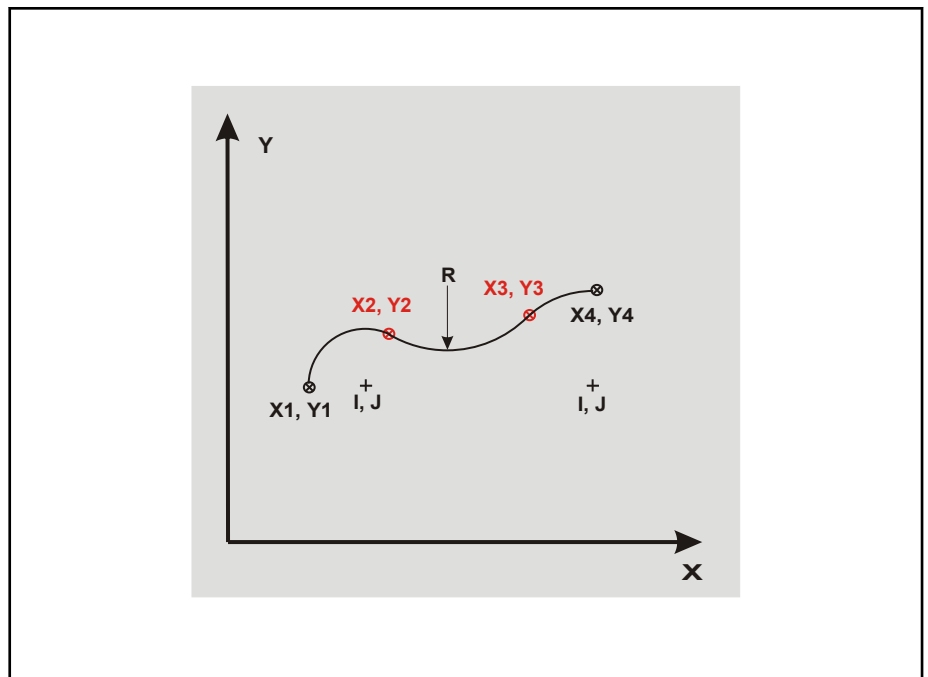


Fig. 12-16: Tangential transitions from circle-circle-circle

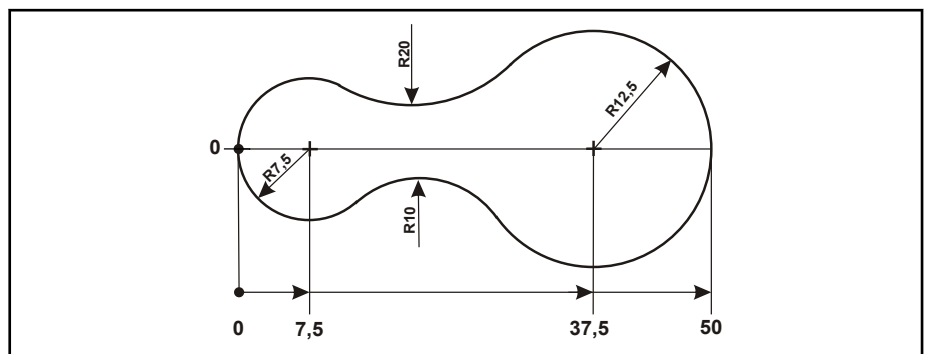


Fig. 12-17: Sketch: Tangential transition circle, circle, circle

Program excerpt: Tangential transition circle, circle, circle

```

N10 G17 G0 X0 Y0           ; approaching the start position
N20 G31 CW I7.5 J0 F1000    ; contour draft on, arc (CW) with circle center
N30 TG R20                  ; tangential connection (circle), circle radius
N40 CW X50 Y0 I37.5 J0      ; arc (CW) with end position and circle center
N50 CW I37.5 J0              ; arc (CW) with circle center
N60 TG R10                  ; tangential connection (circle), circle radius
N70 CW X0 Y0 I 7.5 J0       ; arc (CW) with end position and circle center
N80 G1                       ; contour draft off

```

Intersection transitions

Intersection of straight line-circle

Definition

The intersection of a straight line with a circular arc is calculated.

Description

The starting position X1, Y1 and the angle (A1) are known.

The end position of the circle in X3, Y3 is known.

The center point of the circle is known.

The intersection of the circle with the straight lines X2, Y2 is calculated.

Contour drafts

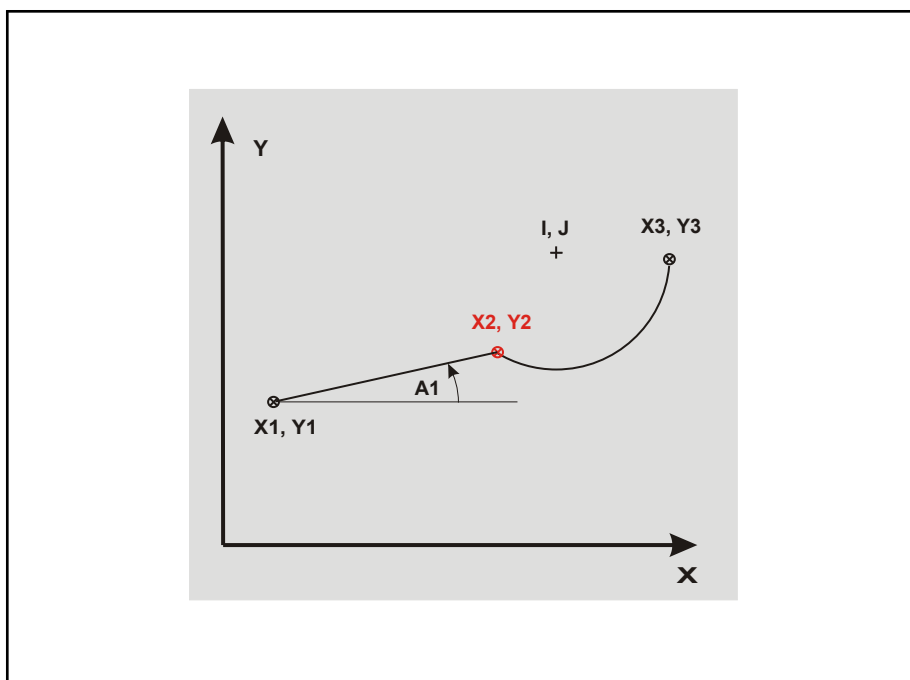


Fig. 12-18: Intersection of straight line-circle

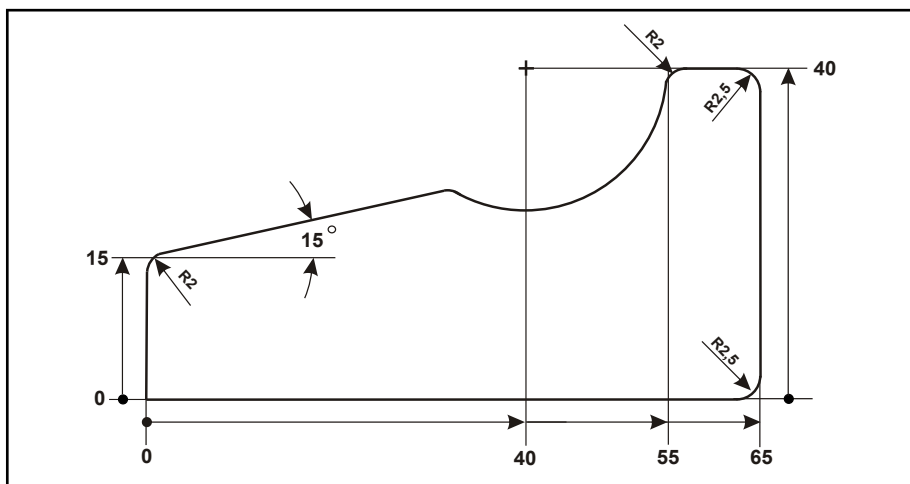


Fig. 12-19: Sketch: Intersection of straight line, circle

Program excerpt: Intersection of straight line, circle

```

N10 G17 G0 X0 Y0           ;approaching the start position
N20 G1 Y15 RNDL(2) F1000    ;straight line with rounding
N30 G31 A15                 ;contour draft on, inclined straight line with angle
N40 CCW X55 Y40 I40 J40 RNDL(2) ;arc (CCW) with end position, circle center and rounding
N50 G1 X65 RNDL(2.5)         ;contour draft off, horizontal straight line with rounding
N60 Y0 RNDL(2.5)            ;vertical straight line with rounding
N70 X0                      ;horizontal straight line

```

Intersection of circle-straight line**Definition**

Intersection of a circular arc with a straight line is calculated.

Description

The starting position X1, Y1 of the circular arc and the center point coordinates I, K of the circular arc are known.

The end position Z3, Y3 and the angle are known.

The intersection of the circular arc X2, Y2 with the straight lines is calculated.

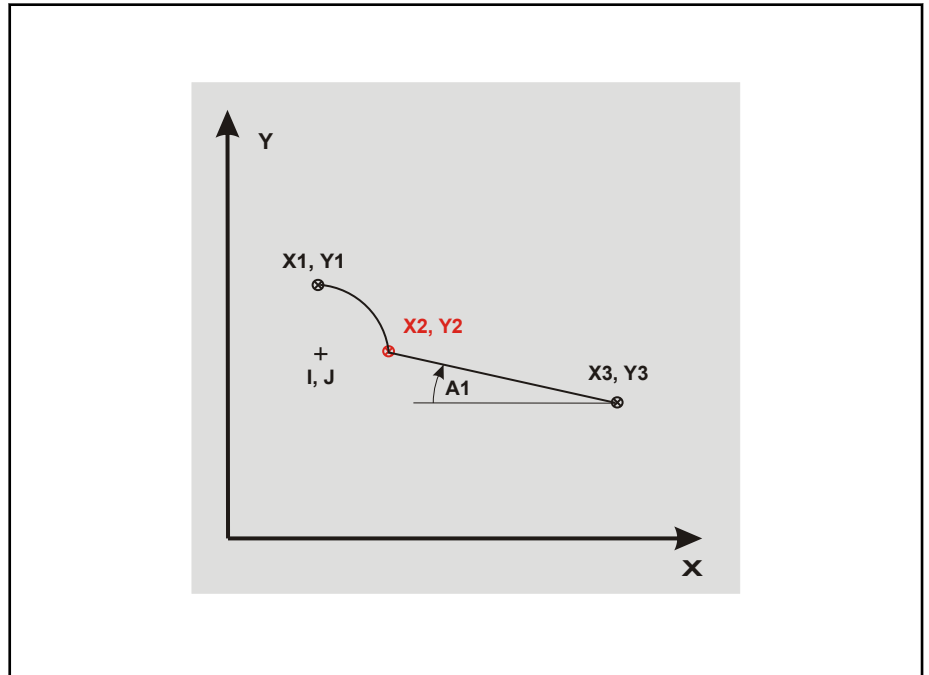


Fig. 12-20: Intersection of circle-straight line

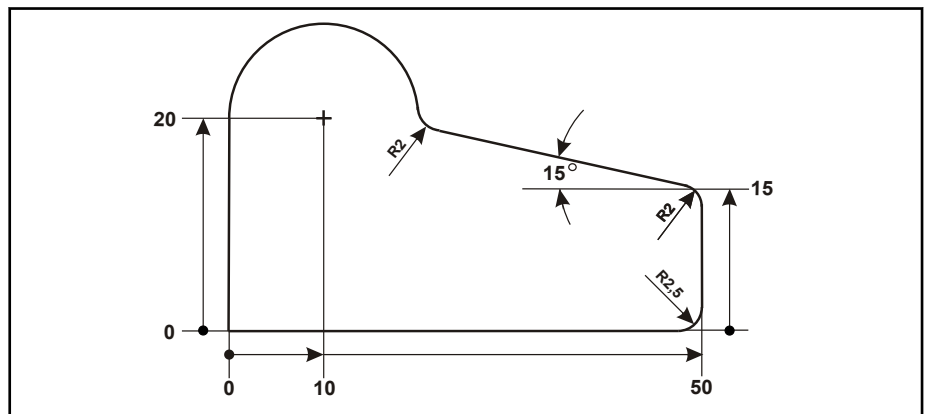


Fig. 12-21: Sketch: Intersection of circle, straight line

Program excerpt: Intersection of circle, straight line

```

N10 G17 G0 X0 Y0      ;approaching the start position
N20 G1 Y20 F1000      ;vertical straight line
N30 G31 CW I10 J20     ;contour draft on, arc(CW) with center and rounding
N40 X50 Y15 A165 RNDL(2) ;inclined straight line with angle and rounding
N50 G1 Y0 RNDL(2.5)    ;contour draft off, vertical straight line with rounding
N60 X0                ;horizontal straight line

```

Intersection of circle-circle

Definition The intersection of two circular arcs is calculated.

Description

The circle center point coordinates I, J of the first circular arc are known.

The end position X3, Y3 of the second circular arc and the center point coordinates I, K of the circular arc are known.

The intersections of both circles X2, Y2 are calculated.

Contour drafts

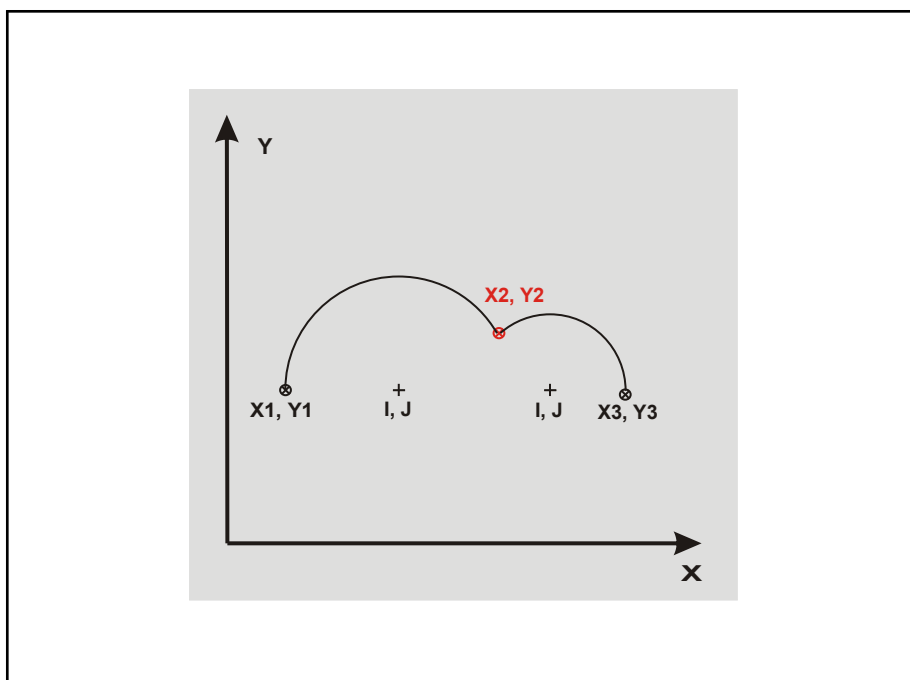


Fig. 12-22: Intersection of circle-circle

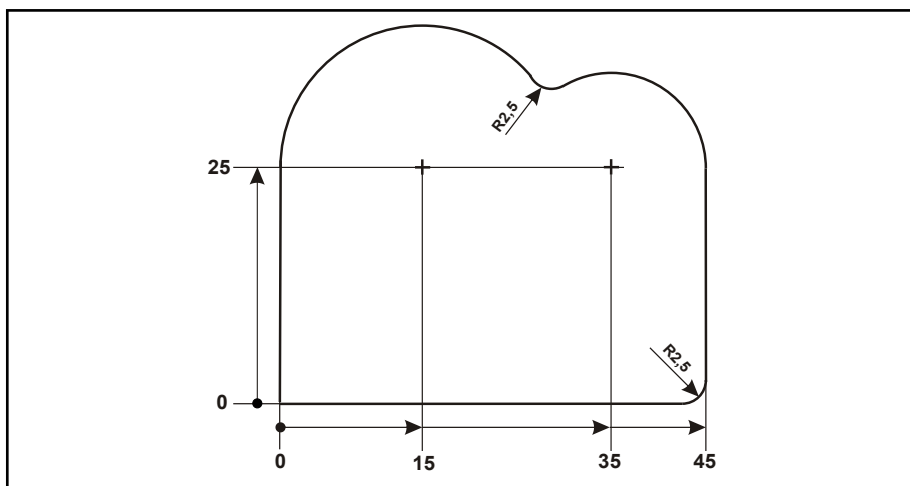


Fig. 12-23: Sketch: Intersection of circle-circle

Program excerpt: Intersection of circle-circle

```

N10 G17 G0 X0 Y0          ;approaching the start position
N20 G1 Y25 F1000          ;vertical straight line
N30 G31 CW I15 J25 RNDL(2.5) ;contour draft on, arc (CW) with center and rounding
N40 CW X45 Y25 I35 J25    ;Kreisbogen (CW) mit Endposition und Mittelpunkt
N50 G1 Y0 RNDL(2.5)       ;contour draft off, vertical straight line with rounding
N60 X0                    ;horizontal straight line

```

12.1.3 Contour programming G31 for turning

Point contours

2-point contour

Definition A 2-point contour is programmed in an NC block.
This NC block always contains an angle and an end point coordinate.

The position of the previous NC block is assumed as starting position.

The second coordinate of the end position Z2 or X2 is calculated.

Description The starting position of the chamfer in Z1 and X1 is known.
The angle of the chamfer (A1) and a coordinate of the end position is known (Z2 or X2).
The second coordinate of the end position is calculated (Z2 or X2).

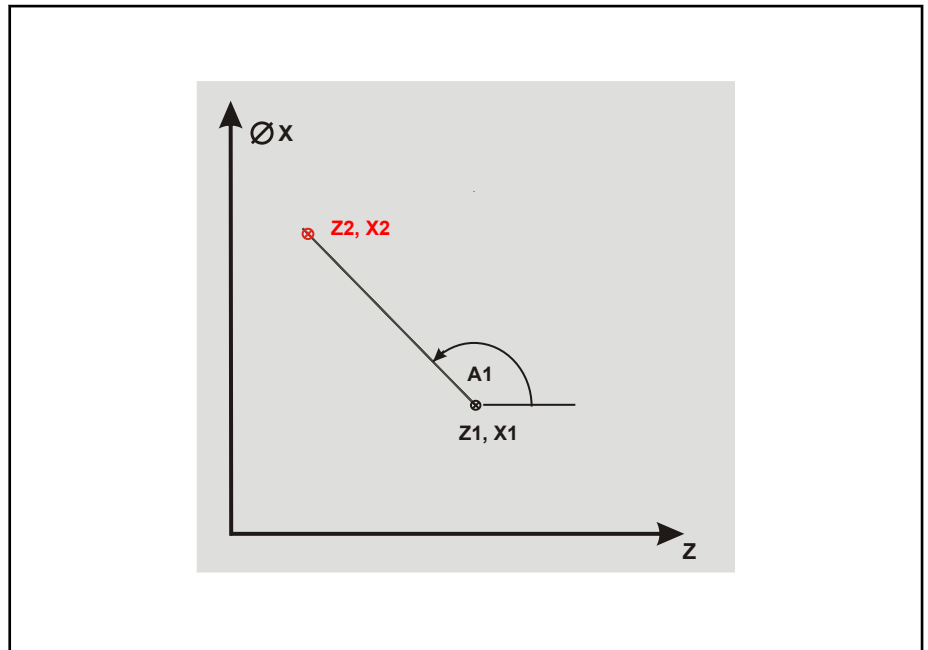


Fig. 12-24: 2-point contour

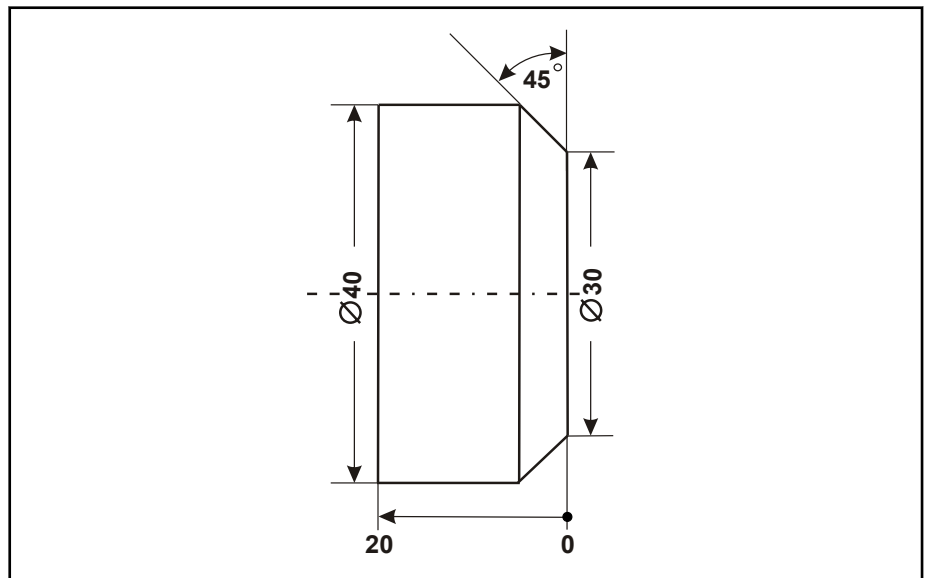


Fig. 12-25: Sketch: 2-point contour, vertical straight line, chamfer
Program excerpt: 2-point contour, vertical straight line, chamfer

```
N10 G96 S200 M4      ;constant cutting velocity and spindle on
N20 G0 G18 Z0 X0      ;approaching the start position
N30 G95 G1 X30 F0.1    ;vertical straight line
N40 G31 A135 X40       ;contour draft on, inclined straight line with angle
```

Contour drafts

N50 G1 Z-20

```
specification, end point in X
;contour draft off, horizontal straight line
```

3-point contour

Definition A 3-point contour is programmed in 2 NC blocks.
 The first NC block always contains an angle.
 The second NC block always contains an angle and the end position.
 The position of the previous NC block is assumed as starting position.
 The starting position of the chamfer in Z1 and X1 is known.
 The coordinates of the point of intersection Z2, X2 are calculated.

Description The starting position Z1, X1 and the angle (A1) of the first chamfer are known.
 The end positions Z3, X3 and the angle of the second chamfer (A2) are known.
 The end position of the first chamfer Z2, X2 is calculated.

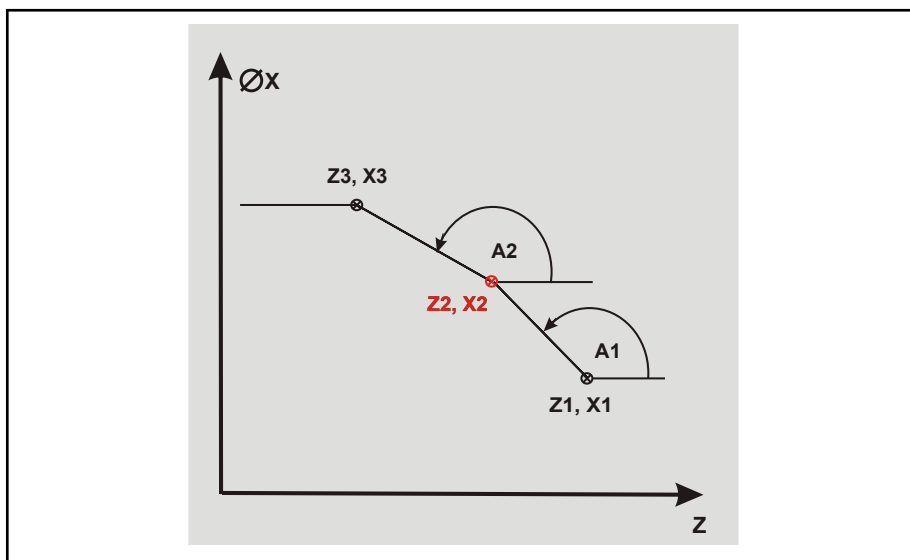


Fig. 12-26: 3-point contour

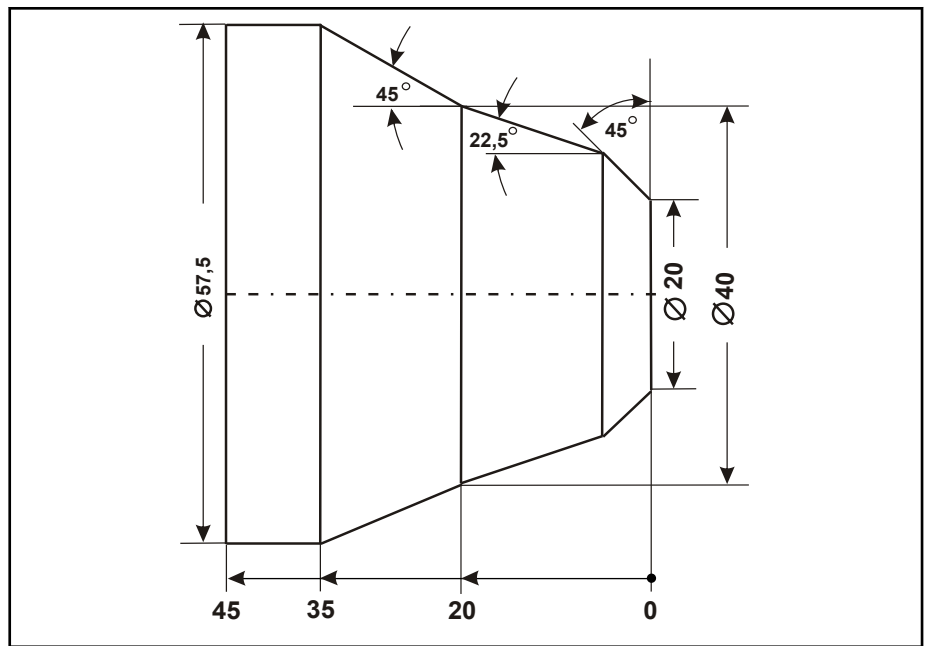


Fig. 12-27: Sketch: 3-point contour, vertical straight line, chamfer, chamfer
 Program excerpt: 3-point contour, vertical straight line, chamfer, chamfer

```

N10 G96 S200 M4 ;constant cutting velocity and spindle on
N20 G0 G18 Z0 X0 ;approaching the start position
N30 G95 G1 X20 F0.1 ;vertical straight line
N40 G31 A135 ;contour draft on, inclined straight line with
;angle specification
N50 Z-20 X40 A167.5 ;inclined straight line with angle
;specification and both end positions
N60 Z-35 A135 ;inclined straight line with angle specification
N70 G1 Z-45 ;contour draft off, horizontal straight line
  
```

4-point contour

Definition A 4-point contour is programmed in 3 NC blocks.

The first NC block always contains an angle.

The second NC block always contains an angle.

The third NC block always contains an angle, the end position and the chamfer length or chamfer section.

The position of the previous NC block is assumed as starting position.

The coordinates of the intermediate points Z2, X2 and Z3, X3 are calculated.

Description The starting position Z1, X1 and the angle (A1) of the first chamfer are known.

The angle (A2) of the second chamfer is known.

The angle (A3) of the third chamfer and the end positions Z4, X4 and the chamfer lengths/sections are known.

The intermediate position of the first chamfer Z2, X2 is calculated.

The intermediate position of the second chamfer Z3, X3 is calculated.

Contour drafts

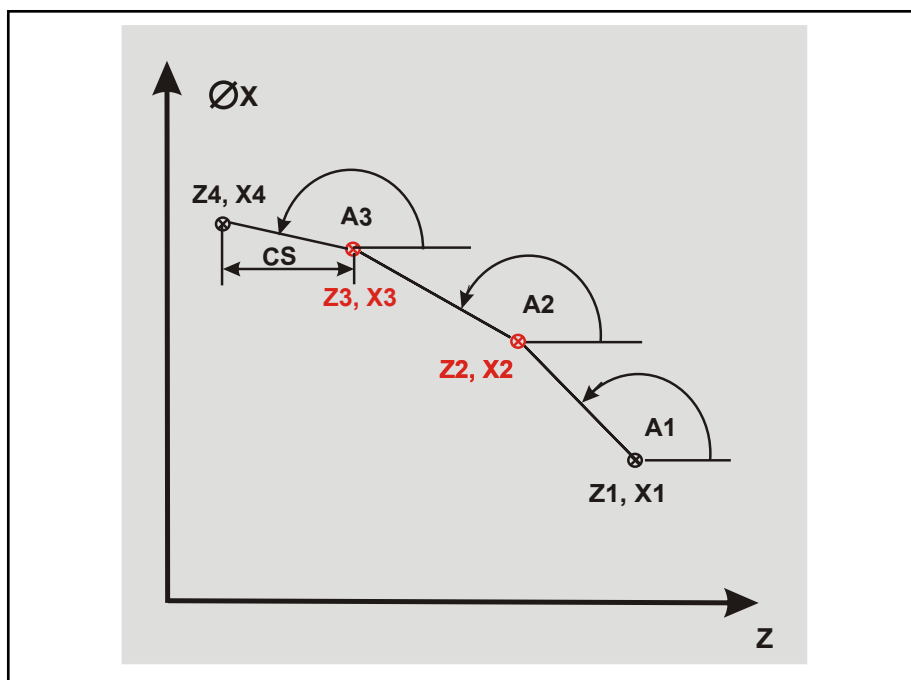


Fig. 12-28: 4-point contour

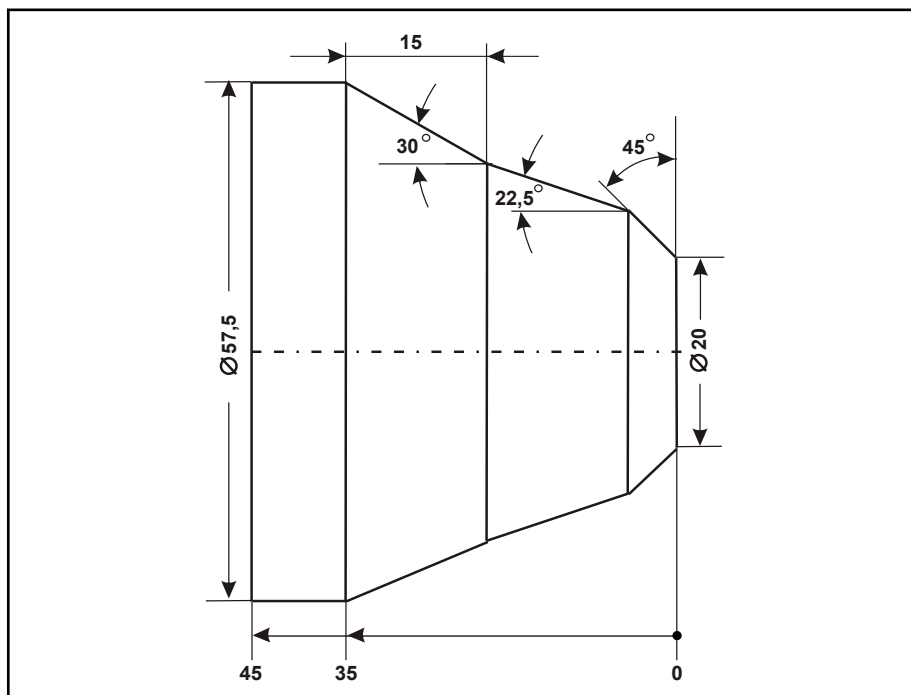


Fig. 12-29: Sketch: 4-point contour, vertical straight line, chamfer, chamfer, chamfer, horizontal straight line

Program excerpt: 4-point contour, vertical straight line, chamfer, chamfer, chamfer, horizontal straight line

```

N10 G96 S200 M4           ;constant cutting velocity and spindle on
N20 G0 G18 Z0 X0          ;approaching the start position
N30 G95 G1 X20 F0.1       ;straight line
N40 G31 A135              ;contour draft on, inclined straight line with
                           ;angle specification
N50 A167.5                ;inclined straight line with angle specification
N60 Z-35 X57.5 A150 CS15  ;inclined straight line with angle specification

```

N70 G1 Z-45

both end position and chamfer length
;contour draft off, horizontal straight line

Tangential transitions

Tangential transition of straight line-circle

- Definition** A straight line tangentially intersects a circular arc.
The tangential intersection point is calculated.
The circle is programmed as follows in the following examples:
- Case 1: with radius specification
 - Case 2: with center point specification

Description Case 1 The starting position Z_1, X_1 and the angle of the straight line are known.
The end point Z_3, X_3 and the radius of the circular arc are known.
The tangential transition point of the straight lines in the circular arc Z_2, X_2 is calculated.

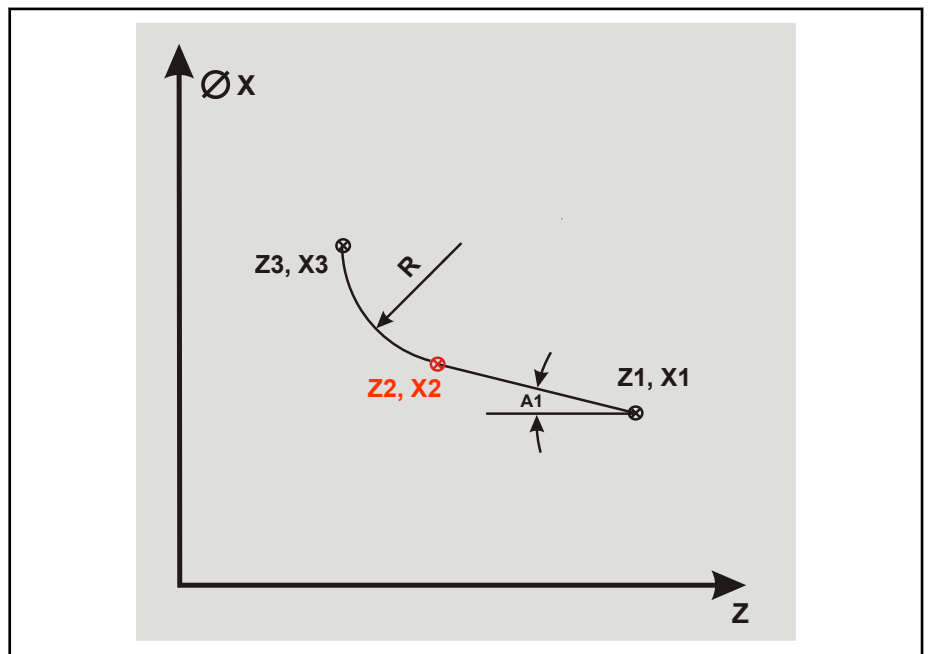


Fig. 12-30: Tangential transition straight line-circle, case 1

Contour drafts

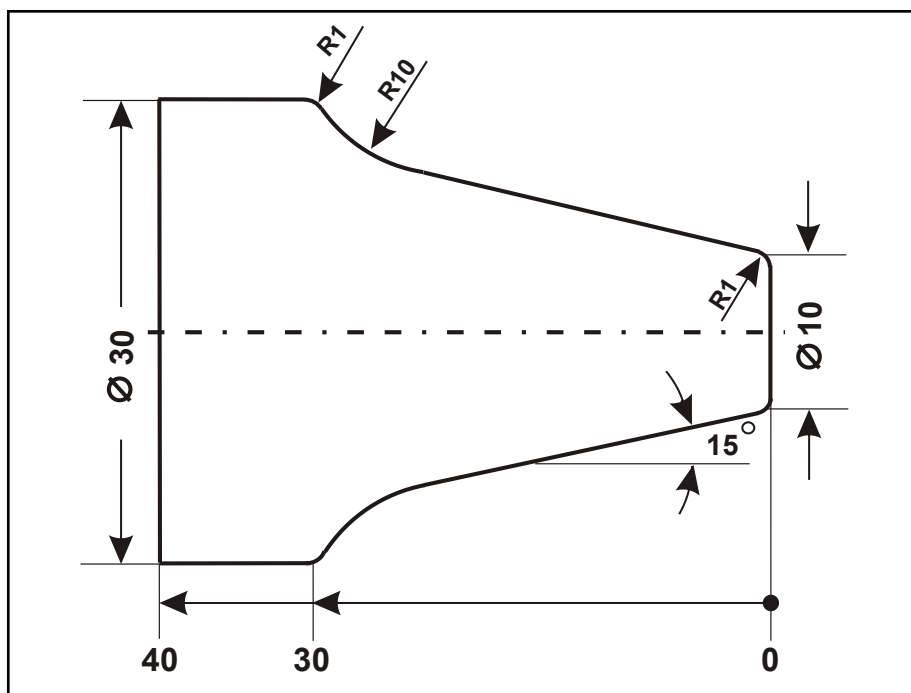


Fig. 12-31: Sketch: Tangential transition straight line-circle, (1)

Program excerpt: Tangential transition straight line-circle, (case 1)

```

N10 G96 S200 M4           ;constant cutting velocity and spindle on
N20 G0 G18 Z0 X0          ;approaching the start position
N30 G95 G1 X10 RNDL(1) F0.1 ;vertical straight line with rounding R1
N40 G31 A165              ;contour draft on, inclined straight line with angle
N50 CW Z-30 X30 R10 TG RNDL(1) ;arc (CW) with end point and circle radius,
                             ;tangential connection with rounding
N60 G1 Z-40               ;contour draft off, horizontal straight line

```

Description Case 2 The starting position Z1, X1 and the angle of the straight lines is known.
 The end point Z3, X3 and the center point coordinates of the circular arc are known.
 The tangential transition point of the straight lines in the circular arc Z2, X2 is calculated.

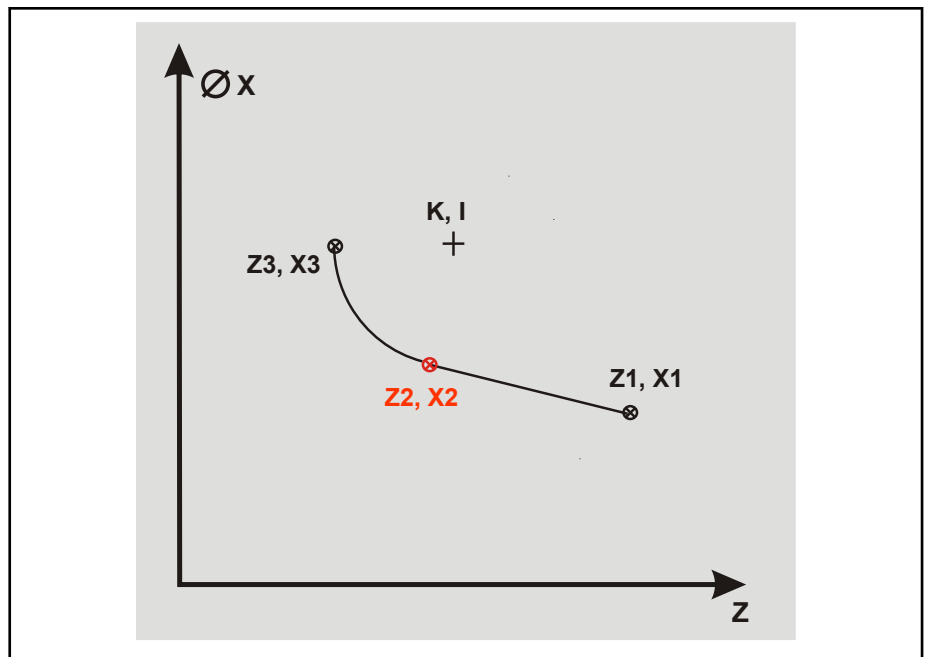


Fig. 12-32: Tangential transition straight line-circle, case 2

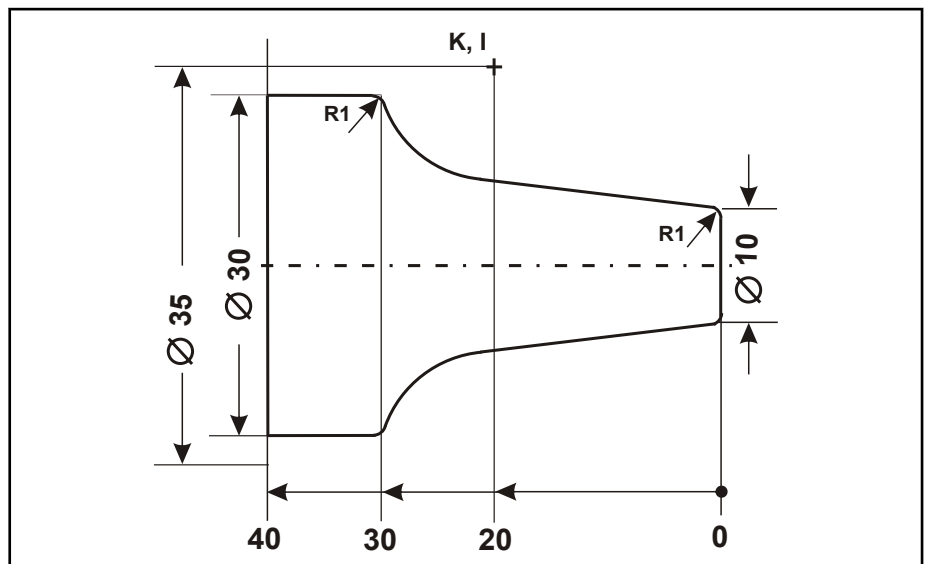


Fig. 12-33: Sketch: Tangential transition straight line-circle, (2)

Program excerpt: Tangential transition straight line-circle, (case 2)

```

N10 G96 S200 M4           ;constant cutting velocity and spindle on
N20 G0 G18 Z0 X0          ;approaching the start position
N30 G95 G1 X10 RNDL(1) F0.1 ;vertical straight line with rounding R1
N40 G31 TG                ;contour on, tangential straight line
                             (without coordinate specification)
N50 CW Z-30 X30 K-20 I35 RNDL(1) ;arc (CW) with end point and center point,
                             tangential connection with rounding
N60 G1 Z-40                ;contour draft off, horizontal straight line

```

Tangential transition circle, straight line

Definition

A circular arc intersects a straight line.

The tangential intersection point is calculated.

Description

The starting position $Z1, X1$ with the circle center point coordinates K, I of the circular arc are known.

The end position $Z3, X3$ is known.

The tangential transition point from the circular arc to the straight line $Z2, X2$ is calculated.

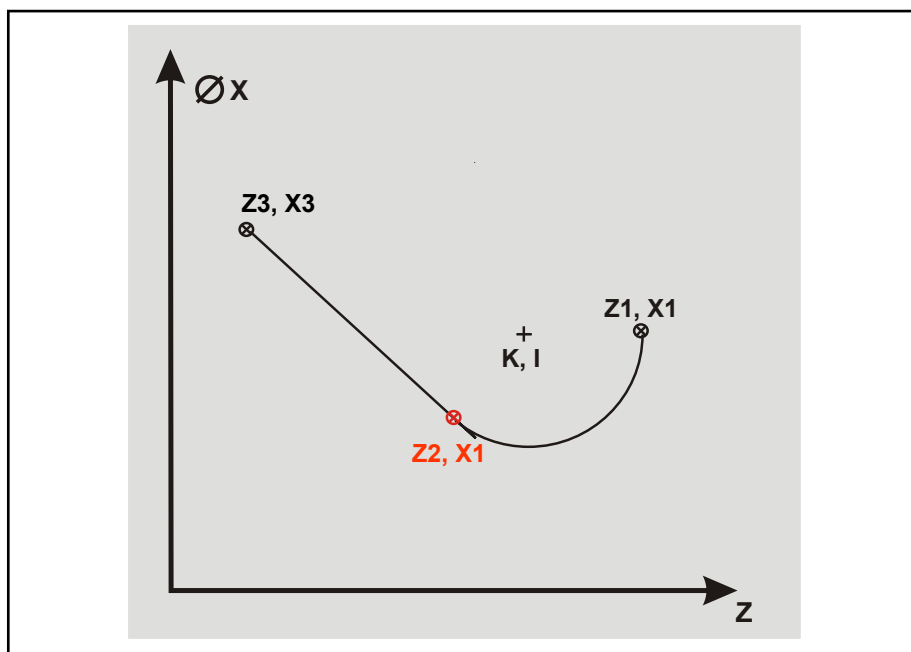


Fig. 12-34: Tangential transition circle, straight line

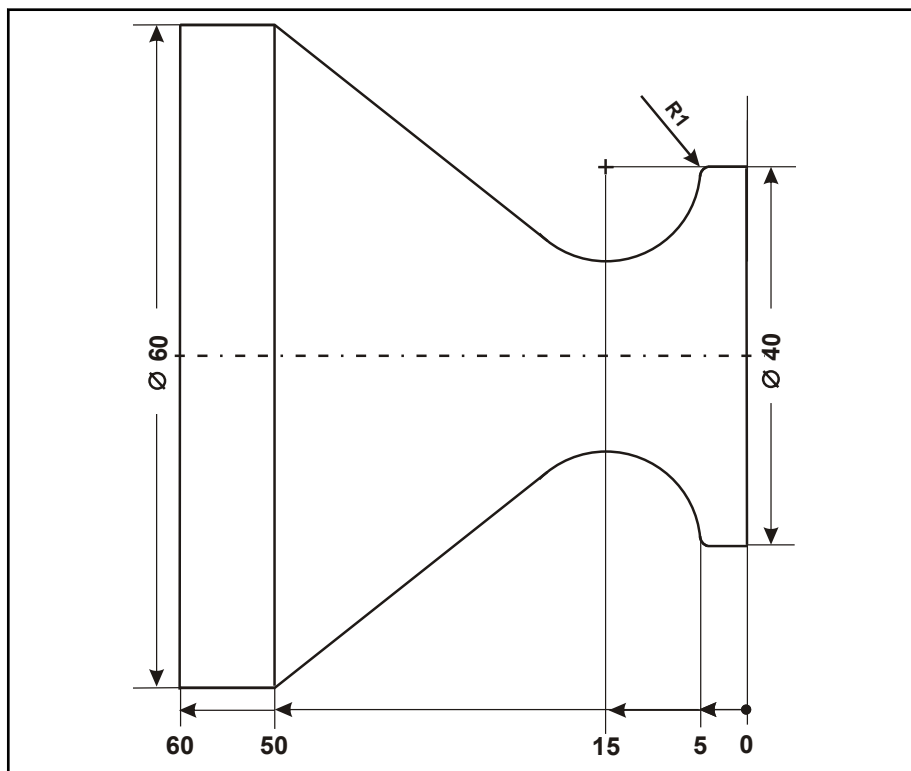


Fig. 12-35: Sketch: Tangential transition circle, straight line

Program excerpt: Tangential transition circle, straight line

```

N10 G96 S200 M4      ;constant cutting velocity and spindle on
N20 G0 G18 X0 Z0     ;approaching the start position
N30 G95 G1 X40 F0.1  ;vertical straight line
N40 Z-5 RNDL(1)      ;horizontal straight line with rounding radius
N50 G31 CW K-15 I40   ;contour draft on, arc (CW) with center point
N60 Z-50 X60 TG      ;end position pf the straight line tangential
                      ;in Z and X as well as TG
N70 G1 Z-60          ;contour off, horizontal straight line

```

Tangential transitions from circle-straight line-circle**Definition**

Two circular arcs are connected to a straight line tangentially.

The contact points of the straight lines at the circular arcs are calculated.

Description

The first circular arcs is determined by the circle center point coordinates K, I.

The second circular arc is determined by the end points Z4, X4 and the circle center point coordinates K, I.

The starting position of the straight lines Z1, X1 is calculated.

The end position of the straight lines Z2, Z2 is calculated.

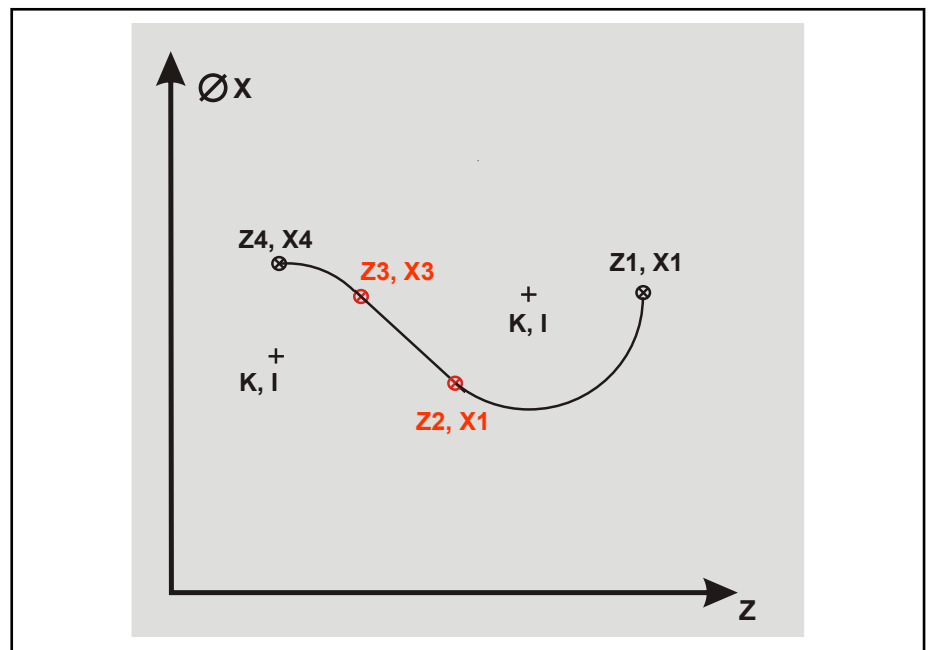


Fig. 12-36: Tangential transitions from circle-straight line-circle

Contour drafts

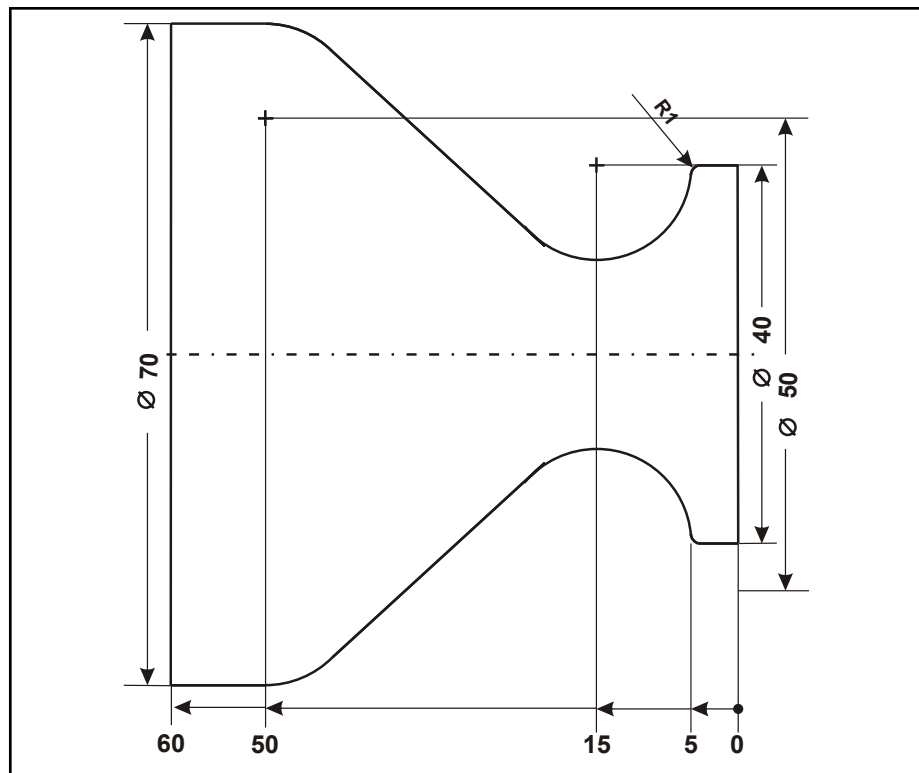


Fig. 12-37: Sketch: Tangential transition circle, straight line, circle

Program excerpt: Tangential transition circle, straight line, circle

```

N10 G96 S200 M4 ;constant cutting velocity and spindle on
N20 G0 G18 Z0 X0 ;approaching the start position
N30 G95 G1 X40 F0.1 ;vertical straight line
N40 Z-5 RNDL(1) ;horizontal straight line with rounding radius
N50 G31 CW K-15 I40 ;contour draft on, arc (CW) with circle center
N60 TG ;tangent to connection circle
N70 CCW Z-50 X70 K-50 I50 ;arc (CCW) with end position, circle center
N80 G1 Z-60 ;contour draft off, horizontal straight line

```

Tangential transitions from circle-circle-circle

Definition Two circular arcs are tangentially connected to a straight line.

The contact points of the three circle are calculated.

Description The first circular arcs is determined by the circle center point coordinates K, I.

The second circular arc is only determined by the radius.

The third circular arc is determined by the end points Z4, X4 and the circle center point coordinates K, I.

The contact points of the second circular arc Z2, X2 and Z3, Z3 are calculated.

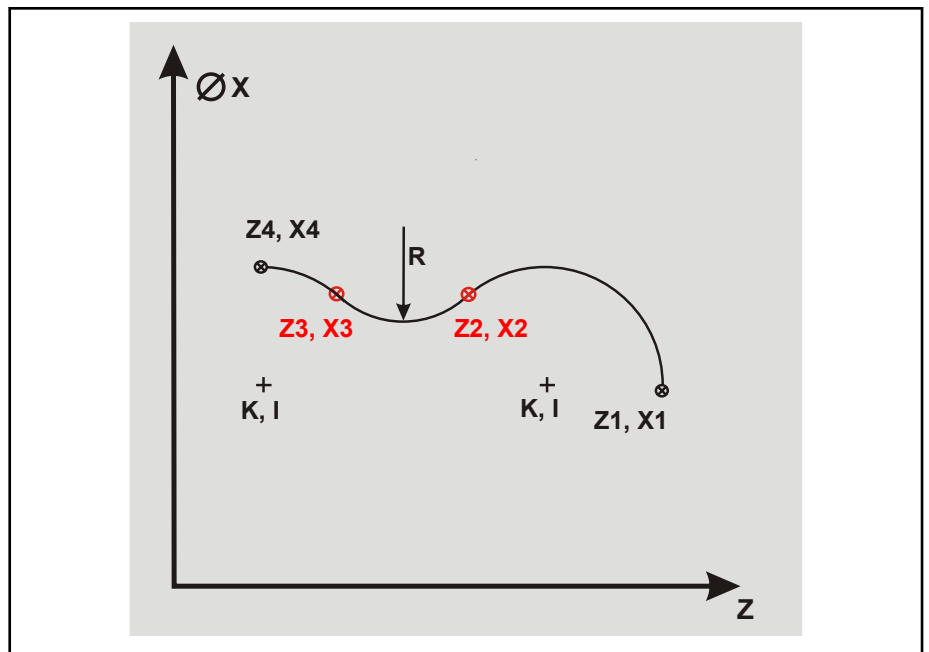


Fig. 12-38: Tangential transitions from circle-circle-circle

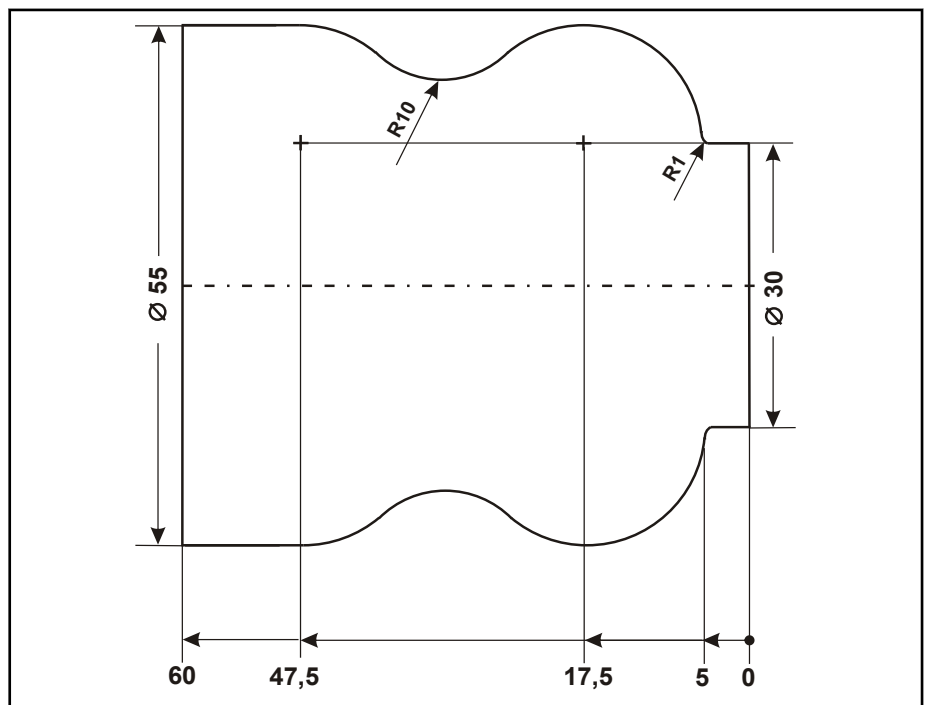


Fig. 12-39: Sketch: Tangential transition circle, circle, circle

Program excerpt: Tangential transition circle, circle, circle

```

N10 G96 S200 M4 ;constant cutting velocity and spindle on
N20 G0 G18 Z0 X0 ;approaching the start position
N30 G95 G1 X30 F0.1 ;vertical straight line
N40 Z-5 RNDL(1) ;horizontal straight line with rounding radius
N50 G31 CCW K-17.5 I30 ;contour draft on, arc (CCW) with
;circle center
N60 TG R10 ;tangential arc with radius
N70 CCW X55 Z-47.5 K-17.5 I30 ;arc (CCW) with circle center and
;end position
N80 G1 Z-60 ;contour draft off, horizontal straight line

```

Intersection transitions

Intersection of straight line-circle

Definition

The intersection of a straight line with a circular arc is calculated.

Description

The starting position Z_1, X_1 and the angle (A_1) are known.

The center point of the circle and the direction of rotation is known.

The end position of the circle in Z_3, X_3 is known.

The intersection of the circle with the straight lines Z_2, X_2 is calculated.

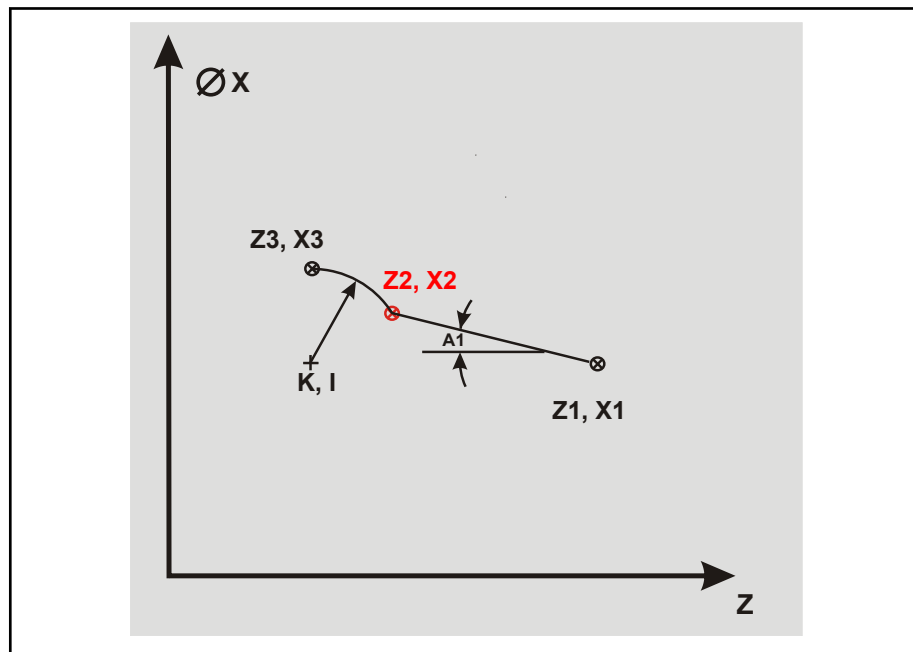


Fig. 12-40: Intersection of straight line-circle

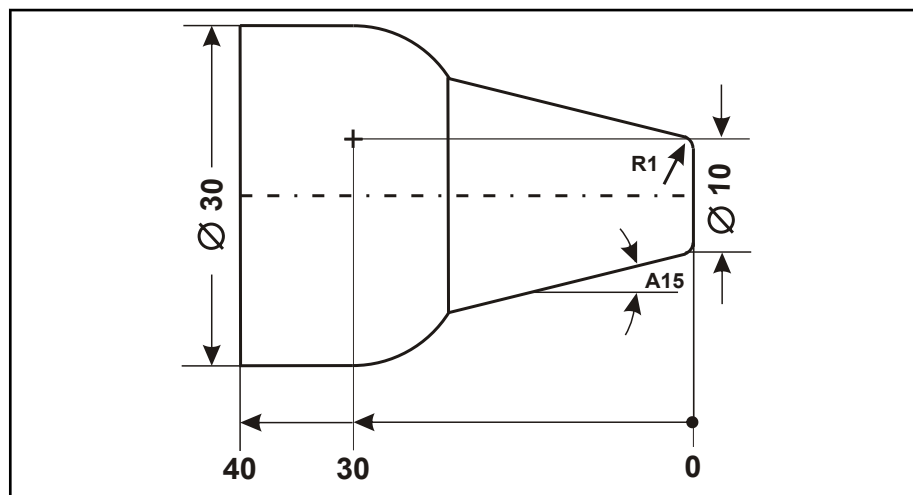


Fig. 12-41: Sketch: Intersection of straight line, circle

Program excerpt: Intersection of straight line, circle

```
N1000 G96 S200 M4           ;constant cutting velocity and spindle on
N1010 G0 G18 Z0 X0          ;approaching the start position
N1020 G95 G1 X10 RNDL(1) F0.1 ;straight line with rounding
```

N1040 G31 A165	;contour draft on, inclined straight line with angle
N1050 CCW Z-30 X30 K-30 I10	;arc (CCW) with end position and circle center
N1060 G1 Z-40	;contour draft off, horizontal straight line

Intersection of circle-straight line

Definition

Intersection of a circular arc with a straight line is calculated.

Description

The starting position Z1, X1 of the circular arc and the circle center point coordinates K, I are known.

The direction of rotation of the circle is determined.

The end position of the straight lines Z3, X3 and the angle are known.

The intersection of the circular arc Z2, X2 with the straight lines is calculated.

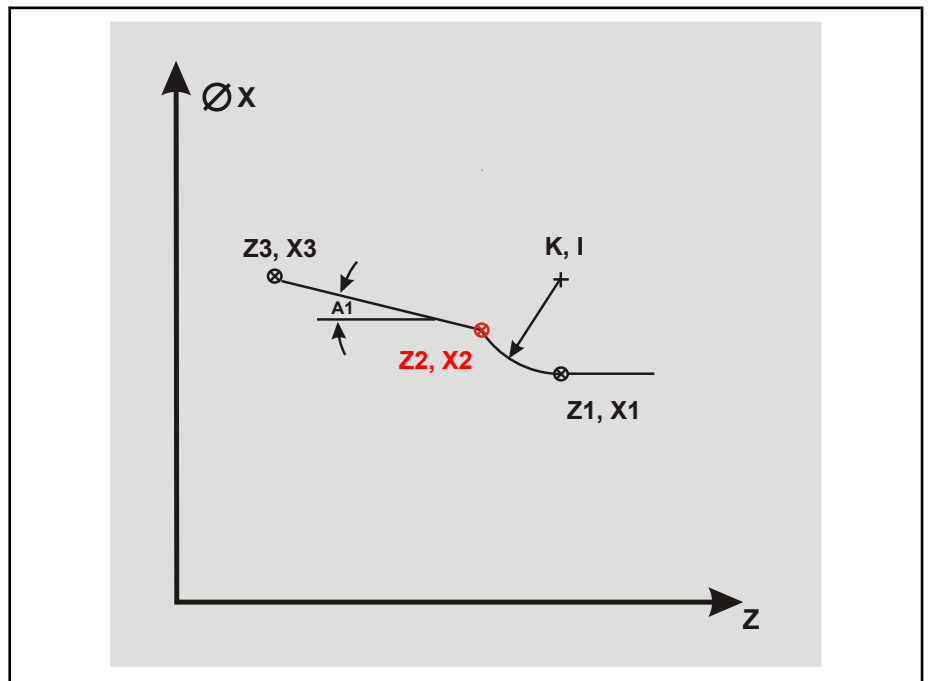


Fig. 12-42: Intersection of circle-straight line

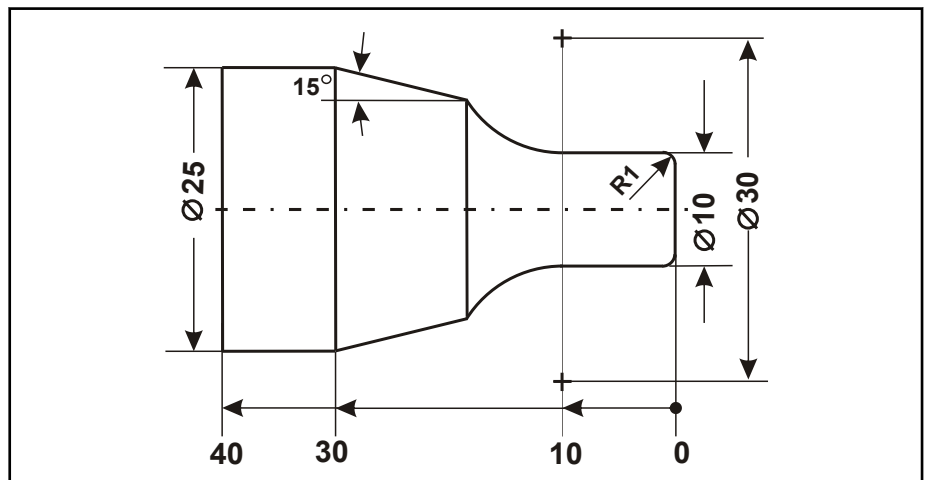


Fig. 12-43: Sketch: Intersection of circle, straight line

Contour drafts

Program excerpt: Intersection of circle, straight line

```

N1000 G96 S200 M4           ; constant cutting velocity and spindle on
N1010 G0 G18 Z0 X0          ; approaching the start position
N1020 G95 G1 X10 RNDL(1) F0.1 ; straight line with rounding
N1030 Z-10                  ; horizontal straight line
N1040 G31 CW K-10 I30       ; contour draft on, arc (CW) with center
N1050 Z-30 X25 A165         ; inclined straight line with end point
                                and angle specification
N1060 G1 Z-40               ; contour draft off, horizontal straight line

```

Intersection of circle-circle

Definition The intersection of two circular arcs is calculated.

Description

The circle center point coordinates K, I of the first circular arc are known.

The end position Z3, X3 of the second circular arc and the center point coordinates I, K of the circular arc are known.

The intersections of both circles Z2, X2 are calculated.

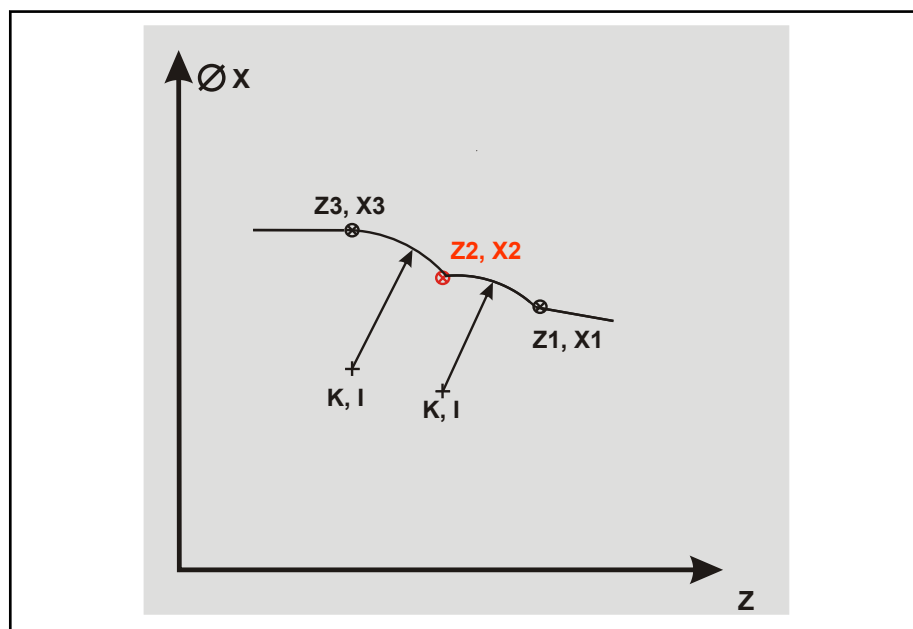


Fig. 12-44: Intersection of circle-circle

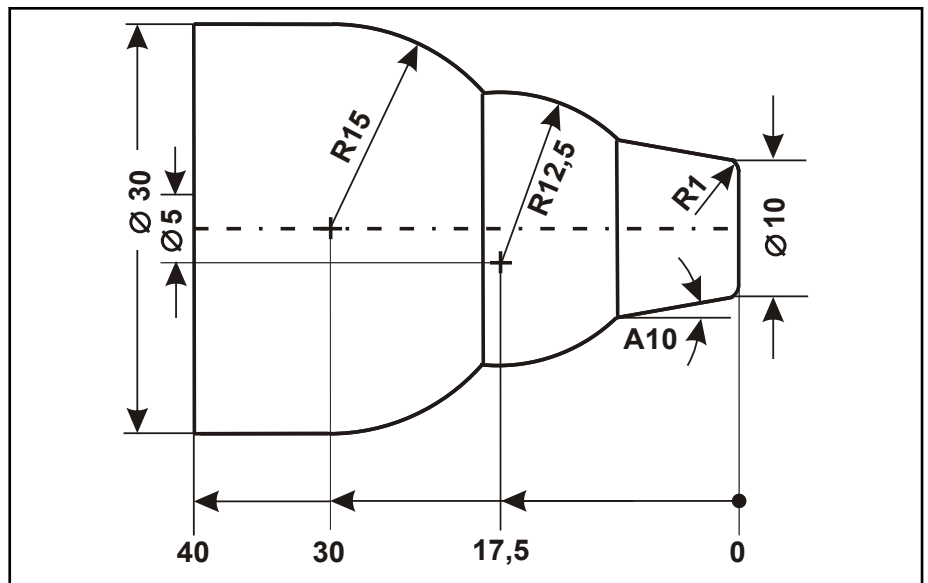


Fig. 12-45: Sketch: Intersection of circle-circle

Program excerpt: Intersection of circle-circle

```

N10 G96 S200 M4           ;constant cutting velocity and spindle on
N20 G0 G18 Z0 X0          ;approaching the starting position
N30 G95 G1 X10 RNDL(1) F0.1 ;vertical straight line with rounding
N40 G31 A170              ;Contour draft on, straight line inclined with angle
                           ;specification
N50 CCW K-17.5 I-5 R12.5   ;Arc (CCW) with center point and radius
N60 CCW Z-30 X30 K-30 I0   ;Arc (CCW) with end and center point
N70 G1 Z-40                ;contour draft off, horizontal straight line

```


13 G230 Swiveling

Syntax G230(TNR, TSD, OB, OX, OY, OZ, A1, A2, A3, RS, PB, XX, XY, XZ, YX, YY, YZ, R1, R2, OM, OA1, OA2, OS, OT, RM, RI, AX, AY, AZ, DIR)

Short description The "Swiveling" cycle (G230) activates an inclined plane (and can swivel there). 2½D machining can take place in this plane.

Overview

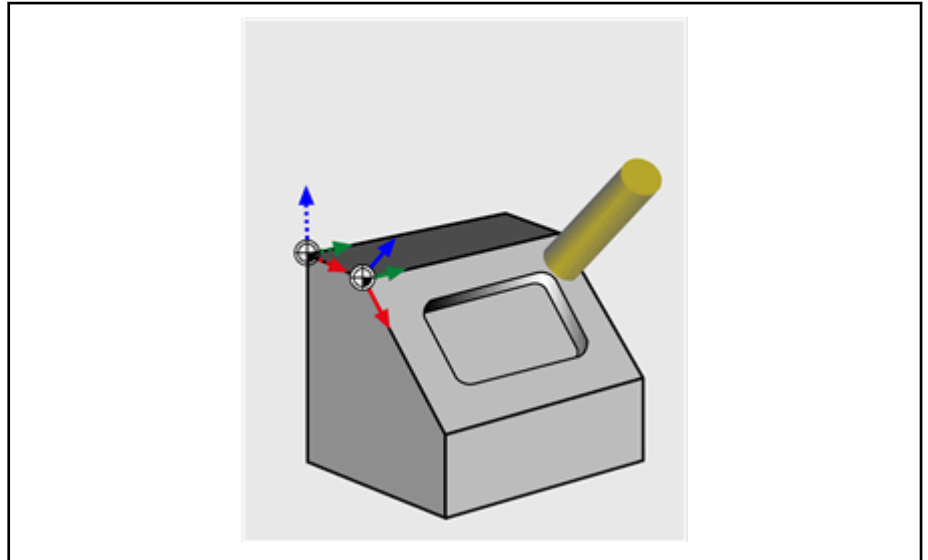


Fig. 13-1: Cycle "G230 Swiveling"

Parameter

G230	Swiveling	G230(TNR, TSD, OB, OX, OY, OZ, A1, A2, A3, RS, PB, XX, XY, XZ, YX, YY, YZ, R1, R2, OM, OA1, OA2, OS, OT, RM, RI, AX, AY, AZ, DIR)		
TNR	Number of the memory with transformation data	optional	Integer	1 to 20 Prest value, can be configured in the system data variable "SysSwivel"
TSD	Name of the system date with transformation data	optional	String	max. 100 characters
OB	Reference system for offset and rotation	optional	Integer	1 BCS + fix in space 2 BCS + following 3 act. plane + fix in space 4 act. plane + following 1 is preset
OX, OY, OZ	New zero point	optional	Real	0 preset
A1, A2, A3	Angle 1 to 3 for rotations	optional	Real	0 preset
RS	Sequence and axes for rotations	optional	String(3)	XYZ ... ZYX ZYX preset

G230 Swiveling

PB	Reference system for the spatial coordinates	optional	Integer	1 all points in the BCS 2 coordinate jump in the BCS, the remaining points are relative to the coordinate jump 3 all points in current plane 4 coordinate jump in current plane, the remaining points are relative to the coordinate jump 2 is the default setting
XX, XY, XZ	Point in x-direction for spatial coordinates	optional	Real	0 preset
YX, YY, YZ	Point in y-direction for spatial coordinates	optional	Real	0 preset
R1, R2	Rotary axis values for "rotary axes, direct" R2 only for 5 axis kinematics	optional	Real	0 preset
AX, AY, AZ	Additional offset	optional	Real	0 preset
OM	Orientation mode	optional	Integer	1 not orienting 2 swiveling around the tool tip 3 only rotary axes 2 is the default setting
OT	Reference to the rotary angle for orientation purposes	optional	Integer	1 orient following the angles 2 orient following the angles fixed in space 3 same orientation as on old plane 2 is the default setting
OA1, OA2	Orientation angle	optional	Real	0 preset
OS	Sequence of the rotations of orientations	optional	String(2)	XY, XZ, YX, YZ, ZX, ZY ZY preset
RM	Retract mode	optional	Integer	1 no retracting 2 retract axis Z 3 retract axes Z, XY 1 is preset
RI	Retract increment in tool direction	optional	Real	0 preset
DIR	Preferred direction of the second rotary axis	optional	Integer	0 shortest swivel distance Traverse -1 in the negative range Traverse 1 in the positive range 0 preset

Tab. 13-1: Parameter G230

Description TNR transformation data number

Number of the memory with transformation data in the configuration data

TSD transformation data system date

Name of the system date variables with transformation data

OB reference system for offset and rotation

Reference system for angle and zero point at plane definition by offset and rotation

OX, OY, OZ new zero point

Coordinate origin (zero point) of the new, swiveled coordinate system

A1, A2, A3 angles for rotations

Angles 1 to 3 for plane definition by offset and rotation

RS sequence & axes for rotations

Sequence of rotations and axes along which is to be turned for plane definition by offset and rotation

PB reference system for the spatial coordinates

Reference system of vectors/points for the plane definition by spatial coordinates

XX, XY, XZ point/vector in x-direction for spatial coordinates

Vector/point along the new x-axis for plane definition by spatial coordinates

YX, YY, YZ point/vector in y-direction for spatial coordinates

Vector/point along the new y-axis for plane definition by spatial coordinates

R1, R2 rotary axis values for "rotary axes, direct"

Axis positions of the rotary axes 1 and 2 for plane definition by "rotary axes, direct"

AX, AY, AZ additional offset

Additional offset in the swiveled plane

OM orientation mode

Specification how the swiveling motion is to be executed

OT reference to the rotary angle for orientation purposes

Reference system when specifying the rotary angles for an additional orientation on the swiveled plane

OA1, OA2 orientation angle

Orientation angles for an additional orientation on the swiveled plane

OS sequence of the rotations of orientations

Sequence of the rotations and axes along which is to be rotated for an additional orientation of the swiveled plane

RM retract mode

Specification how it is to be retracted from the workpiece without swiveling

RI retract increment in tool direction

Length traveled in opposite direction of the workpiece before swiveling

DIR Preferred direction of the second rotary axis

Specify preferred direction of the second rotary axis during swiveling

Special features and restrictions

- No axis transformation at point of action AT2 may be active
- No tool radius correction (G41/G42) may be active
- No input help (SCL, MIR, ROT, SHT) may be active
- A 5-axis transformation has to be available for the kinematics

Currently supported kinematics:

- 4-axis transformation LLLB/LLLA (3131002)
- 4-axis transformation BLLL/ALLL (3131003)

G230 Swiveling

- 5-axis transformation LLLCB/LLLCA (3232201)
- 5-axis transformation CLLLL/CLLLB (3232202)
- 5-axis transformation CALLL/CBLLL (3232203)
- Cardanic 5-axis transformation CLLLLB (3232212)
- Cardanic 5-axis transformation LLLCB (3232211)
- If rapid traverse has been active during the cycle call (G0), swiveling takes place in rapid traverse. If another motion function has been active, the currently active feed is used for the swiveling motion

Preparation Before using the swiveling cycle, report the axis configuration to the MTX. The commissioner stores the relevant parameters in the NC configuration (TRA/AxTrafo[1..20], MACODA 1030 001xx). The parameters can also be stored in an SD variable (SD type "SysAxTrafo_t"). The kinematics corresponds to an axis transformation of the point of action AT2. For a description of the valid machine types and the configuration parameters, refer to the chapter "Axis transformation" in the manual "Functional Description".

The retract position has to be stored in the permanent system date "SysSwivel".

If only one axis transformation is required at the machine, the number of this axis transformation can be entered in the permanent system date "SysSwivel". Thus, parameter TNR can be omitted during the cycle call.

The importance of the channel axes when operating (CHAN/Ch[k]/Coord/Acs/ChAx[1..8]/ChAxClass, MACODA 7010 00030) has to be set correctly.



It is recommended to write G153 and CRD() to the "state after channel home position" (machine parameter CHAN/Ch[k]/Ini/ChResetState). Thus, a reset of the initial machine state (without inclined plane or transformation) can be restored.

Description The cycle operates with any axis name. In the following description, the axes x, y and z (main axis, secondary axis and infeed axis) are on the currently active plane.



In the swiveled plane, only the tool length correction in the direction of the tool should be used (often, L3 along the Z-axis). Off-center tool corrections are not supported.

The swiveling cycle sequence is described in the following figure:

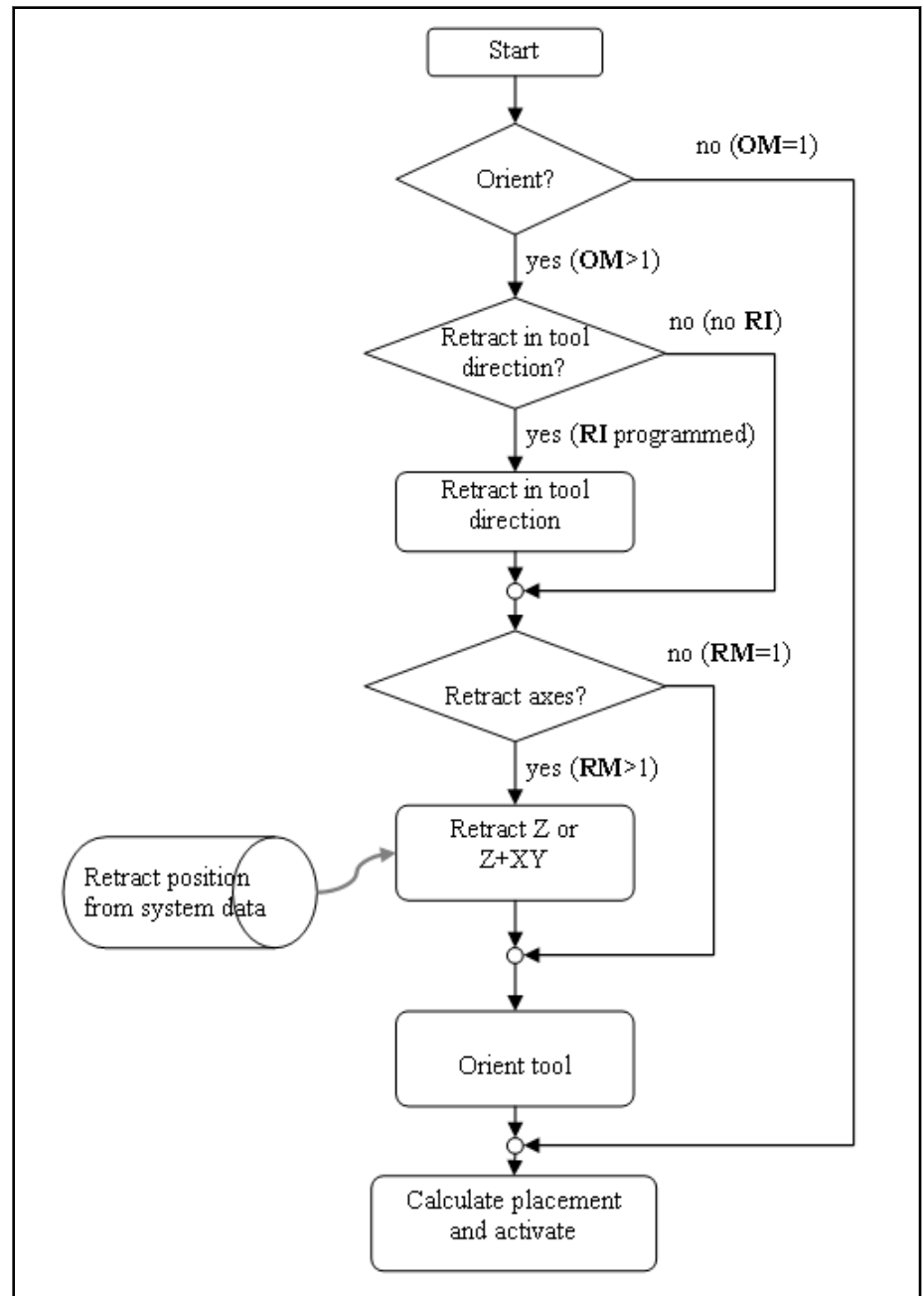


Fig. 13-2: G230 sequence

In the first step, the cycle (optional) moves away from the component. For that step, an incremental distance along the current z-axis can be programmed (RI). A positive value means moving away from the workpiece.

Subsequently, it can be traveled along the machine axes to a swiveling position. This position has to be stored before in the system data "SD.SysSwivel.Cfg". The system data can also be removed from the part program. It can be selected via the parameter RM whether only the z-axis is traveled or first the z-axis and then the x-axis and the y-axis. The position is referenced in the machine coordinate system (G76).

G230 Swiveling

The cycle calculates a swiveled plane from the programmed parameters. The rotary axes of the machine (optional) are moved until the tool is in a vertical position on the plane.

After the cycle has been processed, two placements (refer to the "Functional Description", chapter "Placement": Inclined Plane G152.1...G159.5) are active for the swiveled plane. The first placement comprises an offset and a rotation (corresponds to the parameters *AX*, *AY* and *AZ*). Due to these placements, programming x and y causes a motion in the inclined plane. If z is programmed, the machine moves vertically to the inclined plane. Changing the rotary axis positions is not reasonable. To swivel to another plane, program the cycle again.

G152.1 and G152.2 are always used as placements. Existing values are overwritten. The calculated offsets and rotations are saved in the permanent system date "SysSwivel". All other placements are disabled.

To restart the last placement of the swiveling cycle after an NC restart, the auxiliary cycle "RestoreSwiveling" (case sensitive!) can be programmed.

To disable the swiveled plane after processing, program G230 () (without parameters). This resets the system date "SysSwivel".

Specifying kinematic data

The kinematics of the machine has to be known to the swiveling cycle to calculate the inclined plane correctly. Therefore, enter the corresponding parameters (as for an axis transformation) into the configuration data. Axis transformations can be configured in the configuration data (parameter *TNR* with the index 1 to 20) or in a system date *TSD* with SD variable name).

If neither *TNR* nor *TSD* are programmed, the cycle retrieves the axis transformation number programmed from the "SD.SysSwivel.Cfg.TRN" system date. An invalid number generates a runtime error.

Parameters of the plane

The inclined plane can be programmed using four different variants:

- With three measured points
- With one offset and up to three rotations
- With three spatial coordinates
- With the position values of the two rotary axes

One variant has to be programmed in each cycle.

Measured points

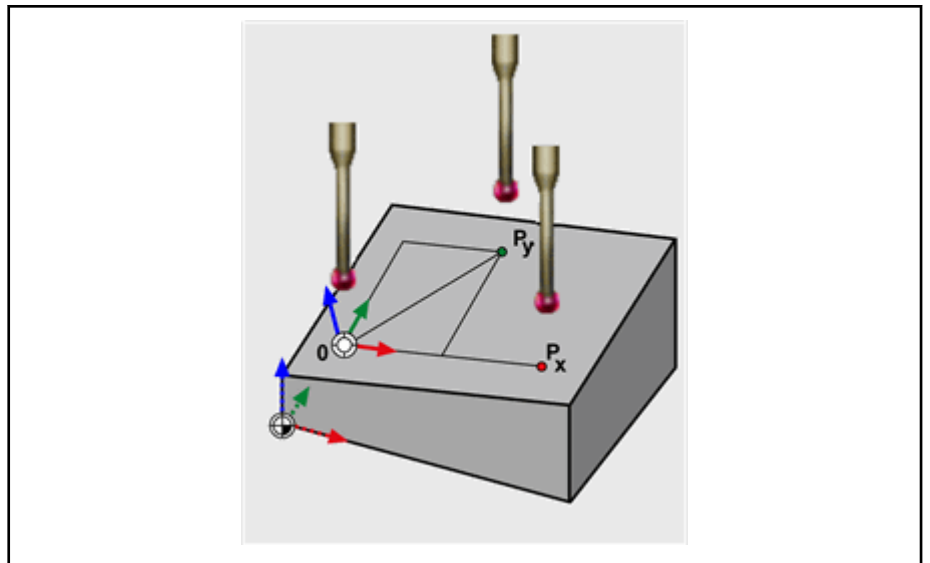


Fig. 13-3: Plane definition on the basis of measured points

Measured points can be entered in the `SD.SysPrResult.PMT` system data. This is realized using the manual measuring function "Inclined plane" <Measuring for G230>. To enable this inclined plane, the swivel cycle is called with parameter MS1.

Offset and rotation

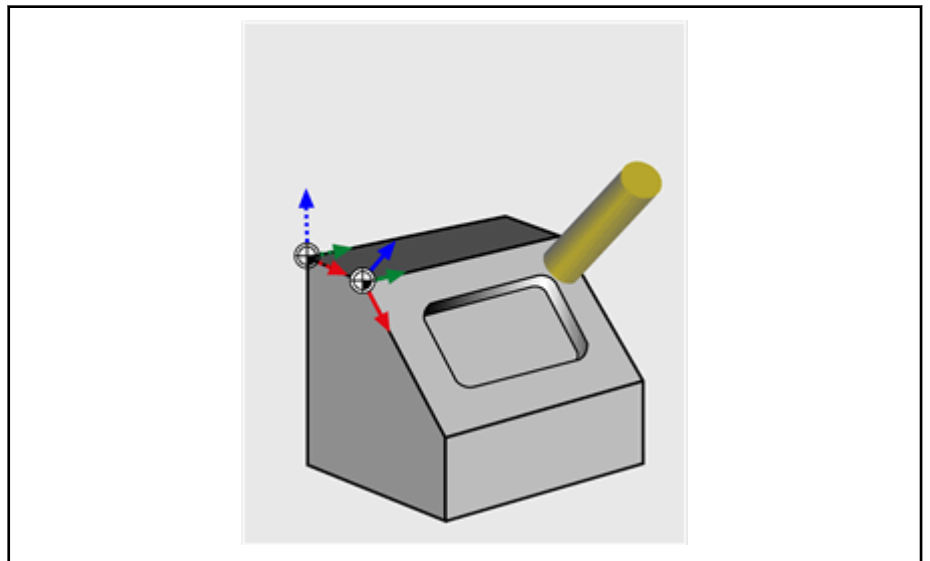


Fig. 13-4: Plane definition with offset and rotation

The zero point of the inclined plane is specified via the parameters `OX`, `OY` and `OZ`. There are different rotations: Rotations calculated along axes fixed in space or along axes of the new coordinate system (following). The parameter `OB` specifies the reference point for the new zero point and the effect of the rotations.

Not more than three rotations may be specified. The angles of the rotations are programmed using the parameters `A1`, `A2` and `A3`. The number of rotations and along which axes is to be rotated is specified as character string in the parameter `RS`.

Less than three letters may be programmed in RS . If "ZYZ" is specified or if RS has not been programmed and $OB=2$ or $OB=4$, the angles correspond to the Euler angles.

Three spatial coordinates

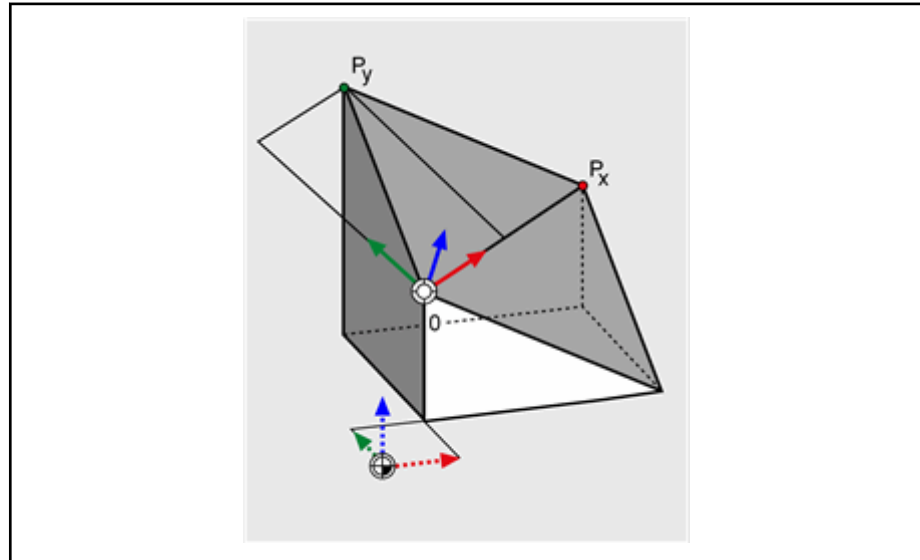


Fig. 13-5: Plane definition with three points

Three spatial coordinates define one plane. The zero point of the new plane is specified via the parameters OX , OY and OZ . The x-axis of the new plane is specified from the zero point towards the point (XX , XY and XZ). The third point (YX , YY and YZ) specifies the direction of the y-axis.

If neither XX , XY nor XZ is programmed, the cycle uses $XX=1$, $XY=0$ and $XZ=0$ (x-axis remains equal). If not programmed, the following applies: YX , YY and YZ any $YX=0$, $YY=1$ and $YZ=0$ (y-axis remains equal). This is only to be used if - as reference system to the points - (parameter PB) to "relative to zero point" (value 0).

The points can also be related to the basic coordinate system or to the currently inclined plane (parameter PB).

Position values of two rotary axes

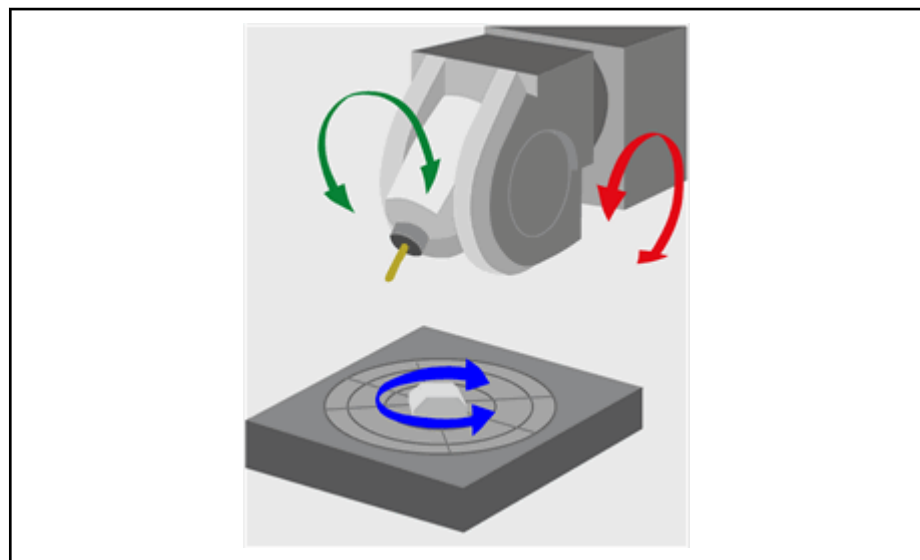


Fig. 13-6: Plane definition with two rotary axis positions

The direction of the tool is determined by the position values of two rotary axes of a 5-axis machine. Thus, the swiveling cycle can calculate a plane vertical to the tool. The axis positions are programmed in the parameters `R1` and `R2`. `R1` is the first rotary axis located in the kinematic configuration (assignment: system axes) and `R2` is the second rotary axis.

The zero point of the new plane is specified via the parameters `OX`, `OY` and `OZ`.

Swiveling

If the inclined plane is determined, the cycle can swivel the tool accordingly. The parameter `OM` is used to specify whether there is any swiveling at all, whether swiveling takes place around the tool tip or whether only the rotary axes are moved while swiveling.

Orientation on the inclined plane

The tool can be additionally oriented on the inclined plane. The tool is then not vertical to the plane. The orientation angles are programmed in the parameters `OA1` and `OA2`. The parameter `OS` determines the number of calculated rotations (one or two) and along which axes is to be swiveled.

Specify the reference to the orientation angle in the parameter `OT`.

If no orientation is specified, the orientation is vertical to the inclined plane.



If `OM1` is programmed, the cycle generates no motion (and it is also not retracted).

Additional zero point offset

An additional zero point offset can be programmed in the inclined plane using the parameters `AX`, `AY` and `AZ`.

Preferred direction of the second rotary axis

If an inclined plane is specified for a 5-axis kinematics, there are two options to swivel to the kinematics. This variant differs by the sign of the second rotary axis (e.g. A-axis or B-axis). The positions of the first rotary axis differ by 180°.

Example:

Possible axis positions for the same inclined plane

In the example machine (LLRR), the rotary axes C and B have to be available. Requirement: Swiveling to an inclined plane that has been generated by a rotation around the x-axis by 30°.

Option 1: C=90°, B=-30°

Option 2: C=270°, B=30°

If the cycle is to execute the swivel motion (parameter `OM=2` or `OM=3`), a motion range can be specified in parameter `DIR` for the second rotary axis.

If `DIR` is not programmed, the cycle automatically selects the range with the shortest distance. The current rotary axis position at the time of the cycle call determines which variant is selected. By smart positioning of the rotary axes, the user can thus influence the area.

If `DIR=1` is programmed, the cycle executes a motion for pre-positioning prior to the actual swivelling (if required) to ensure that the second axis swivels to the positive area (option 2 in the example).

If `DIR=1` is programmed, the cycle executes a motion for pre-positioning prior to the actual swivelling (if required) to ensure that the second axis swivels to the positive area (option 2 in the example).

G230 Swiveling

⚠ WARNING

If `DIR=1` or `DIR=2` is programmed, a pre-positioning motion is executed (if required). This motion generally does not take place around the tool tip (irrespective of the programmed orientation mode `OM`).

Therefore, retract from the part to be machined.

Example 1: Tool swiveling

Task: Drill eight holes (depth 5 mm) in a circle in an inclined plane. The center point of the circle is X30, Y20, Z30. The radius of the circle is 10 mm. The inclined plane is 45° inclined on the corner of the cube.

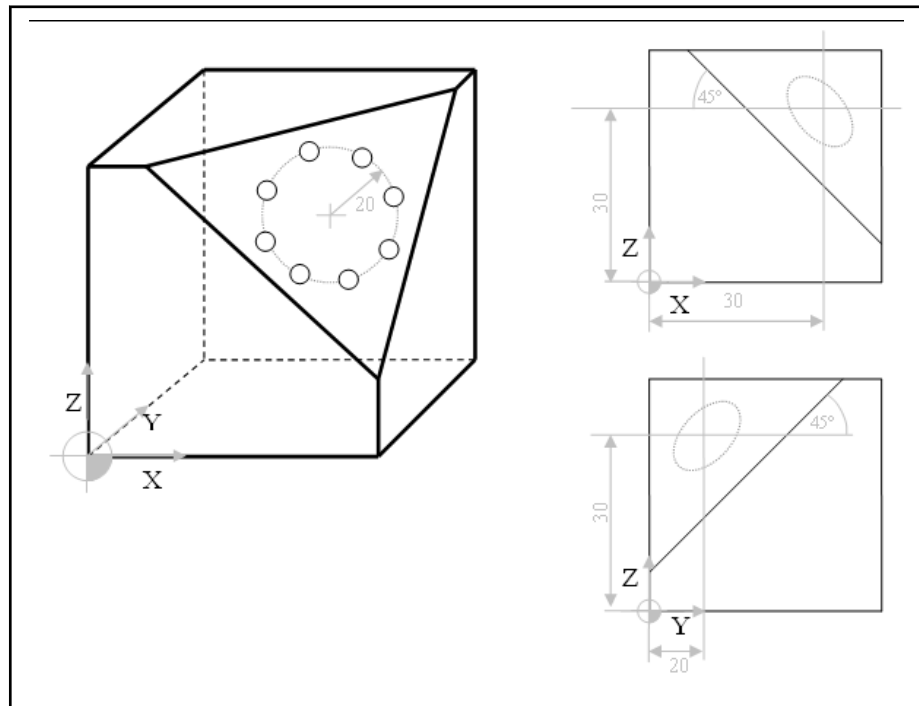


Fig. 13-7: Example 1 Swiveling (inclined plane 1 axes)

Program for G230 Swiveling (inclined plane 1 axis)

```

N10 T1 M6 ; Tool change
N20 G17 S2500 M3 F3000 ; Initialize
N30 G230(TNR1, A1 45, A2 45, RS "XZ", OX30, OY20, OZ30, RM3) ; Swiveling
N40 G81(IX=Z, SL2, DT-5, DW1, RL3) F800 ; Drilling cycle + feed
N50 G113(MCP0, SCP0, PR20, NH8) ; Hole circle cycle
N60 G80 Z10 M5 ; Leave contour, delete modal (G81) call
N70 G230() ; Delete inclined plane

```

Example 2: Tool swiveling Task: Engrave the text "MTX" in an inclined plane.

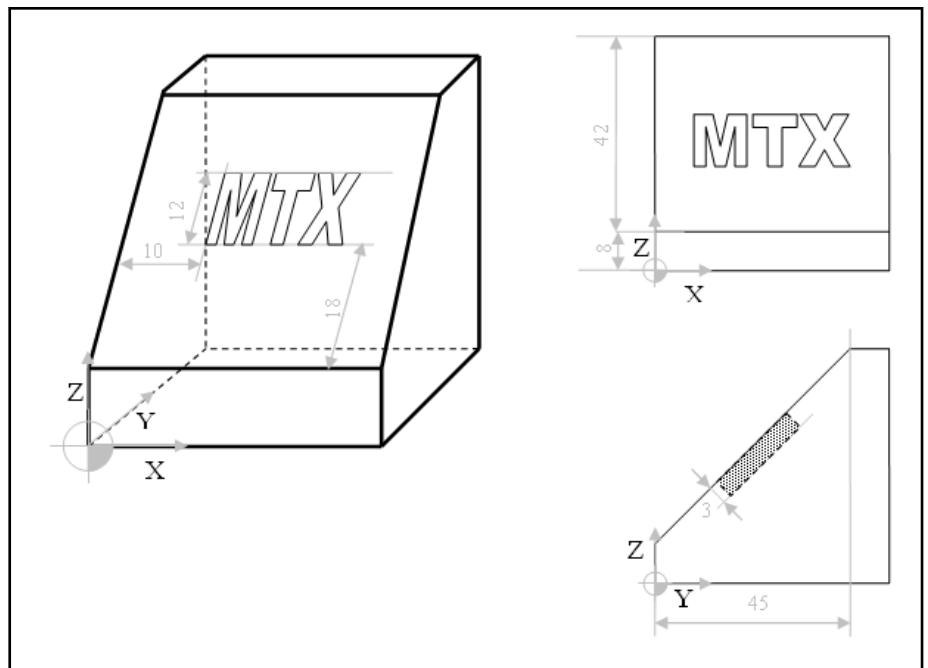


Fig. 13-8: Example 2 Swiveling (inclined plane 2 axes)

Program for G230 Swiveling (inclined plane 2 axes)

```

N10 T1 M6                                ; Tool change
N20 G17 G0 X0 Y0 Z100 AB0 AC0 S2500 M3 F3000 ; Prepositioning
N30 G230(TNR1, OZ8, YY45, YZ42, AX10, AY18) ; Swiveling
N40 G201(TX MTX, LE12, MSP0, SSP0, SL1, DT-3, FI200, FF500) ; Engraving text
N50 SHT() SCL() MIR() ROT() PLS()          ; of G201; switched on if required
N60 G0 Z20                                ; Retract
N70 G230()                                ; Delete inclined plane

```

Example 3: Tool swiveling incl. orient tool

Task: Milling of a guideway via tool orientation with one axis.

G230 Swiveling

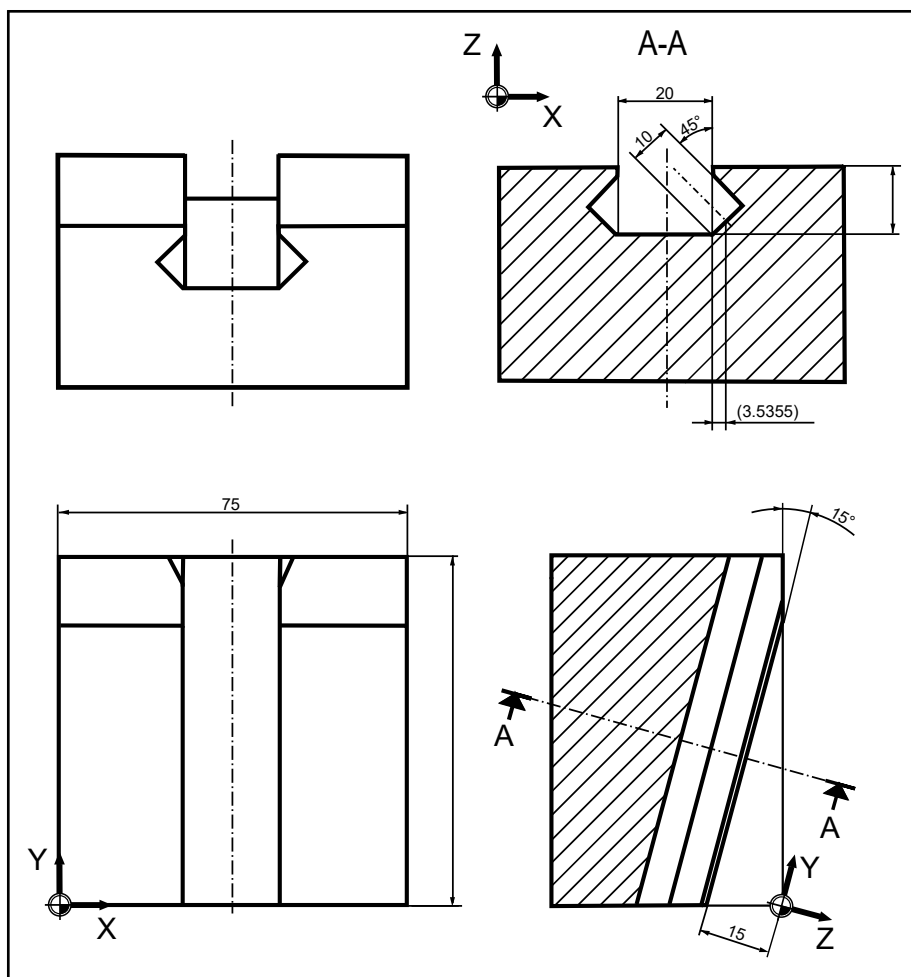


Fig. 13-9: Example 3 Swiveling (tool oriented with 1 axis)

Program for G230 Swiveling (tool oriented with 1 axis)

```

;
N110 G0 G54 G17 G94 F10000 ;workpiece zero point
; --- Swiveling by offset and rotation ---
N200 T[3]
N210 G48
N110 G230 (TNR1, RS X, A1 15, OM3, DIR1)
N220 G47
; Groove with boring groove milling tool D20
N250 G0 X37.5 Y90 Z10 S2500 M3
N260 G1 Z-31 F3000
N270 Y-20 F500
N280 G0 Z100
N330 G230()

N200 T[27]
N220 G54 G48
; and now with additive tool orientation +45°
N110 G230 (TNR3, RS X, A1 15, OM3, DIR1, OS Y, OA1 45)
;
100 DIM EULER!(3)
110 EULER!(1)=0 : EULER!(2)=0 : EULER!(3)=0
120 ABSROT2EUL(EULER!,"Y",45) ;Conversion tool orientation +45 ° to Euler angle
N230 STO([EULER!(1)], [EULER!(2)], [EULER!(3)]) ;Conversion of tool lengths to tool orientation +45 °
N240 G47

; Groove with boring groove milling tool D10
N210 G0 X23.9645 Y90 Z10 S3800 M3
N260 G1 Z[-31+3.5355] F3000
N270 Y-20 F500
G0 Z100
N330 G230()

```

```

N340 G54 G48
;
; and now with additive tool orientation -45 °
N110 G230 (TNR1, RS X, A1 15, OM3, DIR1, OS Y, OA1 -45)
110 EULER!(1)=0 : EULER!(2)=0 : EULER!(3)=0
120 ABSROT2EUL(EULER!,"Y",-45) ;Conversion tool orientation -45° to Euler angle
N230 STO([EULER!(1)], [EULER!(2)], [EULER!(3)]) ;Conversion of tool lengths to the tool orientation -45°
N240 G47

; Groove with boring groove milling tool D10
N210 G0 X51.0355 Y90 Z10 S3800 M3
N260 G1 Z[-31+3.5355] F3000
N270 Y-20 F500
G0 Z100
N330 G230()
N610 M30

```

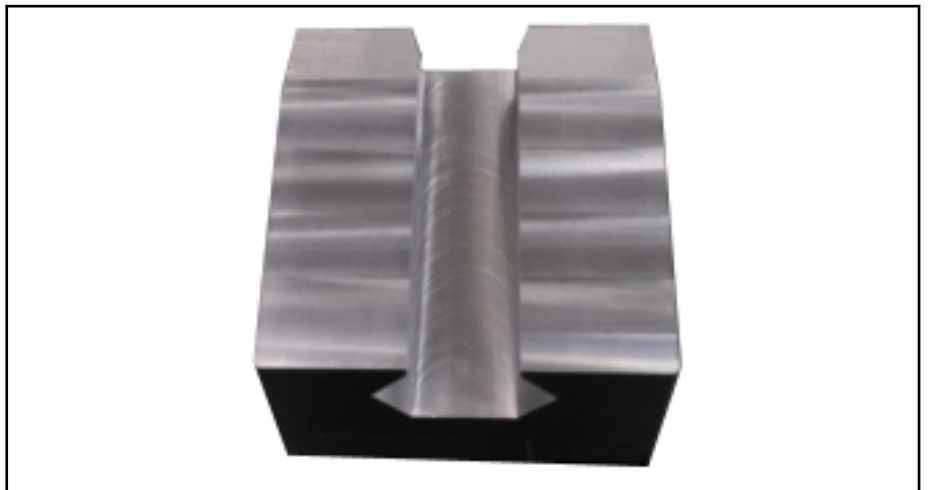


Fig. 13-10: Machining result, example 3

Example 4: Tool swiveling incl. ori-ent tool

When using tools that do not cut through the tool center (e.g. ball cutter, radius form cutter, slot cutter, etc.), it is recommended to position the tool under an angle to the normalized space vector.

Task: Milling of a guideway via tool orientation with two axes.

Program for G230 Swiveling (tool oriented with 2 axes)

```

;
N110 G0 G54 G17 G94 F10000 ;Workpiece zero point
; --- Swiveling by offset and rotation ---
N200 T[3]
N210 G48
N110 G230 (TNR1, RS X, A1 15, OM3, DIR1)
N220 G47
; Groove with boring groove milling tool D20
N250 G0 X37.5 Y90 Z10 S2500 M3
N260 G1 Z-31 F3000
N270 Y-20 F500
N280 G0 Z100
N330 G230()
N340 G48 G54
;
; and now with additive tool orientation +Y 20 und X 15°
N110 G230 (TNR1, RS X, A1 15, OM3, DIR1, OS YX, OA1 20, OA2 15)
100 DIM EULER!(3)
110 EULER!(1)=0 : EULER!(2)=0 : EULER!(3)=0
120 ABSROT2EUL(EULER!,"Y",20,"X",15) ;Conversion tool orientation +45° to Euler angle
N230 STO([EULER!(1)], [EULER!(2)], [EULER!(3)])
;Conversion of tool lengths to the tool orientation +45°
N240 G47
;
; Groove with boring groove milling tool D20
N210 G0 X37.5 Y90 Z10 S3500 M3
N260 G1 Z-15.2 F3000
N270 Y-20 F800
G0 Z100

```

G230 Swiveling

N330 G230 ()
N610 M30

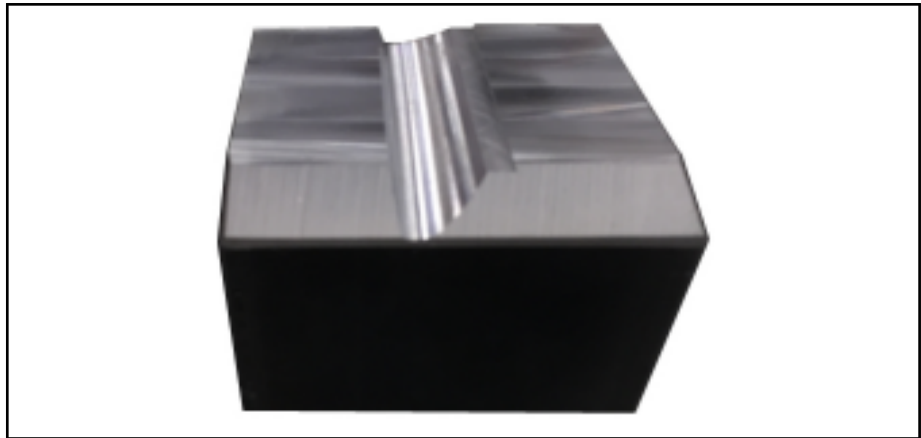


Fig. 13-11: Machining result, example 4

14 Technology cycles

14.1 General information

Technology cycles are programming tools allowing the effectively implementation of predefined technology sequences in an NC program. Input masks support partially.

14.2 G501 TFS Cycle turning

Brief description Support cycle to initialize NC programs in the "turning" technology. Tool, feed and spindle data are set.

Overview figure

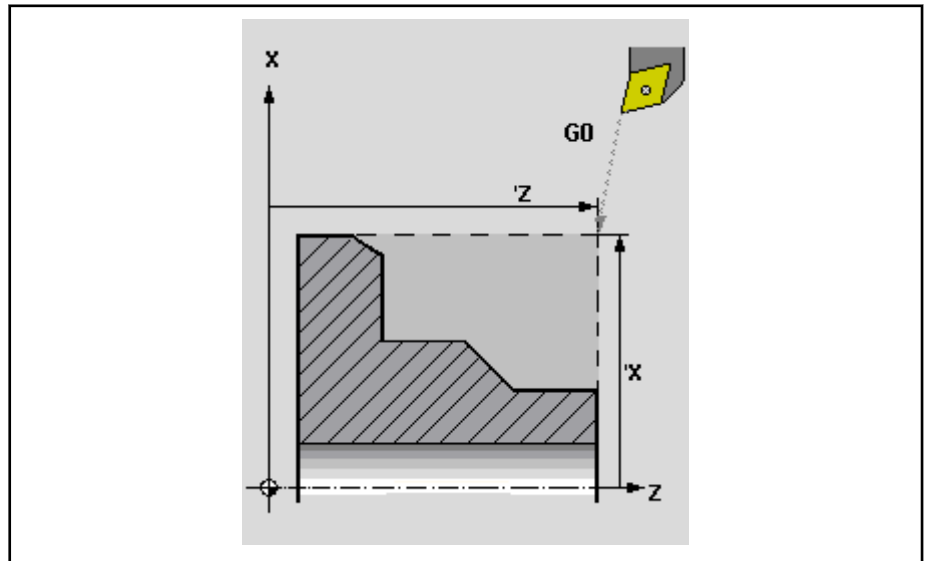


Fig. 14-1: Cycle G501 TFS Turning

Parameter

Parameter	Name	Properties	Values	Pre-assignment
T	Tool number	Mandatory		
ED	Tool edge number	Optional		
F	Feed	Mandatory		
FC	Feed condition	Optional	G94 , G95	G95
S	Spindle speed	Mandatory		
DIR	Direction of spindle rotation	Mandatory	M3, M4	
SMX	Maximum spindle velocity	Optional		
CS	Constant cutting velocity	Optional	G96 , G97	G96
ZG	ZO group	Mandatory	G54..G59	G54
ZB	ZO bank	Optional	1..5	1
X	X-position	Optional		

Parameter	Name	Properties	Values	Pre-assignment
Z	Z-position	Optional		
POS	Strategy to approach the starting position	Optional	1 - X then Z 2 - Z then X 3 - Both (XZ)	3
TCH	Strategy to approach the tool change position	Optional	0 - None 1 - X then Z 2 - Z then X 3 - Both	3

Tab. 14-1: Parameter call

Description**T-tool number and ED edge number:**

Select tool and set active edge

F feed and FC feed conditions:

Set feed and feed conditions. The feed/revolution G95 is pre-assigned

S spindle velocity, DIR direction of spindle rotation and SMX maximum spindle velocity

Set the spindle velocity and the direction of rotation of the spindle for machining. The upper limit of the permitted speed range is specified with SMX.

CS constant cutting velocity:

Set whether the machining takes place with or without the cutting velocity constant

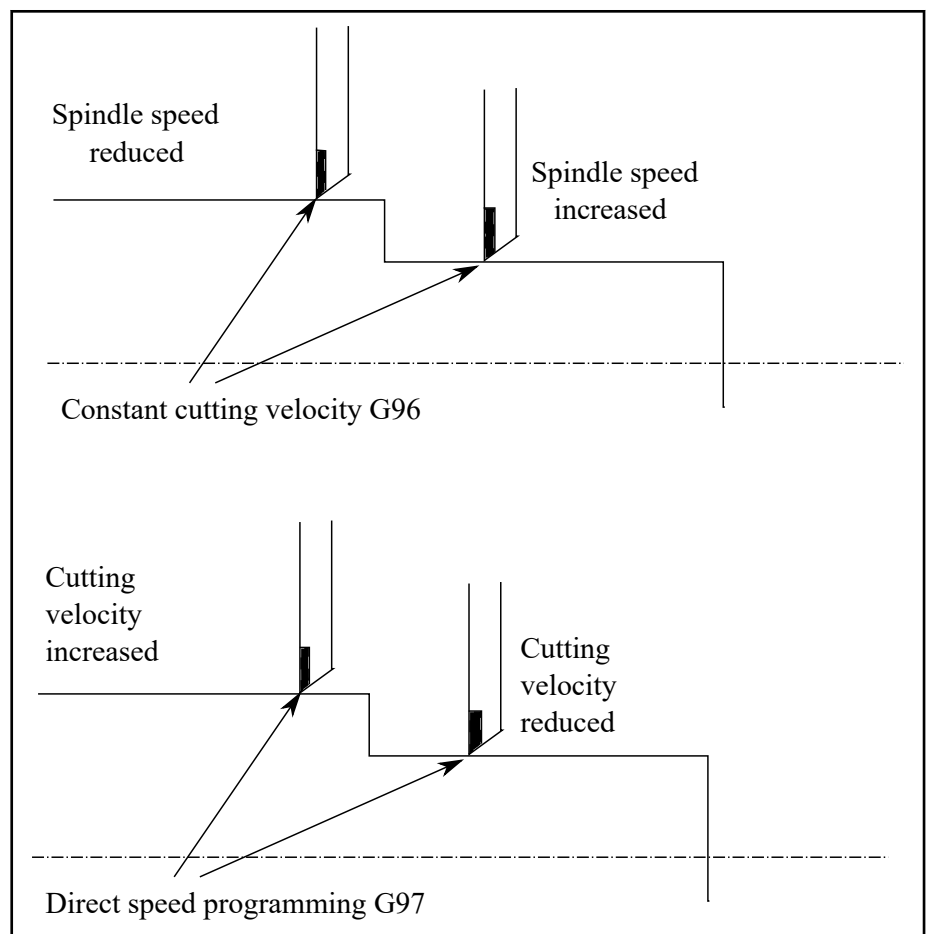


Fig. 14-2: CS parameter

ZG zero offset group and ZV zero offset bank:

Set active zero point data for machining

X-position and z-position:

Set the starting position for the x- and z-axis

POS strategy to approach the starting position:

Set the strategy to approach the starting position before machining the work-piece

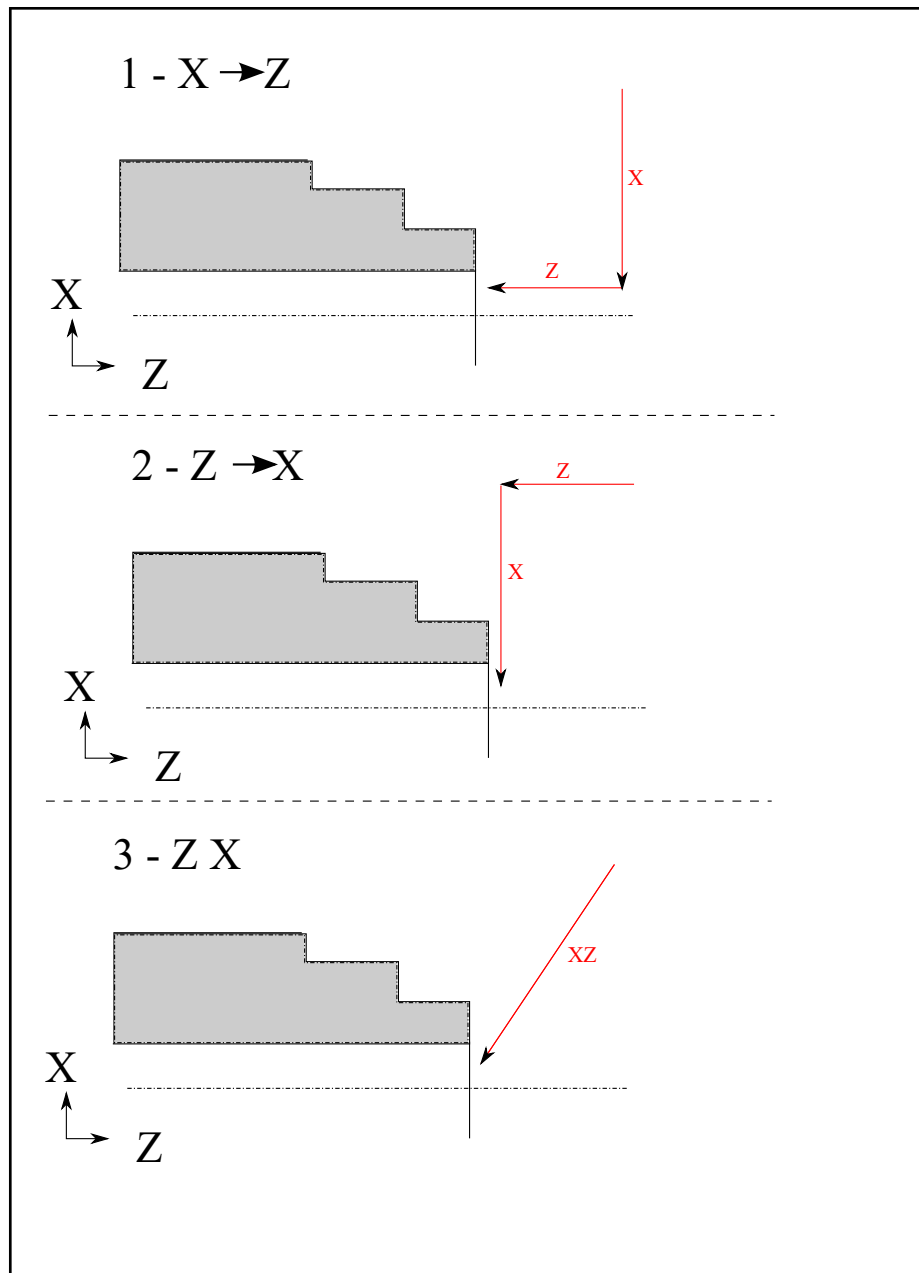


Fig. 14-3: Approach strategy POS

TCH strategy for the tool change position

Set the strategy to approach the tool change position. With 0, there is no tool change.

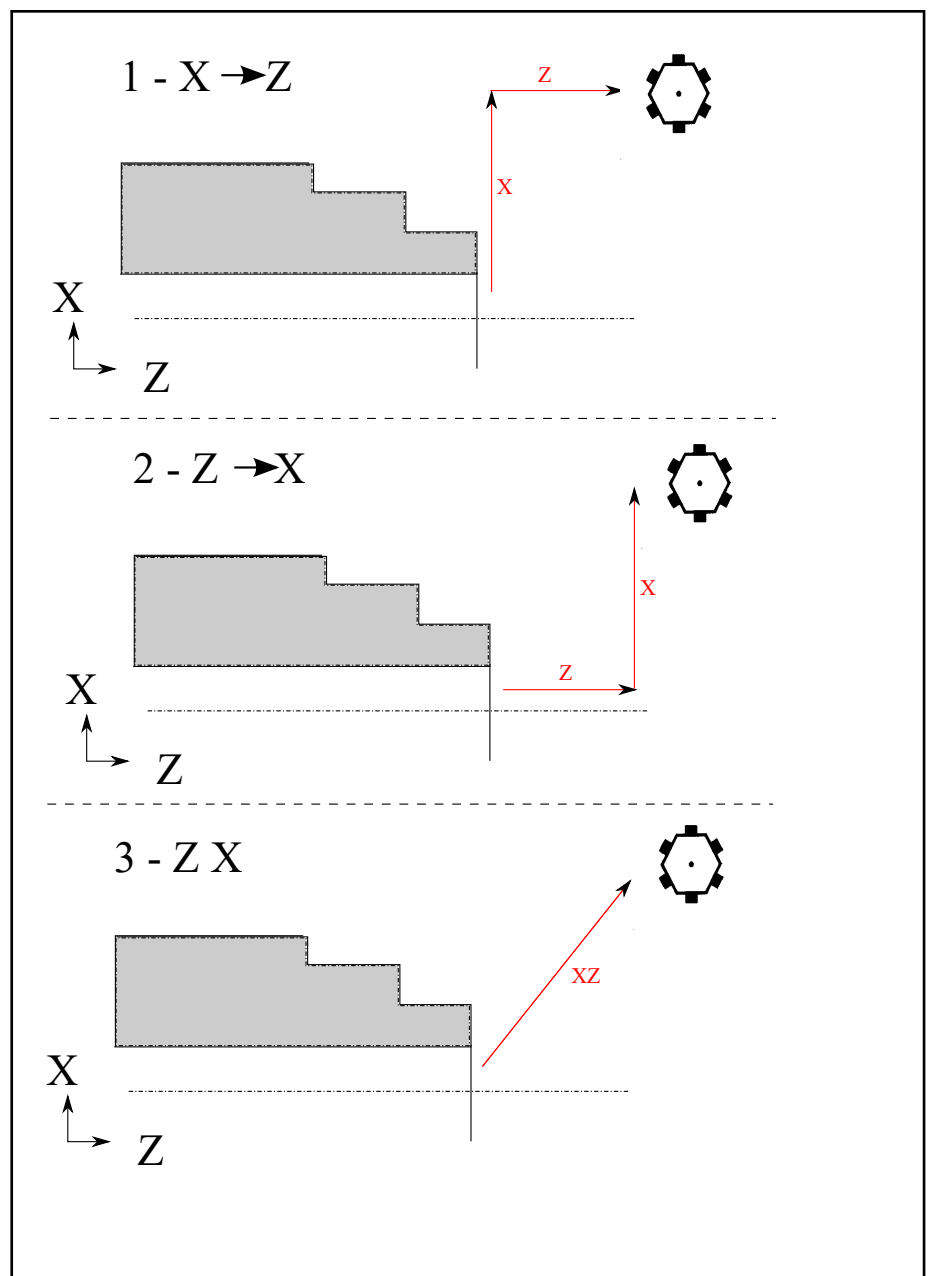


Fig. 14-4: TCH strategy for the tool change position

Special features

To approach the tool change position, write to the following system data outside the cycle:

- SD.SysG5xx.X_TCH_POS : Tool change position for the x-axis
- SD.SysG5xx.Z_TCH_POS : Tool change position for the z-axis

Procedure/processing

1. Approaches the tool change position related to the TCH value (strategy to approach the tool change position). Pre-assignment: X and z are traversed at the same time
2. Change the tool using the preselected T (tool number)
3. Start the spindle with the specified values S (spindle speed) and DIR (direction of rotation of the spindle)

Set the machining conditions; pre-assignment: Constant cutting velocity, feed, feed per revolution

4. Approach the starting point with the strategy specified in POS (strategy to approach the starting position). Pre-assignment: X and z are traversed at the same time. If no values are defined for x and z, no starting point is approached

Example *Program for G501 TFS Cycle turning*

```

N1000 G18 DIA          ; Initialization
; Call cycle
N1010 G501(T1,F100, S500,DIR M3,ZG G55, ZB2, X60,
==> Z10, TCH3)
; Start machining
N1020 G1 X40 Z0
N1020 G1 Z-20
N1030 G1 X60
N1040 G0 Z10           ; Return to starting position
N1050 M5               ; Spindle stop
N1060 M30              ; Program end

```

14.3 G511 TFS Cycle milling

Brief description Support cycle to initialize NC programs in the "milling" technology. Tool, feed and spindle data are set.

Overview figure

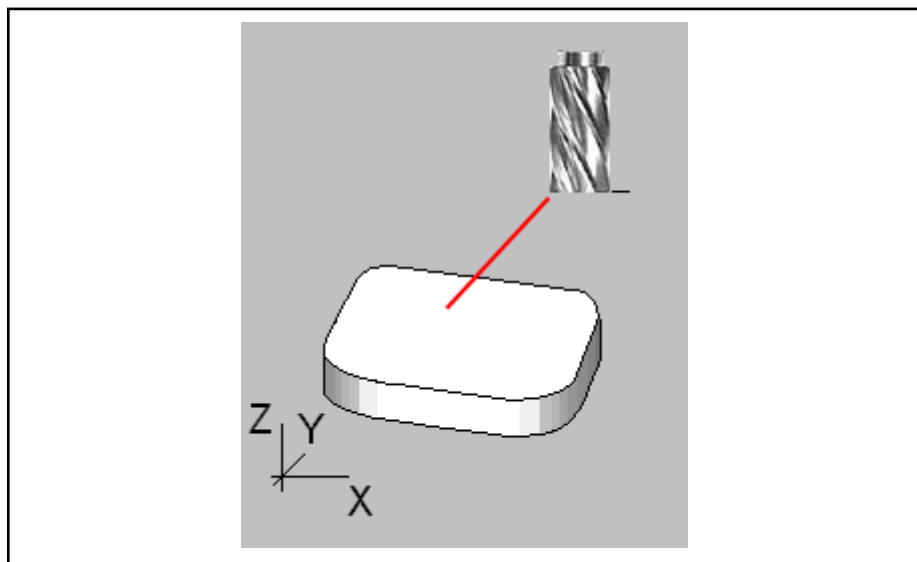


Fig. 14-5: Cycle G511 TFS Milling

Parameter

Parameter	Name	Properties	Values	Pre-assignment
T	Tool number	Mandatory		
ED	Tool edge number	Optional		
F	Feed	Mandatory		
S	Spindle speed	Mandatory		
DIR	Direction of spindle rotation	Mandatory	M3, M4	
SMX	Maximum spindle velocity	Optional		
ZG	ZO group	Mandatory	G54..G59	G54
ZB	ZO bank	Optional	1..5	1

Parameter	Name	Properties	Values	Pre-assignment
X	X-position	Optional		
Y	Y-position	Optional		
Z	Z-position	Optional		
POS	Strategy to approach the starting position	Optional	1 - X, Y then Z 2 - Z then X, Y 3 - X, Y, Z	3
TCH	Strategy to approach the tool change position	Optional	0 - None 1 - X, Y, Z 2 - Z then X, Y 3 - Only Z	3

Tab. 14-2: Parameter call

Description**T-tool number and ED edge number:**

Select tool and set active edge

F feed:

Set feed for machining

S spindle velocity, DIR direction of spindle rotation and SMX maximum spindle velocity

Set the spindle velocity and the direction of rotation of the spindle for machining. The upper limit of the permitted speed range is specified with SMX.

ZG zero offset group and ZV zero offset bank:

Set active zero point data for machining

X-position, Y-position and Z-position:

Set starting position for the x-, y- and z-axis

POS strategy to approach the starting position:

Set the strategy to approach the starting position before machining the work-piece

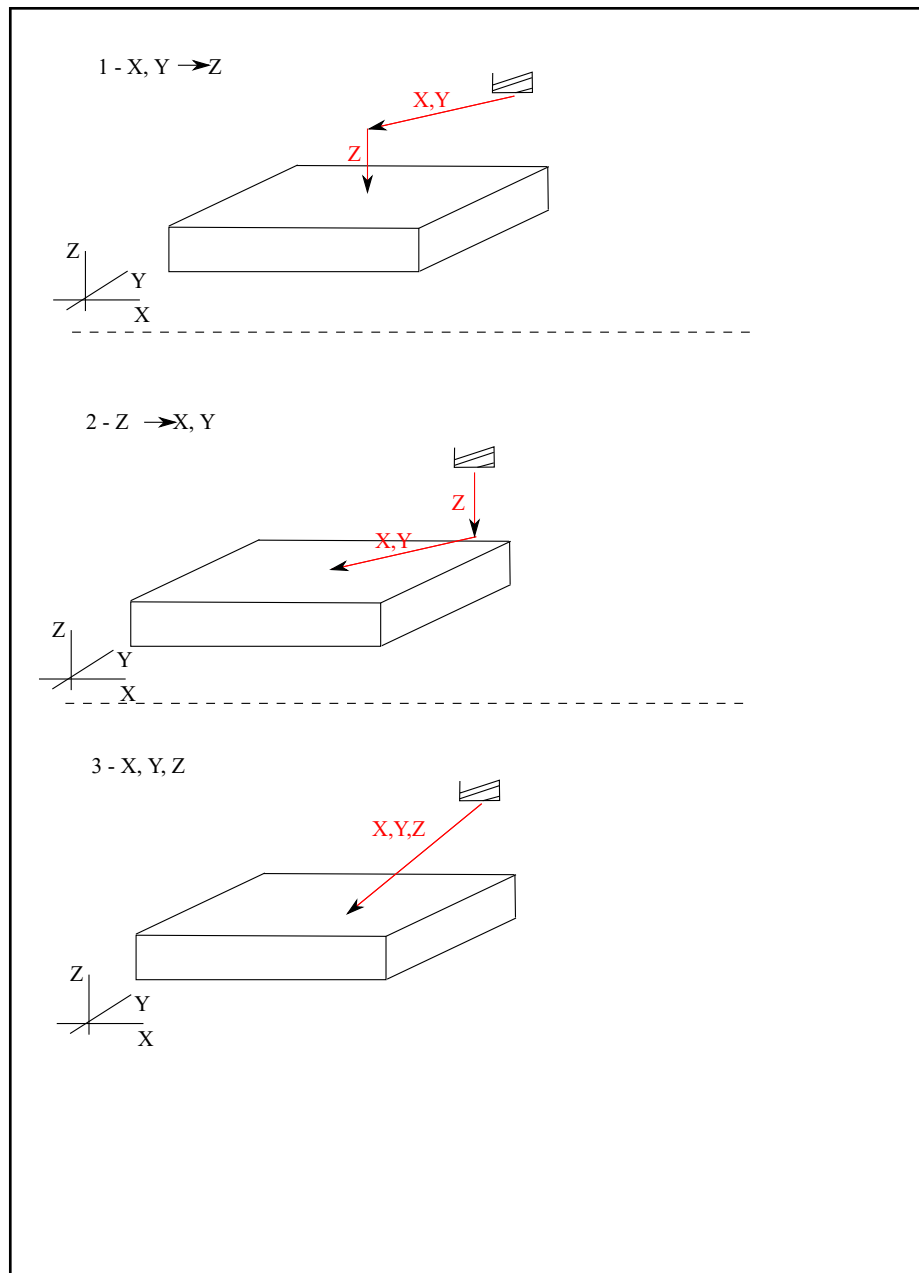


Fig. 14-6: Approach strategy POS

TCH strategy for the tool change position

Set the strategy to approach the tool change position. With 0, there is no tool change.

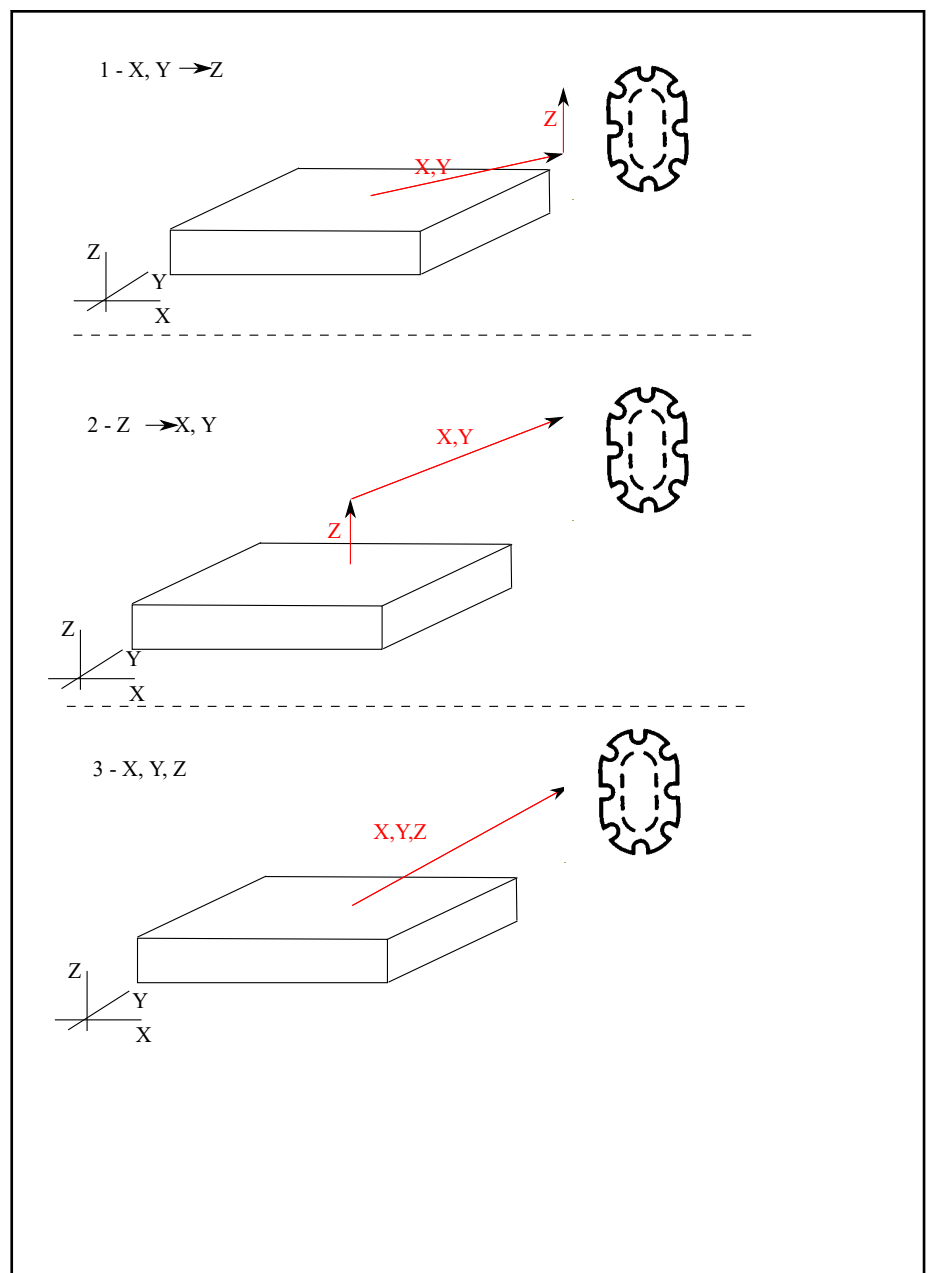


Fig. 14-7: Approach strategy: tool change

Special features TCH strategy for the tool change position

To approach the tool change position, write to the following system data outside the cycle:

- SD.SysG5xx.X_TCH_POS : Tool change position for the x-axis
- SD.SysG5xx.Y_TCH_POS : Tool change position for the y-axis
- SD.SysG5xx.Z_TCH_POS : Tool change position for the z-axis

- Procedure/processing**
1. Approaches the tool change position related to the TCH value (strategy to approach the tool change position). Pre-assignment: Only z is traversed
 2. Change the tool using the preselected T (tool number)

3. Start the spindle with the specified values S (spindle speed) and DIR (direction of rotation of the spindle)
Set the machining conditions; pre-assignment: Constant cutting velocity, feed, feed per revolution
4. Approach the starting point with the strategy specified in POS (strategy to approach the starting position). Pre-assignment: Pre-assignment: X, y and z are traversed at the same time. If no values are defined for x, y and z, no starting point is approached

Example *Program for G511 TFS Cycle milling*

```

N1000 G18 DIA                ; Initialization
; Call cycle
N1010 G511(T1,F100, S1500,DIR M3,ZG G56, ZB2, X60,
==> Y60, Z10, TCH3)
; Start machining
N1020 G1 Z-5
N1020 G1 X0
N1030 G1 Y0
N1040 G1 X60
N1030 G1 Y60
N1040 G0 Z10                ; Return to starting position
N1050 M5                    ; Spindle stop
N1060 M30                  ; Program end

```

14.4 G591 HSC settings



The cycle G591 is not supported anymore in MTX 14VRS and above. Instead of cycle G591, the cycle "G592 Approximated free form surface milling" can be used for free form surface milling.

Syntax G591(MOD)**Brief description** The coordination of the following factors are decisive for the free form surface milling:

- Machining velocity
- Surface quality
- Accuracy of parts

The cycle "G591 HSC settings" facilitates programming of the optimized NC settings for the high speed machining (HSM) of free form surfaces. The cycle selected the spline type, compatible with the selected machining mode and sets the optimized acceleration and jerk values.

Parameter

Parameter setting	Description
MOD0	HSC off
MOD1	Machining quality "very rough"
MOD2	Machining quality "rough"
MOD3	Machining quality "fine"/ (ACS) off
MOD31	Machining quality "fine"/ (ACS) low
MOD32	Machining quality "fine"/ (ACS) average
MOD33	Machining quality "fine"/ (ACS) high
MOD4	Machining quality "very fine"/ (ACS) off
MOD41	Machining quality "very fine"/ (ACS) low

MOD42	Machining quality "very fine"/ (ACS) average
MOD43	Machining quality "very fine"/ (ACS) high

Tab. 14-3: Parameter call

Mode machining "MOD"

The cycle machining mode has to be set according to the program requirements.

Four machining types are available:

- MOD1 "Pre-roughing"
- MOD2 "Roughing"
- MOD3 "Finishing"
- MOD4 "Extra-fine finishing"

Basic rule for the MOD parameter:

The higher the **first position** of the MOD, the better the accuracy and the condition of the surface of the workpiece. A higher precision entails a lower machining velocity.

By means of the **second position** of the MOD parameter, there is the option to activate a smoothing function **ACS (Active Contour Smoothing)**. The smoothing effect mainly depends on the distance between the programmed points.



The smoothing function can be used when machining concave workpiece surfaces to eliminate sample structures generated by the CAD/CAM process chain.

ACS is exclusively intended to improve the visual appearance of workpiece surfaces and should **not be activated permanently**.

- MOD1** The "mode pre-roughing" MOD1 is used for very fast machining where the accuracy of parts is of less relevance. A typical MOD1 use case is roughing as preparation of the workpiece for a finishing cut.
- MOD2** The "mode roughing" MOD2 (similar to MOD1) is used for very fast machining where the accuracy of parts is of less relevance. A typical MOD2 use case is roughing as preparation of the workpiece for a finishing cut. MOD2 should be used if the accuracy of MOD1 is not satisfactory once the roughing process has been completed.
- MOD3** In the "roughing mode" MOD3, accuracy and cut velocity are perfectly synchronized. A typical MOD3 use case is the finishing cut of a tool part or the molded part.
- MOD31** In addition to the properties of the machining type "finishing", a smoothing function is activated for NC program sections for MOD31 in which the distances to adjacent points (NC function block lengths) are smaller than **1 mm**.
- MOD32** In addition to the properties of the machining type "finishing", a smoothing function is activated for NC program sections for MOD32 in which the distances to adjacent points (NC function block lengths) are smaller than **3 mm**.
- MOD33** In addition to the properties of the machining type "finishing", a smoothing function is activated for NC program sections for MOD32 in which the distances to adjacent points (NC function block lengths) are smaller than **10 mm**.

- MOD4** The "mode extra-fine finishing" MOD4 is used if the expected accuracy or the quality of the surface condition is not satisfactory after having used MOD3. A typical MOD4 use case is the finishing cut of a tool part or the molded part.
- MOD41** In addition to the properties of the machining type "extra-fine finishing", a smoothing function is activated for NC program sections for MOD41 in which the distances to adjacent points (NC function block lengths) are smaller than **0.5 mm**.
- MOD42** In addition to the properties of the machining type "extra-fine finishing", a smoothing function is activated for NC program sections for MOD42 in which the distances to adjacent points (NC function block lengths) are smaller than **1.5 mm**.
- MOD43** In addition to the properties of the machining type "extra-fine finishing", a smoothing function is activated for NC program sections for MOD43 in which the distances to adjacent points (NC function block lengths) are smaller than **5 mm**.
- MOD0** When using MOD0, the machining mode is reset. All parameters that have already been adjusted are reset to the default value.

The following example shows the use of MOD for a roughing or finishing cut of a part:

Program:

```
; G591 sample for MOD
...
N10 T1      ;select rough tool
N15 G0 G90 G54 X50 Y50 Z100 S12000 M3      ;go to start
N20 G591(MOD2)      ;define rough machining mode
N25 P rough_part.npg      ;call roughing subroutine
N30 G591(MOD0)      ;reset machining mode
N35 T2 ;select finish tool
N40 G0 G90 G54 X50 Y50 Z50 S20000 M3      ;go to start
N45 G591(MOD3)      ;define finish machining mode
N50 P finish_part.npg      ;call finish subroutine
N55 G591(MOD0)      ;reset machining mode
...
```



It is recommended to program G591(MOD0) at the start of the tool change subroutine.

- Cycle mask** The cycle mask supports the parameterization and contains information about the current setting of the G591 free form surface cycle.

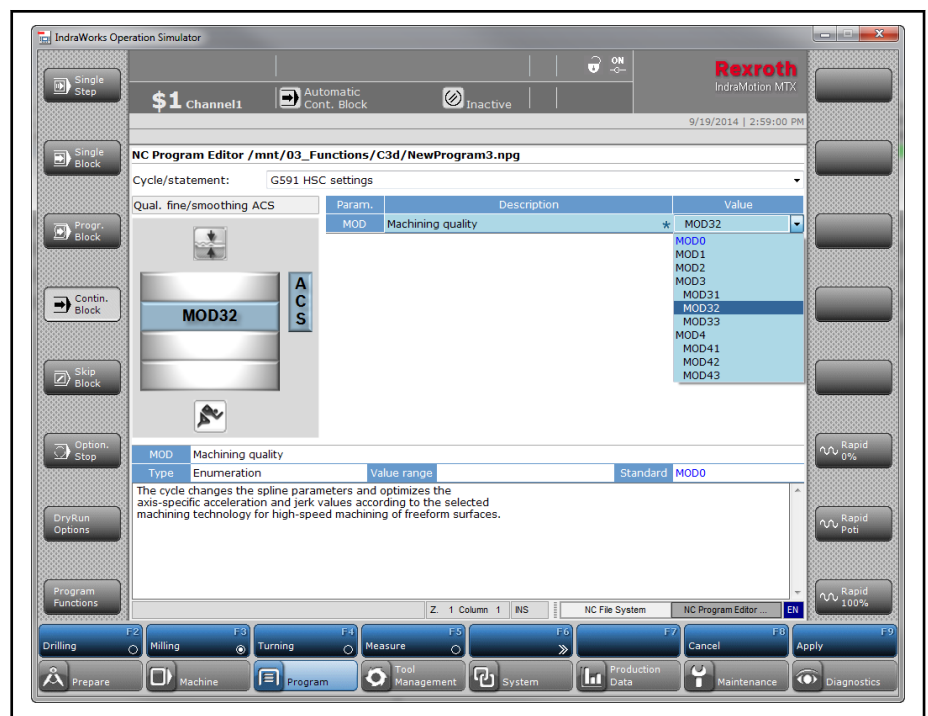


Fig. 14-8: G591 cycle mask

Use the drop-down menu to select the relevant **MOD** or the **machining type**. Based on this selection, the cycle description and the cycle view is adjusted in the left area.

14.5 G592 Approximated free form surface milling

Syntax G592(TOL)

Brief description The cycle "G592 Approximated free form surface milling" defines the spline parameters and sets the tolerance for free form surface milling.

Overview figure

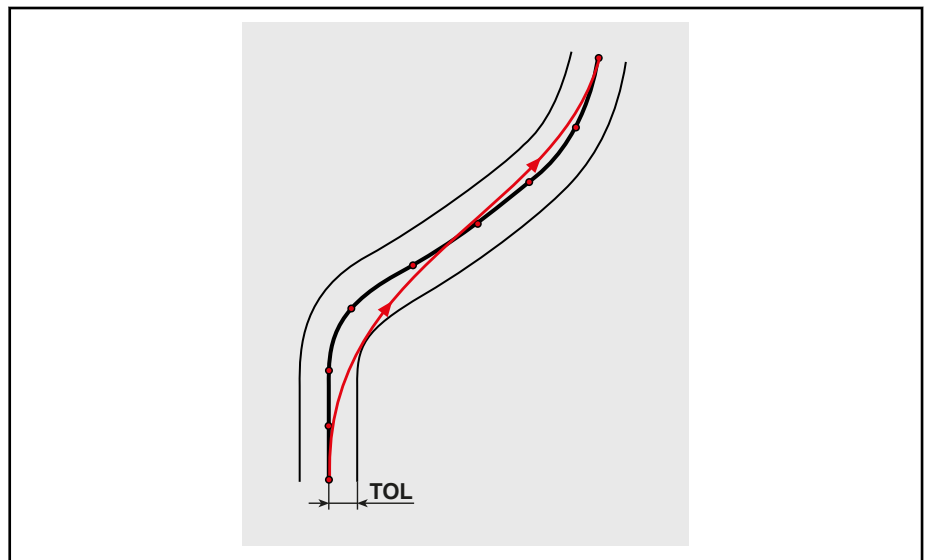


Fig. 14-9: Cycle G592 Approximated free form surface milling

Technology cycles

Parameter

Parameter	Name	Properties	Values	Pre-assignment
TOL	Tolerance band	Mandatory		0
LCTS	LCTS use	Optional		0

Tab. 14-4: Parameter call

Parameter setting	Description
TOL0	Approximated free form surface milling off
TOL>0	Specification tolerance band machining on
LCTS=0	Linear interpolation (G1) is replaced by spline interpolation (G6)
LCTS=1	Linear interpolation and circle interpolation (G1, G2 and G3) are replaced by spline interpolation (G6)

Tab. 14-5: Parameter value assignment

Description

TOL tolerance:

Setting the tolerance margin

Within the defined tolerance margin, a new, approximated path curve is calculated that results in an improved program processing.

LCTS use:

Setting to select whether LCTS or LTS is used.

When using LCTS, the circular interpolation is replaced by the spline interpolation in addition to the linear interpolation.

Special features

The cycle G592 does not change any kinematic settings.

All kinematic settings such as jerk and acceleration adjustments have to be programmed after calling the cycle. Otherwise, the settings in the machine parameters apply or the settings that have been set at a different position.

Sequence/processing

The path defined by the NC program is recalculated, depending on parameter "TOL". The recalculated path is within the tolerance margin and has an improved kinematic behavior.

Example

Program for G592 Approximated free form surface milling

```

...
N10 T1 ;select rough tool
N15 G0 G90 G54 X50 Y50 Z100 S12000 M3 ;go to start
N20 G592(TOL0.1) ;define rough machining mode
;N21 JKC(KLL3,TIME25) ;Kinematics limitation or preset in machine parameters
N25 P rough part.npg ;call roughing subroutine
N30 G592(TOL0) ;reset machining mode
N35 T2 ;select finish tool
N40 G0 G90 G54 X50 Y50 Z50 S20000 M3 ;go to start
N45 G592(TOL0.01) ;define finish machining mode
;N21 JKC(KLL3,TIME25) ;Kinematics limitation or preset in machine parameters
N50 P finish part.npg ;call finish subroutine
N55 G592(TOL0) ;reset machining mode
...

```

For a detailed description, refer to the documentation "IndraMotion MTX Free Form Surface Milling" (DOK-MTX***-FREEFORM***-APRS-EN-P).

14.6 STP static tool placement

Description: The STP cycles is intended for tools consisting of an adapter on which the tool and its holder are positioned and can be swiveled. The tool correction vector L results from the sum of vector $L^{(W)}$ from the tool tip to the pivot point and the vector $L^{(A)}$ from the pivot point to the tool reference point at the machine: $L = L^{(W)} + L^{(A)}$

If the tool correction vector L for a reference orientation is known and saved under a certain edge number of the active tool, cycle STP can determine the tool correction vector L for every other orientation and to store the vector L under a different or under the same edge number.

Using G47(...) or G78, tool length corrections L_1 , L_2 and L_3 are assigned to the L_1 , L_2 and L_3 coordinate directions.

These coordinate directions form a coordinate system. The tool length correction vector $L = L^{(W)} + L^{(A)}$ has the following components in this coordinate system:

$$L_1 = L^{(W)}_1 + L^{(A)}_1$$

$$L_2 = L^{(W)}_2 + L^{(A)}_2$$

$$L_3 = L^{(W)}_3 + L^{(A)}_3$$

Using the cycle STP, $L^{(W)}$ is rotated around the Euler angles Phi (ϕ), Theta (θ) and Psi (ψ) in this coordinate system while $L^{(A)}$ is retained. L_1 , L_2 and L_3 are recalculated and is saved under the selected edge number.

The vector $L^{(W)}$ is rotated as follows:

1. first about the L_3 coordinate direction by the angle **Phi**,
2. then around the angle **Theta** around the L_2 coordinate direction **rotated during the previous rotation**
3. then around the angle **Psi** around the coordinate direction **rotated during the previous rotation** L_3 .

The following rotations of L have the same result:

1. first about the L_3 coordinate direction by the angle **Psi**,
2. then around the angle **Theta** around the **original, not rotated** L_2 coordinate direction
3. then around the angle **Phi** around the **original, not rotated** L_3 coordinate direction.

As both approaches can lead to the same result, use the approach that appeals more to you.

Syntax:

STP (<EDstart> , <L ^(A) ₁ > , <L ^(A) ₂ > , <L ^(A) ₃ > , <φ> , <θ> , <ψ> , <EDres>)	
with	
<EDstart>	Edge number of the edge of the active tool to be rotated
<L ^(A) ₁ > , <L ^(A) ₂ > , <L ^(A) ₃ >	Vector from the pivot point to the tool reference point at the machine

<φ>,<θ>,<ψ>	Euler angle in degree Value range: $0^\circ \leq \varphi < 360^\circ$ $0^\circ \leq \theta \leq 180^\circ$ $0^\circ \leq \psi \leq 360^\circ$ Values outside this range are automatically converted to the respective interval
<EDres>	Edge number under which the tool correction vector L for the rotated edge is to be saved

Tab. 14-6: Syntax STP

Rotation of $L^{(W)}$ around the angle Alpha around the L_1 coordinate direction:

L_1 coordinate direction:

STP (<EDstart> , <L^(A)₁> , <L^(A)₂> , <L^(A)₃> , -90 , <Alpha> , 90 ,
<EDres>)

Example: Program:

```
1 DCT(1,1,0) = 150
1 DCT(2,1,0) = 0
1 DCT(3,1,0) = 0 : REM ( DCT(1,2,0), DCT(2,2,0), DCT(3,2,0) )
STP(1, 0, 0, 0, 0, 0, 0, 2) ; ( 150 , 0 , 0 )
STP(1, 0, 0, 0, 90, 0, 0, 2) ; ( 0 , 150 , 0 )
STP(1, 0, 0, 0, 0, 90, 0, 2) ; ( 0 , 0 , 150 )
STP(1, 50, 0, 0, 0, 0, 0, 2) ; ( 150 , 0 , 0 )
STP(1, 50, 0, 0, 90, 0, 0, 2) ; ( 50 , 100 , 0 )
STP(1, 50, 0, 0, 0, 90, 0, 2) ; ( 50 , 0 , 100 )
M30
```



Also see "Parameterizing the static tool orientation StatToolOri STO" in the documentation "IndraMotion MTX Programming Manual" (DOK-MTX***-NC**PRO*V14-RERS-EN-P). In this chapter, there are more examples for a rotation around the Euler angle.

14.7 SETBQ Set/reset tool/position status bit

Syntax SETBQ(<SEC>,<LOC>,<BQ>,<BIT>,<VAL>)

Brief description The cycle SETBQ facilitates setting/resetting of a position or tool status bit in the tool data block.

Parameter

Parameter	Name	Properties	Values	Pre-assignment
SEC	Sector	Mandatory	Absolute	1
LOC	Place	Mandatory	Absolute	1
BQ	Data element	Mandatory	1	Place status
			2	Tool status
BIT	Identifier BQ1/2	Mandatory	>0,<=32	Selection via drop-down box See table

SET	Action	Mandatory	1 or TRUE	Set
			0 or FALSE	Reset

Tab. 14-7: Parameter call

Description SEC sector**LOC place**

Write these two parameters (via database key: sector number, position number) of the tool data block in which the status bit is to be changed.

BQ data element

This parameter defines the data element (BQ1 - for position status and BQ2- for tool status) in which the status bit is to be changed.

BIT identifier BQx (x = 1.2)

To define this parameter, the short description of the status bit to be changed can be selected.

The bit no. is transmitted as result of the selection

Status group	Designation	Brief descr.	Bit no.	Description
Place status	Free	pf	1	Free place
	Blocked	PB	2	Blocked place
	Ignore	pi	3	Place ignored
	Reserved	pr	4	Place reserved
	ReservedLeft	pri	5	Left half-place reserved
	ReservedRight	prr	6	Right half-place reserved
	ReservedLeftLower	pril	7	Left lower half-place reserved
	ReservedRightLower	prri	8	Right lower half-place reserved
	OccupiedLeft	pol	9	Left half-place assigned
	OccupiedRight	por	10	Right half-place assigned
	OccupiedLeftLower	poll	11	Left lower half-place assigned
	OccupiedRightLower	porl	12	Right lower half-place assigned
	PlaceTypeMonitoring	ptm	13	Place type machining
	DeviceTypeMonitoring	dtm	14	Device type machining
	PlaceTypeMonitoring	ftm	15	Form type machining
	DeviceInPlace	dip	16	Device in place
	PlaceUserStatus 1	PS1	25	Location user status 1
	PlaceUserStatus 2	PS2	26	Place user status 2
	PlaceUserStatus 3	PS3	27	Place user status 3
	PlaceUserStatus 4	PS4	28	Place user status 4
	PlaceUserStatus 5	PS5	29	Place user status 5
	PlaceUserStatus 6	PS6	30	Place user status 6

Technology cycles

Status group	Designation	Brief descr.	Bit no.	Description
	PlaceUserStatus 7	PS7	31	Place user status 7
	PlaceUserStatus 8	PS8	32	Place user status 8

Tab. 14-8: BQ1 - Place status

Status group	Designation	Brief descr.	Bit no.	Description
Tool status	ActiveTool	ta	1	Tool active
	PrimaryTool	tp	2	Primary tool
	UsedTool	tu	3	Used tool
	WarnLimit	tw	4	Warning limit reached
	WornOut	two	5	Tool worn
	PlaceCoded	TPC	6	Fixed place-coded tool
	Locked	TL	7	Tool locked
	Defective	TD	8	Tool defective
	Ignore	ti	9	Ignore tool
	Measured	tm	10	Probe tool
	ToLoad	tli	11	Tool loaded
	ToUnload	ttu	12	Tool unloaded
	MasterTool	mt	13	Master tool
	ToolChange	tc	14	Tool change
	SisterTool	ts	15	Replacement tool
	HeavyTool	TH	16	Heavy tool
	NonEditable	tne	17	Tool cannot be edited
	UserStatus 1	TS1	25	Tool user status 1
	UserStatus 2	TS2	26	Tool user status 2
	UserStatus 3	TS3	27	Tool user status 3
	UserStatus 4	TS4	28	Tool user status 4
	UserStatus 5	TS5	29	Tool user status 5
	UserStatus 6	TS6	30	Tool user status 6
	UserStatus 7	TS7	31	Tool user status 7
	UserStatus 8	TS8	32	Tool user status 8

Tab. 14-9: BQ2 - Tool status

SET action

Setting or resetting can be selected.

Examples: Program to set the SETBQ tool/place status bit

```
N10 SETBQ(SEC1, LOC1, BQ2, BIT 26,SET TRUE) ;the tool status bit 26 (TS2) of the tool on
                                           ;sector 1 / position 1 is set
```

N10 SETBQ(SEC2, LOC5, BQ1, BIT 1,SET 1) ;the position status bit 1 (pf) of the tool position on
;sector 2 / position 5 is set

Program to reset the SETBQ tool/place status bit

N20 SETBQ(SEC2, LOC1, BQ1, BIT 2,SET 0) ;the position status bit 2 (PB) of the tool position on
;sector 2 / position 1 is reset
N30 SETBQ(SEC1, LOC2, BQ2, BIT 4,SET FALSE) ;the tool status bit 4 (tw) of the tool on
;sector 1 / position 2 is reset

14.8 GETBQ Read tool/position status bit

Syntax GETBQ(<SEC>,<LOC>,<BQ>,<BIT>)
Brief description The cycle GETBQ facilitates reading of a position or tool status bit in the tool data block.

Parameter	Parameter	Name	Properties	Values	Pre-assignment
	SEC	Sector	Mandatory	Absolute	1
	LOC	Place	Mandatory	Absolute	1
	BQ	Data element	Mandatory	1	Place status
				2	Tool status
	BIT	Identifier BQ1/2	Mandatory	>0,<=32	Selection via drop-down box See table

Tab. 14-10: Parameter call

Description **SEC sector**

LOC place

Write these two parameters (via database key: sector number, position number) of the tool data block in which the status bit is to be changed.

BQ data element

This parameter defines the data element (BQ1 - for position status and BQ2- for tool status) in which the status bit is to be changed.

BIT identifier BQx (x = 1.2)

To define this parameter, the short description of the status bit to be changed can be selected.

The bit no. is transmitted as result of the selection

Status group	Designation	Brief descr.	Bit no.	Description
Place status	Free	pf	1	Free place
	Blocked	PB	2	Blocked place
	Ignore	pi	3	Place ignored
	Reserved	pr	4	Place reserved
	ReservedLeft	prl	5	Left half-place reserved
	ReservedRight	prr	6	Right half-place reserved
	ReservedLeftLower	prll	7	Left lower half-place reserved
	ReservedRightLower	prrl	8	Right lower half-place reserved

Technology cycles

Status group	Designation	Brief descr.	Bit no.	Description
	OccupiedLeft	pol	9	Left half-place assigned
	OccupiedRight	por	10	Right half-place assigned
	OccupiedLeftLower	poll	11	Left lower half-place assigned
	OccupiedRightLower	porl	12	Right lower half-place assigned
	PlaceTypeMonitoring	ptm	13	Place type machining
	DeviceTypeMonitoring	dtm	14	Device type machining
	PlaceTypeMonitoring	ftm	15	Form type machining
	DeviceInPlace	dip	16	Device in place
	PlaceUserStatus 1	PS1	25	Location user status 1
	PlaceUserStatus 2	PS2	26	Place user status 2
	PlaceUserStatus 3	PS3	27	Place user status 3
	PlaceUserStatus 4	PS4	28	Place user status 4
	PlaceUserStatus 5	PS5	29	Place user status 5
	PlaceUserStatus 6	PS6	30	Place user status 6
	PlaceUserStatus 7	PS7	31	Place user status 7
	PlaceUserStatus 8	PS8	32	Place user status 8

Tab. 14-11: BQ1 - Place status

Status group	Designation	Brief descr.	Bit no.	Description
Tool status	ActiveTool	ta	1	Tool active
	PrimaryTool	tp	2	Primary tool
	UsedTool	tu	3	Used tool
	WarnLimit	tw	4	Warning limit reached
	WornOut	two	5	Tool worn
	PlaceCoded	TPC	6	Fixed place-coded tool
	Locked	TL	7	Tool locked
	Defective	TD	8	Tool defective
	Ignore	ti	9	Ignore tool
	Measured	tm	10	Probe tool
	ToLoad	ttl	11	Tool loaded
	ToUnload	ttu	12	Tool unloaded
	MasterTool	mt	13	Master tool
	ToolChange	tc	14	Tool change
	SisterTool	ts	15	Replacement tool
	HeavyTool	TH	16	Heavy tool

Status group	Designation	Brief descr.	Bit no.	Description
	NonEditable	tne	17	Tool cannot be edited
	UserStatus 1	TS1	25	Tool user status 1
	UserStatus 2	TS2	26	Tool user status 2
	UserStatus 3	TS3	27	Tool user status 3
	UserStatus 4	TS4	28	Tool user status 4
	UserStatus 5	TS5	29	Tool user status 5
	UserStatus 6	TS6	30	Tool user status 6
	UserStatus 7	TS7	31	Tool user status 7
	UserStatus 8	TS8	32	Tool user status 8

Tab. 14-12: BQ2 - Tool status

Return value

Return value	Name	Value range
#VALUE?	Boolean variable	TRUE;FALSE

Tab. 14-13: Return value

Example: Program for reading the GETBQ tool/position status bit

```
N10 GETBQ(SEC1, LOC1, BQ2, BIT 26) ;the tool status bit 26 (TS2) of the tool is read
;the value is contained in variable #VALUE?
```

14.9 INT_TO_STR Format conversion Integer to string

Description: The INT_TO_STR cycle is an auxiliary program to convert an integer value to a string.

Only 9 digits and the sign can be converted.

Value range integer: -2147483647 to 2147483647

Value range integer INT_TO_STR: -999999999 to 999999999

Parameter	Name	Value range
VALUE%	Integer value	-999999999 to 999999999

Tab. 14-14: Transfer parameters

Return value	Name	Value range
#INT_TEXT\$	String variable	DIM TEXT\$(11)

Tab. 14-15: Return value

Syntax:

INT_TO_STR (<value>)	
with	
<Value>	-999999999 to 999999999

Tab. 14-16: Syntax INT_TO_STR

Example: Program:

```
10 DIM #INT_TEXT$(11)
N20 INT_TO_STR(123456789)
20 PRN#(0,"results :",#INT_TEXT$); the string "123456789" is contained in the #INT_TEXT$ variable
```

14.10 REAL_TO_STR Format conversion Real to string

Description: The INT_TO_STR cycle is an auxiliary program to convert a Real value to a string in a compact floating comma representation.

Usually, all seven significant positions of the variable type are displayed in the result. The maximum number of decimal positions is limited to 6. Integer numbers are written without decimal points. No trailing zeros after the decimal point.

Parameter	Name	Value range
VALUE	Real value	-10^{38} to 10^{38}

Tab. 14-17: Transfer parameters

Return value	Name	Value range
#REAL_TEXT\$	String variable	DIM TEXT\$(42)

Tab. 14-18: Return Value

Syntax:

REAL_TO_STR(<value>)	
with	
<Value>	-10^{38} to 10^{38}

Tab. 14-19: Syntax REAL_TO_STR

Example: Program:

```
100 DIM #REAL_TEXT$(42)           ; contains the character string after
                                   ; the transformation
N10 REAL_TO_STR(1234.123)
110 PRN#(0,"result=",#REAL_TEXT$); results in: "result=1234.123"
N30 REAL_TO_STR(-1.234567)
140 PRN#(0,"result=",#REAL_TEXT$); results in: "result=-1.234567"
N20 REAL_TO_STR(0.01234567)
120 PRN#(0,"result=",#REAL_TEXT$); results in: "result=0.012346", max.
                                   ; 6 decimal places
N30 REAL_TO_STR(-123456789)
140 PRN#(0,"result=",#REAL_TEXT$); results in: "result=-123456700"
```

15 Punching cycles

15.1 General information

The cycles for punching and nibbling can only be used in case of a corresponding machine configuration.

Automatic setting of motion profile

It is possible or required to individually specify the path acceleration and the jerk limitation depending on the distance between the two punching strokes. For a user-friendly adjustment of the motion profile, the subroutine ACC_CFG was introduced. This subroutine is installed in a simple implementation in the /feprom/cycles directory. Overlay this subroutine during commissioning of a punching machine. If this is not done, a message is displayed.

The punching cycles directly triggering strokes, call the ACC_CFG subroutine before the first motion and transmit the smallest distance between two strokes (always in mm) in the only parameter without address of type double or real (P1 or P1!).

Example ACC_CFG

The following code shows a program sample, how individual motion profiles can be defined for different distance limits.

Sample implementation of the subroutine ACC_CFG

```

;*****
;* ACC_CFG[P1!]
;* P1! Distance between 2 punching strokes in mm *
;*****
100 IF P1!<=1.01 THEN
110 XACC=5 : YACC=0.4
120 XKVP=1.3 : YKVP=1.6
130 XSHAP%=4 : YSHAP%=4
140 ELSE IF P1!<=4.01 THEN
150 XACC=2 : YACC=0.8
160 XKVP=2 : YKVP=1.6
170 XSHAP%=60 : YSHAP%=40
181 ELSE IF P1!<=15.01 THEN
190 XACC=6 : YACC=1
200 XKVP=2 : YKVP=1
210 XSHAP%=80 : YSHAP%=200
220 ELSE IF P1!<=60.01 THEN
230 XACC=10 : YACC=1
240 XKVP=2 : YKVP=1
250 XSHAP%=80 : YSHAP%=200
260 ELSE
270 XACC=12 : YACC=1
280 XKVP=2 : YKVP=1.5
290 XSHAP%=80 : YSHAP%=200
300 ENDIF : ENDIF : ENDIF : ENDIF
N10 G9(X[XSHAP%],Y[YSHAP%])
N20 AxAcc(X[XACC],Y[YACC])
N30 KVP(X[XKVP],Y[YKVP])

```

Calling the punching cycles

The following conditions apply when using the punching and nibbling cycles:

- The cycles individually activate the punching function unless this has been done before the call. Before the cycle call, the punching modes for signing or reforming have to be active. After the cycle, the output punching mode is active
- A motion profile matching the punching process is activated in the cycle (except for G219 Multiple part punching)
If G8 is active before the cycle, G8 is also active after the cycle
The changed modal states of the control in the ACC_CFG subroutine are not restored at the end of the cycle
- The cycles do not change the technology parameter LEN
If a tangential tool guidance (TTL) is required, activate the guidance before the punching cycles start and deactivate them after the guidance is

completed, if required. The standard cycles internally use G1, G2 and G3

- The cycles are non-modal without exception and cannot be parameterized via SD data

15.2 G213 Hole circle punching

Syntax G213(MCP,SCP,R,NH,BA,DA)

Brief description The cycle "G213 Hole circle punching" punches a pattern of holes with alignment on an arc.

Overview

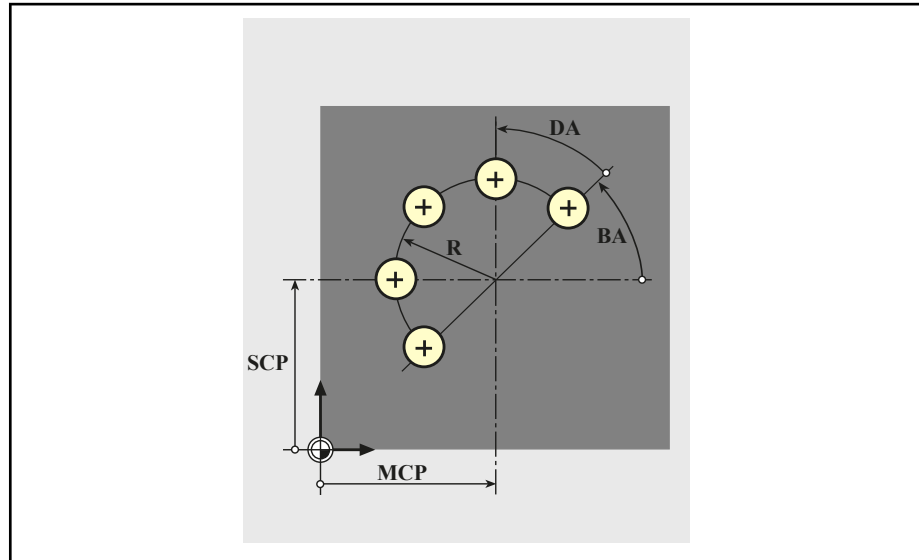


Fig. 15-1: Cycle G213 Hole circle punching

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
MCP	Center point: main axis	Mandatory	Absolute	
SCP	Center point: secondary axis	Mandatory	Absolute	
R	Radius	Mandatory	Incremental >0	
NH	Number of holes	Mandatory	>0	
BA	Angle of first hole	Optional	Absolute Degree	0
DA	Angle between two holes	Optional	Incremental <>0, Degree	360 / NH

Tab. 15-1: Parameter call

Description

MCP Center point: main axis

At this absolute position of the main axis, the center point of the pitch circle can be found

SCP Center point secondary axis

At this absolute position of the secondary axis, the center point of the pitch circle can be found

R Radius

Radius of the pitch circle

NH Number of holes

Total number of all holes on the circle

BA Angle of first hole

The first hole is punched at this first absolute angular pitch

DA Angle between two holes

Angle increment to the next hole

In case of negative DA, the circle is run in clockwise direction

Special features

The cycle executes a machining and independently activates the punching function and the acceleration profile.

Procedure/processing

The punching pattern in circle alignment is calculated in relation to the center point MCP/SCP. The first punching stroke takes place under the angle BA. In case of a positive incremental angle DA, the circle is run in counterclockwise direction, otherwise in clockwise direction. The cycle ends at the last pattern point.

Example

Machine two concentric punching punch patterns are machined with the center point X150 Y200:

- Full circle with 12 holes on a radius of 150 mm
- Punch 5 holes on an arc from angle 25° to the X-axis with a distance of 30° each

Program for G213 Hole circle punching

```
N10 G213 (MCP100, SCP200, R150, NH12) ; full circle radius 150
N20 G213 (MCP100, SCP200, R50, NH5, BA25, DA30) ; full circle radius 50
```

15.3 G214 Hole pattern line punching

Syntax

G214(PD,NH,A,MOD)

Brief description

The cycle "G214 Hole pattern line punching" generates a hole pattern along a line.

Overview

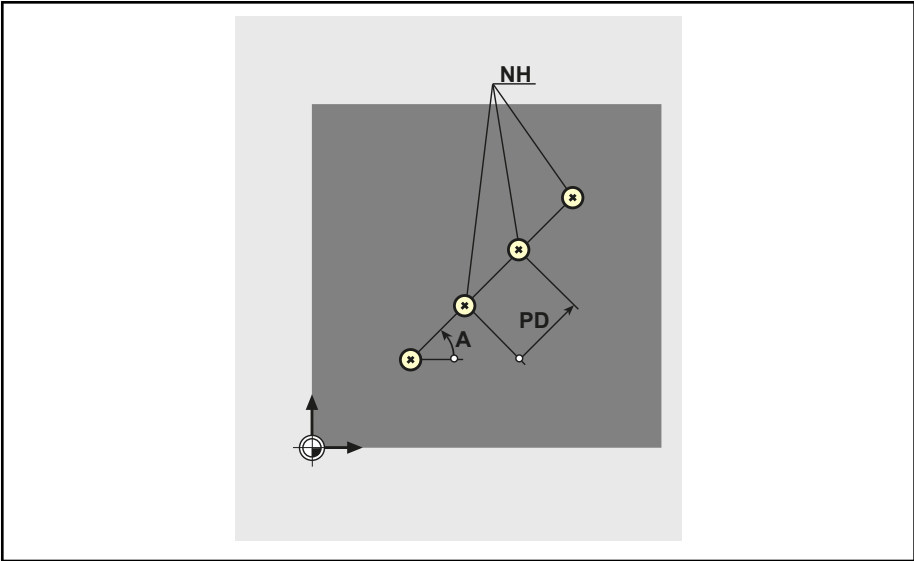
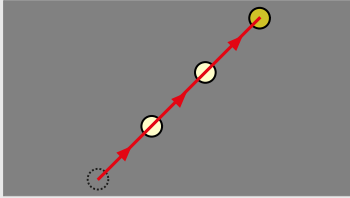
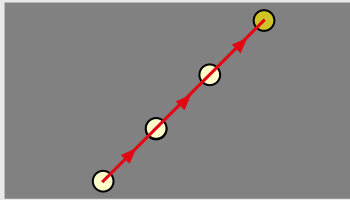


Fig. 15-2: Cycle G214 Hole pattern line punching

Parameter

Parameter	Designation	Properties	Values	Pre-assign-ment
PD	Hole distance	Mandatory	Incremental <>0	
NH	Number of holes	Mandatory	>0	
A	Angle	Optional	Absolute Degree	0
MOD	Selection of holes	Optional		3

Tab. 15-2: Parameter call

MOD	Effect
2	No punching at reference point
3	Punching at reference point
MOD2  MOD3 	

Tab. 15-3: Description of the parameter MOD - Selection of holes

Description**PD Hole distance**

Specifies the increment between two adjacent holes along the line

NH Number of holes

Number of holes without reference point/initial position

A Angle

Absolute angle to the main axis in whose direction all borings are aligned, starting from the reference point

MOD Selection of holes

Selects if another hole is punched at the reference point or initial point of the cycle

Special features

The cycle executes a machining and independently activates the punching function and the acceleration profile.

Procedure/processing

The punching pattern along a line is calculated in relation to the initial position. If MOD2 is programmed, the first stroke starts at the initial position. NH punching strokes along a line. The cycle ends at the last pattern point.

Example

Punch two punch patterns in line alignment:

- 20 holes x-parallel with a distance of 32 mm. Do not punch a hole at the starting point X200 Y100
- 21 holes under an angle of 45 ° to the x-axis with a hole distance of 32 mm. Additionally, execute a stroke at the starting point X150 Y200

Program for G214 Hole pattern line punching

```

N10 G0 X200 Y100 ; approach initial position
N20 G214 (PD32, NH20, MOD2) ; 1. Line X-parallel
N30 X150 Y200 G214 (PD32, NH20, A45) ; 2. Line 45°

```

15.4 G215 Hole pattern matrix punching

Syntax G215(CD,NC,LD,NL,DIR,MOD)

Brief description The cycle "G215 Hole pattern matrix punching" generates a hole pattern in the alignment of an axis-parallel matrix.

Overview

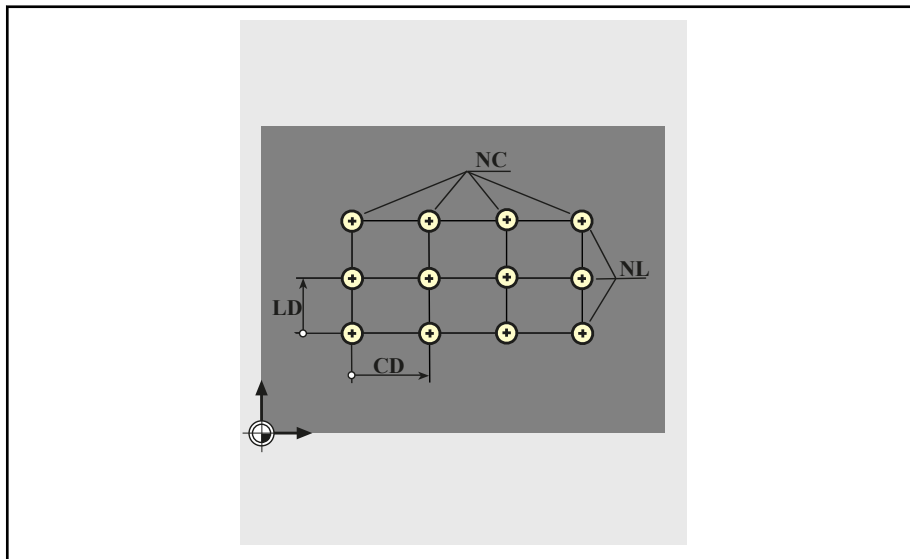


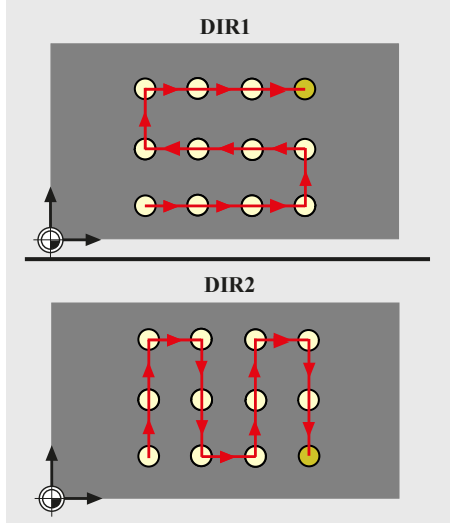
Fig. 15-3: Cycle G215 Hole pattern matrix punching

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
CD	Distance between columns	Mandatory	Incremental <>0	
NC	Number of columns	Mandatory	>0	
LD	Distance between lines	Mandatory	Incremental <>0	
NL	Number of lines	Mandatory	>0	
DIR	Direction of machining	Optional		1
MOD	Selection of holes	Optional		3

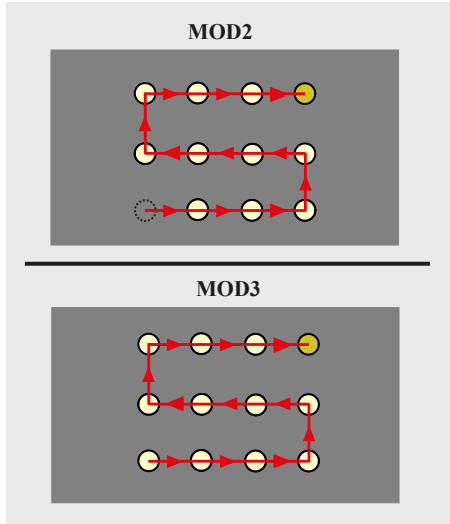
Tab. 15-4: Parameter call

DIR	Effect
1	Line-by-line processing
2	Column-by-column processing



Tab. 15-5: Description of the parameter DIR - Direction of machining

MOD	Effect
2	No punching at reference point
3	Punching at reference point



Tab. 15-6: Description of the parameter MOD - Selection of holes

Description**CD Distance between columns**

Specifies the increment of two subsequent columns (main axis)

NC Number of columns

Number of columns (direction of main axis)

LD Line distance

Specifies the increment of two subsequent lines (secondary axis)

	NL Number of lines
	Number of lines (direction of secondary axis)
	DIR Direction of machining
	Specifies whether the machining of holes follows the lines or the columns
	MOD Selection of holes
	Selects if a hole is punched at the reference point or the initial point of the cycle
Special features	The cycle executes a machining and independently activates the punching function and the acceleration profile.
Procedure/processing	The punching pattern is calculated in relation to the initial position- If MOD2 is programmed, the first stroke starts at the initial position. If DIR1 is programmed, all holes in the direction of lines (main axis) are punched before moving on to the next line. Each second line is run in reverse direction. If DIR2 is defined, all holes in the direction of the column (secondary axis) are punched before changing to the next column. Each second line is run in reverse direction. The cycle ends at the last pattern point.
Example	Punch two punch patters in form of a matrix with 3 lines and 5 columns: - Line-by-line to the reference point X100 Y200 without the initial point - Column-by-column at the reference point X300 Y100 including the initial point

Program for G215 Hole pattern matrix punching

```

N10 G0 X100 Y200 ; initial point 1. approach matrix
N20 G215 (CD30, NC3, LD25, NL5, MOD2) ; 1. matrix, without initial point
N30 X300 Y100 G215 (CD30, NC3, LD25, NL5, DIR2) ; 2. matrix column by column

```

15.5 G219 Multiple part punching

Syntax	G219(P, MOD,MSP,SSP,LE,WI,NC,NL,DIR)
Brief description	The cycle "G219 Multiple part punching" is a frame subroutine for effectively repeating the machining of identical workpieces, aligned as a matrix. The cycle contains no machining but calls a subroutine, containing the machining of a workpiece.

Overview

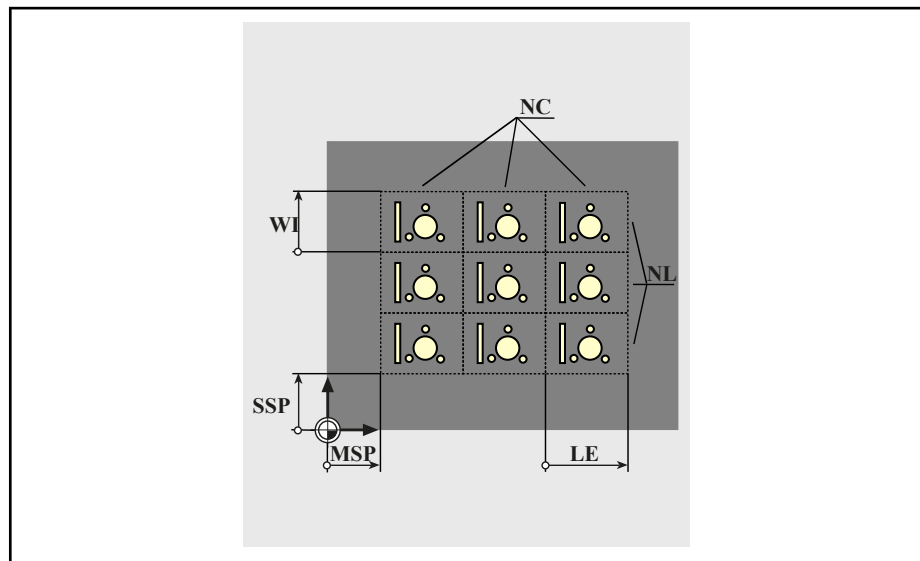


Fig. 15-4: Cycle G219 Multiple part punching

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
P	Name of work-piece subroutine	Mandatory	String, max. 30 characters	
MOD	Selection of workpieces	Optional		3
MSP	Reference point of main axis	Mandatory Not relevant for MOD0	Absolute	
SSP	Reference point of secondary axis	Mandatory Not relevant for MOD0	Absolute	
LE	Length of work-piece	Mandatory Not relevant for MOD0 and MOD1	Incremental <>0	
WI	Width of work-piece	Mandatory Not relevant for MOD0 and MOD1	Incremental <>0	
NC	Number of columns	Mandatory Not relevant for MOD0 and MOD1	>0	
NL	Number of lines	Mandatory Not relevant for MOD0 and MOD1	>0	
DIR	Direction of machining	Optional Not relevant for MOD0 and MOD1		1
TLC	Number of used tools	Optional		1

Tab. 15-7: Parameter call

DIR	Effect
1	Line-by-line processing
2	Column-by-column processing

DIR1

DIR2

Tab. 15-8: *Description of the parameter DIR - Direction of machining*

MOD	Effect
0	The workpiece subroutine P is machined only once, without taking an offset into consideration
1	The workpiece subroutine P is called only once. The machining is offset in relation to the reference point
2	The workpiece subroutine P is called multiple times with changing offsets, compatible with the matrix points, with the exception of the reference point
3	The workpiece subroutine P is called for all matrix points with changing offsets

Tab. 15-9: *Description of the parameter MOD - Selection of workpieces*

Description

P Name workpiece subroutine

Name or complete path of a subroutine file containing the machining of a workpiece

MOD Selection of workpieces

Selects if the first workpiece or all other workpieces of the matrix are to be machined

MSP Reference point of main axis

Specifies the program coordinate offset of the main axis when machining the first workpiece

SSP Reference point of secondary axis

Specifies the program coordinate offset of the secondary axis when machining the first workpiece

LE Length of workpiece

Specifies the increment of the program coordinate offset between two subsequent columns (main axis)

WI Width of workpiece

Specifies the increment of the program coordinate offset between two subsequent lines (secondary axis)

NC Number of columns

Number of columns (direction of main axis)

NL Number of lines

Number of lines (direction of secondary axis)

DIR Direction of machining

Specifies whether the machining of workpieces follows the lines or the columns

TLC Number of used tools

Using this parameter the number of required tool changes can be reduced. The machining can be divided into substeps to machine with the same tool in case of an evaluation of the global variable "#TOCHCO%" and "#TOCHNW?" in the workpiece subroutine (see parameter P).

Special features

Before each workpiece subroutine call, the cycle automatically activates the punching function.

The cycle resets the path condition (G-group G0, G1 etc.) and the incremental dimension programming (G-group G90/G91) to the state before the cycle before each call of the workpiece subroutine. In case of all other conditions, the programmer of the workpiece subroutine has to ensure that no NC state, relevant for machining changes in between two calls and also has to ensure that these conditions are reset in time.

The program coordinate offset ATR/ATrans must not be changed in the workpiece subroutine.

Procedure/processing

The cycle cyclically calls the workpiece subroutine. Different offsets in the main and secondary axis become active to realize a meander-shaped alignment of the workpieces. An offset of MSP in the main axis and SSP in the secondary axis becomes active in the first workpiece (MOD 1 or 3); for other workpieces additionally by a multiple of the length LE and the width WI, corresponding to the matrix position. For MOD0, MSP and SSP are not taken into consideration, i.e., no offset takes place. The cycle ends at the last punching coordinate of the subroutine to the last workpiece.

Example

The subroutine /prog/punch.npg contains the machining of a workpiece. Call this subroutine multiple times, complying with the matrix-shaped alignment of the workpieces with a distance X=100 mm and Y=150 mm. The matrix has to be processed column by column without the first workpiece. During a second call, the workpiece is to be machined at the initial position.

Program for G219 Multiple part punching

```

N10 G219 (P /prog/punch.npg, MOD2, MSP100, SSP200, LE100, WI150, NC5, NL3, DIR2)
      ; all parts except for the first part
      ; processing column by column
N20 G219 (P /prog/punch.npg, MOD1, MSP100, SSP200); first part

```

The subroutine /prog/UPCNC.npg contains the machining of a workpiece. Call this subroutine multiple times, complying with the matrix-shaped alignment of the workpieces with a distance X=220 mm and Y=100 mm. First, machine the matrix with tool T19. The tools T25 and T29 are used in the second and third call.

Program for G219 Multiple part punching with several tools

N10 G219 (P /prog/UPCNC.npg, MSP30, SSP30, LE220, WI100, NC3, NL2, DIR2, TLC3); UPCNC used with 3 tools

Variable name	Value	Description
#TOCHCO%	1...TLC	Counter variable with number of tool call
#TOCHNW?	TRUE	First subroutine call with new tool number #TOCHNO%. Prompts to replace the tool.

Tab. 15-10: Significance of global variables in G219 for optimized tool change
Example for workpiece subroutine for G219 Multiple part punching with several tools

```
1 CASE #TOCHCO% OF
1 LABEL 1:
1 IF #TOCHNW? THEN
N20 T19 M6
1 ENDIF
N30 LEN=20
N40 PUN()
N50 G226 (MSP0, SSP0, LE200, WI80, TL10, TW10) ; nibble rectangle (with tool T19)
1 LABEL 2:
1 IF #TOCHNW? THEN
N60 T25 M6
1 ENDIF
N70 LEN=1
N80 PUPC ; punch subroutine "C" (with tool T25)
N90 SHT(X125)
N100 PUPC ; punch subroutine "C" (with tool T25)
N110 SHT()
1 LABEL 3:
1 IF #TOCHNW? THEN
N120 T29 M6
1 ENDIF
N130 LEN=1
N140 PUPN ;punch subroutine "N" (with tool T29)
1 ENDCASE
```

Fig. 15-5: Subroutine UPCNC

15.6 G224 Nibble line

Syntax	G224(MSP,SSP,A,LE,TL,TW)
Brief description	The cycle G224 Nibble line punches a continuous area by linear strokes with tool overlapping.

Overview

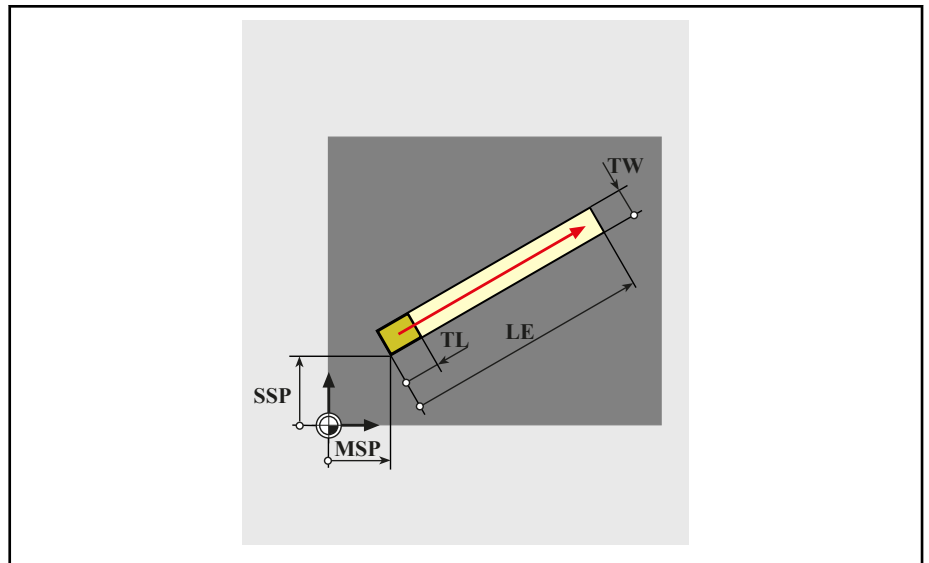


Fig. 15-6: Cycle G224 Nibble line

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
MSP	Reference point of main axis	Mandatory	Absolute	
SSP	Reference point of secondary axis	Mandatory	Absolute	
LE	Length	Mandatory	Incremental <>0, LE >= TL	
A	Angle	Optional	Absolute Degree	0
TL	Tool length	Mandatory	Incremental <>0	
TW	Tool width	Mandatory	Incremental	

Tab. 15-11: Parameter call

Description

MSP Reference point of main axis

The corner of the tool is positioned at this absolute position of the main axis during the first stroke

SSP Reference point of secondary axis

The corner of the tool is positioned at this absolute position of the secondary axis

LE Length

Increment of the total expansion of the cut out plane in line with the straight line

A Angle

Absolute angle to the main axis in whose direction all strokes are aligned, starting from the reference point

	TL Tool length
	Extension of the tool in longitudinal direction of the line The reference point changes to the opposite corner of the tool with a negative sign
	TW Tool width
	Extension of the tool in transverse direction to the line The reference point changes to the opposite corner of the tool with a negative sign
Special features	The cycle executes a machining and independently activates the punching function and the acceleration profile. Orient the tool before the cycle if the angle of rotation A is not part of the tool geometry.
Procedure/processing	The first punching position is calculated in relation to the reference point MSP/SSP, depending on TL/TW and their signs as well as the angle A, assuming that the tool is set to zero in the center point. More punching strokes along a line in angle A are executed in a number resulting in overlapping of the tool. The cycle ends at the last punching coordinate.
Example	Nibble a line at an 45° angle to the x-axis, starting at reference point X100 Y200 with a length of 35 mm. The tool dimension is 12 x 12 mm. <i>Program for G224 Nibble line</i>

```
N10 G224 (MSP100, SSP200, LE35, A45, TL12) ; call nibble line
```

15.7 G226 Nibble rectangular

Syntax	G226(MSP,SSP,LE,WI,TL,TW)
Brief description	The cycle G226 Nibble rectangular cuts an axis-parallel rectangular area, by subsequent strokes with tool overlapping.

Overview

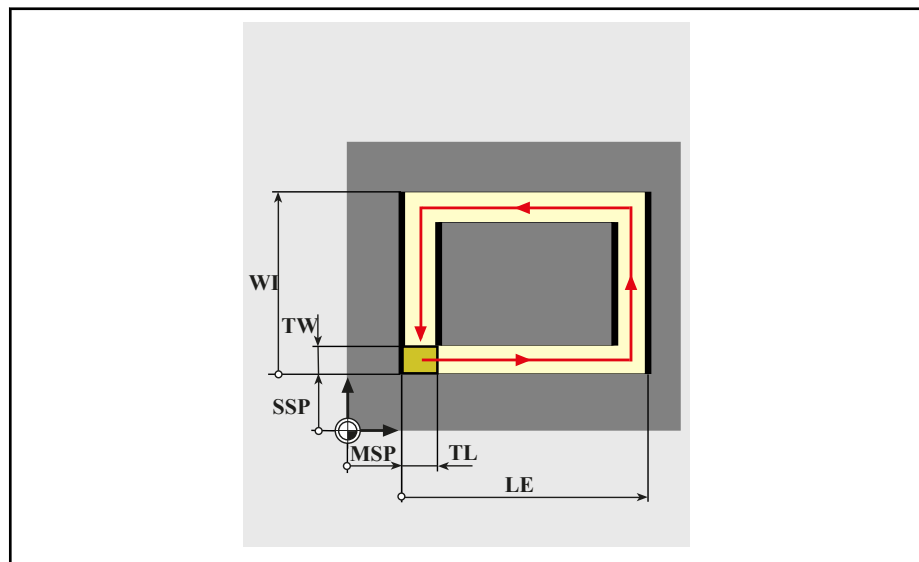


Fig. 15-7: Cycle G226 Nibble rectangular

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
MSP	Reference point of main axis	Mandatory	Absolute	
SSP	Reference point of secondary axis	Mandatory	Absolute	
LE	Length	Mandatory	Incremental <>0, LE >=3* TL	
WI	Width	Mandatory	Incremental <>0, WI >=3* TW	0
TL	Tool length	Mandatory	Incremental >0	
TW	Tool width	Optional	Incremental >0	TL

Tab. 15-12: Parameter call

Description

MSP Reference point of main axis

The external corner of the cut-out rectangle is positioned at this absolute position of the main axis

SSP Reference point of secondary axis

The external corner of the cut-out rectangle is positioned at this absolute position of the secondary axis

LE Length

Increment of the total expansion of the cut out plane, parallel to the main axis, starting from the reference point

WI Width

Increment of the total expansion of the cut out plane, parallel to the secondary axis, starting from the reference point

TL Tool length

Extension of the tool in the direction of the main axis

TW Tool width

Extension of the tool in the direction of the secondary axis

The reference point changes to the opposite corner of the tool with a negative sign

Special features

The cycle executes a machining and independently activates the punching function and the acceleration profile.

Procedure/processing

The first punching position is calculated in relation to the reference point MSP/SSP, depending on TL/TW and the sign of LE/WI, assuming that the tool is set to zero in the center point. More punching strokes in the direction of LE and subsequently in other directions up to closing the rectangle are executed, whereas overlapping of the tool is complied with. The cycle ends at the last punching coordinate before the reference point.

Punching cycles

Example Nibble a rectangle with a length of 150 mm and a width of 170 mm. The reference point is at X100 and Y200. The tool dimension is 10 x 12 mm (length x width).

Program for G226 Nibble rectangular

```
N10 G226 (MSP100, SSP200, LE150, WI170, TL10, TW12) ; rectangle 150x170
```

16 Measuring cycles

Functions *The following functions are available as cycles:*

- *Calibrate tool probe*
 - G311 Tool length
 - G315 Tool radius
- *Probe tool*
 - G301 Tool length
 - G303 Tool radius
- *Calibrate workpiece probe*
 - G317 at a known point
 - G318 in an adjusting ring
 - G319 at a spigot
- *Probe workpiece*
 - G321 Point
 - G323 Check workpiece availability
 - G324 Boring
 - G327 Spigot
 - G329 Land
 - G331 Groove
 - G333 Angle
 - G628 Orientation of a plane
 - G629 Center point of a calibrated ball
- *Measuring value evaluation*
 - G541 Tolerance monitoring
 - G542 Averaging
 - G543 Percentage compensation

A detailed description of this function can be found in "MTX 15VRS Measuring Functions" (DOK-MTX***-MES*FUN*V15-AP01-EN-P), chapter "Measuring cycles in automatic".

17 TCH tool change

Description: The TCH facilitates the tool change for the following automatic tool changer types:

- Pickup automatic tool changer
 - Manual tool changer
 - Pickup automatic tool changer (single-spindle machine)
 - Pickup automatic tool changer (multi-spindle machine)
- Revolver automatic tool changer

TCH cycle description, see documentation "IndraMotion MTX 15VRS Commissioning", (DOK-MTX***-STARTUP*V15-CO01-EN-P), chapter "Tool management with standard tool change".

18 Programming instruction for simulation

18.1 STOCK/BLOCK Cuboid raw piece (milling)

Syntax STOCK("BLOCK",<L>,<W>,<H>,<X>,<Y>,<Z>,<V>)

Brief description A new cuboidal raw part is generated at or added to the selected tool holder. It is intended for machining simulation of prismatic parts.

Overview

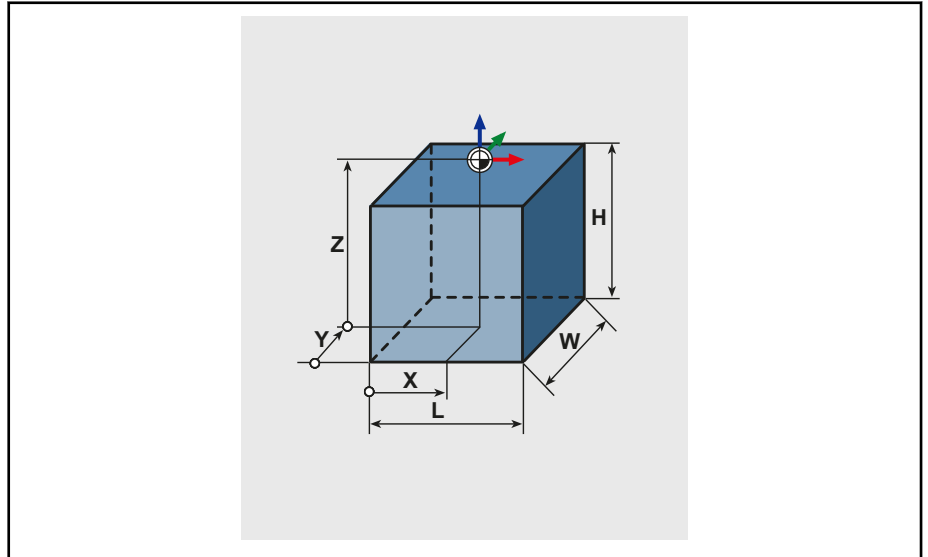
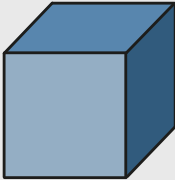
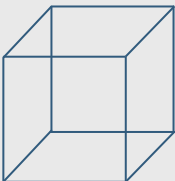


Fig. 18-1: STOCK/BLOCK instruction cuboid raw piece

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
L	Length	Mandatory	>0	
W	Width	Mandatory	>0	
H	Height	Mandatory	>0	
X	Zero point in X	Optional	Incremental	0
Y	Zero point in Y	Optional	Incremental	0
Z	Zero point in Z	Optional	Incremental	0
V	Modeling	Optional		0

Tab. 18-1: Parameter call

V	Effect
0	Volume
1	Enveloping body
V=0  V=1 	

Tab. 18-2: Description of the parameter V - Modeling

Description

- L length**
Defines the X-expansion of the cuboid.
- W width**
Defines the Y-expansion of the cuboid.
- H height**
Defines the Z-expansion of the cuboid.
- X, Y, Z zero point**
Specifies the distance of the current workpiece zero point from the front left and the bottom corner.
- V modeling**
Specifies if a volume model for removal simulation or an enveloping body is to be drawn. The latter is an alternative to import a CAD model in a complex geometry.

- Special features** The parameters are position parameters without an address.
- Process/processing** The workpiece is displayed as compatible with the previously activated work-piece zero point. Toolpath is deleted.
- Example** Cuboids in volume representation, 100 in length, 70 in width and 30 in height. The zero point of the workpiece (G54) should be positioned in the center of the upper side.

Program:

```
N10 G54 STOCK("BLOCK",100,70,30,50,35,30)
```

18.2

STOCK/CYLINDER Cuboid raw piece (milling)

Syntax

STOCK("CYLINDER",<D1>,<D2>,<H>,<X>,<Y>,<Z>,<V>)

Brief description A new cylindrical raw part, optionally with continuous internal boring, is generated at or added to the selected tool holder. It is intended for machining simulation of prismatic parts.

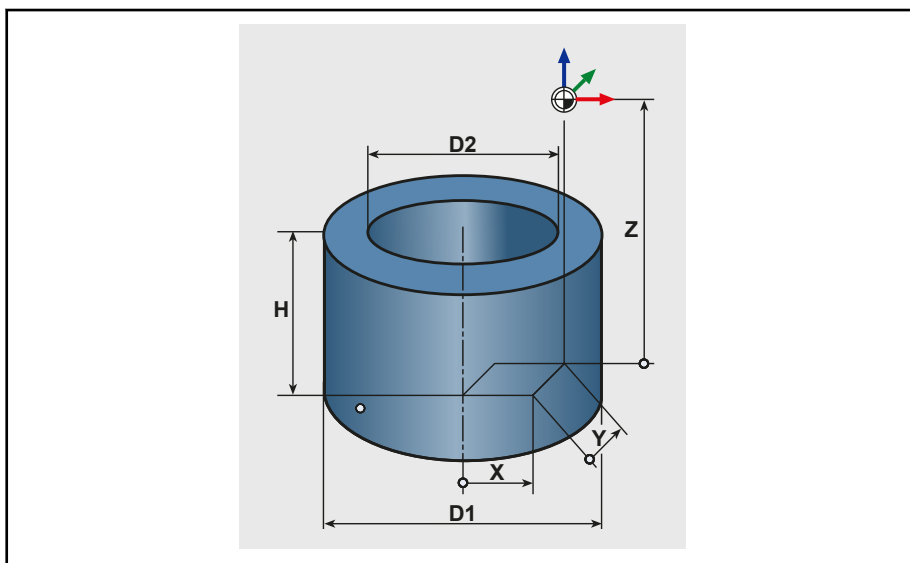
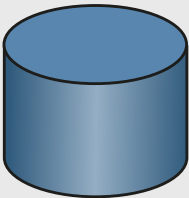
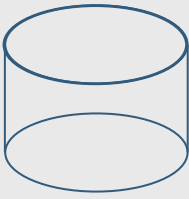
Overview

Fig. 18-2: Instruction STOCK/CYLINDER cylindrical raw part

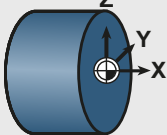
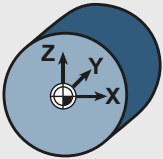
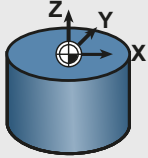
Parameter

Parameter	Designation	Properties	Values	Pre-assignment
D1	Outer diameter	Mandatory	>0	
D2	Inside diameter	Optional	>=0	0
H	Height	Mandatory	>0	
X	Zero point in X	Optional	Incremental	0
Y	Zero point in Y	Optional	Incremental	0
Z	Zero point in Z	Optional	Incremental	0
V	Modeling and orientation	Optional		0

Tab. 18-3: Parameter call

V units digit	Effect
0	Volume
1	Enveloping body
V=x0  V=x1 	

Tab. 18-4: Description of the units digit parameter V-modeling

V tens digit	Effect
0	Center axis Z-parallel
1	Center axis Y-parallel
2	Center axis X-parallel
V=2x  V=1x  V=0x 	

Tab. 18-5: Description of the tens digit of parameter V - orientation

Description**D1 outer diameter**

Defines the outer diameter of the cylinder.

D2 inner diameter

In case of values greater than 0, defines a uniform internal boring.

H height

Defines the height of the cuboid.

X, Y, Z zero point

Specifies the distance of the current workpiece zero point from the center point of the left front or bottom base.

V-modeling and orientation

The units digit specifies if a volume model for removal simulation or an enveloping body is to be drawn. The latter is an alternative to import a CAD model in a complex geometry.

The tens digit influences the center axis orientation.

Special features

The parameters are position parameters without an address.

Process/processing

The workpiece is displayed as compatible with the previously activated workpiece zero point. Toolpath is deleted.

Example

Hollow cylinder in volumetric display with a diameter of 100, an inside diameter of 40 and a height of 30. The center line has to be parallel to Y. The origin of the workpiece zero point (G54.2) is positioned in the center of the Y-positive upper side.

Program:

```
N10 G54.2 STOCK("CYLINDER",100,40,30,,30)
```

18.3 STOCK/BAR Bar section (turning)

Syntax STOCK("BAR",<L>,<D1>,<D2>,<Z>,<V>)

or

STOCK(<L>,<D1>,<D2>,<Z>,<V>)

Brief description

A new cylindrical raw part, optionally with internal bore is generated at or added to the selected tool holder. It is intended for turning.

Overview

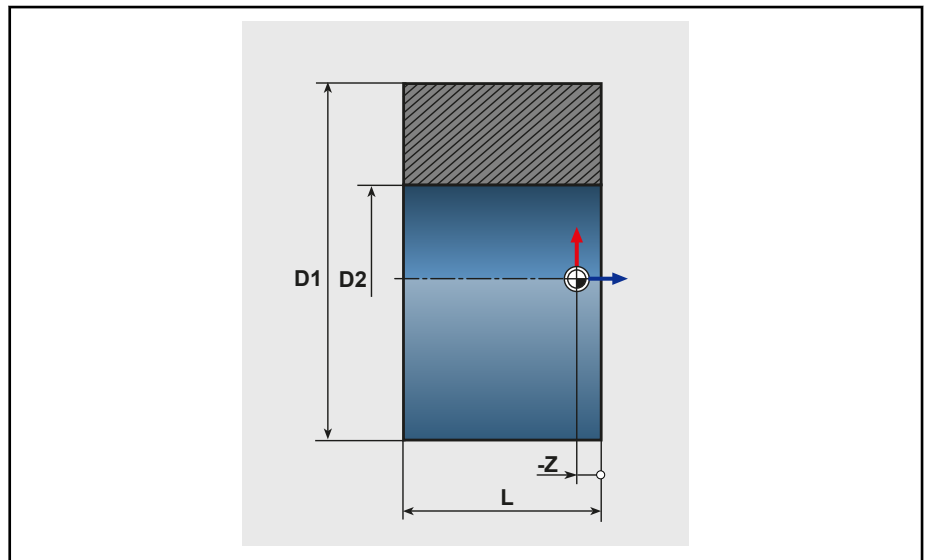


Fig. 18-3: Instruction STOCK/BAR bar section

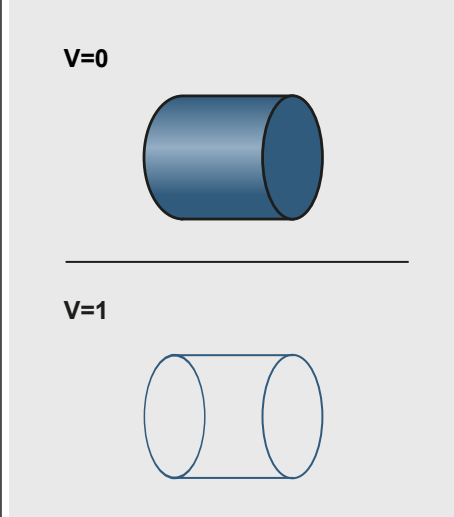
Parameter

Parameter	Designation	Properties	Values	Pre-assignment
L	Length	Mandatory	>0	
D1	Outer diameter	Mandatory	>0	

Programming instruction for simulation

D2	Inside diameter	Optional	≥ 0	0
Z	Zero point in Z	Optional	Incremental	0
V	Modeling	Optional		0

Tab. 18-6: Parameter call

V modeling	Effect
0	Volume
1	Enveloping body
	

Tab. 18-7: Description of the units digit parameter V-modeling

Description	L length Defines the Z-expansion. D1 outer diameter Defines the outer diameter. D2 inner diameter In case of values greater than 0, defines a uniform internal boring. Z zero offset Specifies the distance of the current workpiece zero point from the limiting surface to the right (usually negative). V modeling Specifies if a volume model for removal simulation or an enveloping body is to be drawn. The latter is an alternative to import a CAD model in a complex geometry.
Special features	The parameters are position parameters without an address.
Process/processing	The workpiece is displayed as compatible with the previously activated workpiece zero point. Toolpath is deleted.
Example	Bar section in volumetric display with an outer diameter of 60, a length of 102. The workpiece zero point (G54.3) is to be located 2 mm to the left of the limiting surface to the right.

Program:

N10 G54.3 STOCK("BAR",102,60,,,-2)

18.4 STOCK/POLYGON Multi-edge (turning)

Syntax STOCK("POLYGON",<L>,<D>,<N>,<Z>,<OA>,<V>)

Brief description A new raw part is generated at or added to the selected tool holder as multi-edge. It is intended for turning.

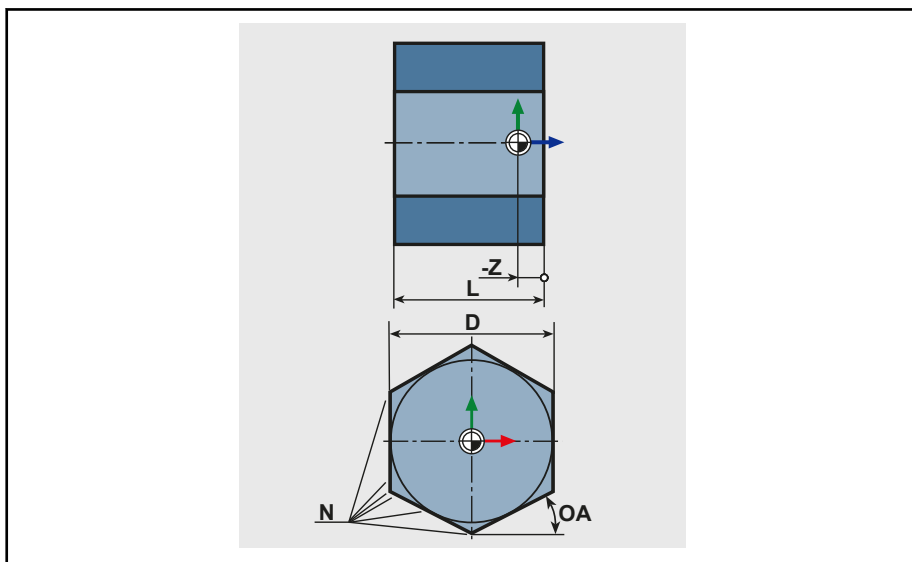
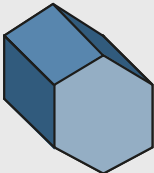
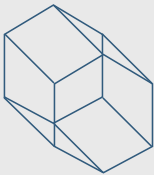
Overview

Fig. 18-4: Instruction STOCK/POLYGON multi-edge

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
L	Length	Mandatory	>0	
D	Diameter (wrench size)	Mandatory	>0	
N	Number of edges	Mandatory	>2	
Z	Zero point in Z	Optional	Incremental	0
OA	Orientation angle	Optional	Degree	0
V	Modeling	Optional		0

Tab. 18-8: Parameter call

V modeling	Effect
0	Volume
1	Enveloping body
V=0  V=1 	

Tab. 18-9: Description of the parameter V - Modeling

Description

L length	Defines the Z-expansion.
D diameter	Defines the diameter as key width.
N Number of edges	Defines the number of edges.
Z zero point	Specifies the distance of the current workpiece zero point from the limiting surface to the right (usually negative).
OA Orientation angle	Specifies the phase offset of a profile edge in the angle of rotation C.
V modeling	Specifies if a volume model for removal simulation or an enveloping body is to be drawn. The latter is an alternative to import a CAD model in a complex geometry.
Special features	The parameters are position parameters without an address.
Process/processing	The workpiece is displayed as compatible with the previously activated work-piece zero point. Toolpath is deleted.
Example	Hexagon in volumetric display with a diameter of 60, a length of 82. The workpiece zero point (G54.4) is to be located 2 mm to the left of the limiting surface to the right. A profile edge has to be available at angle C90. <i>Program:</i> N10 G54.4 STOCK("POLYGON",82,60,6,-2,30)

18.5 STOCK/SHEET Sheet (punching)

Syntax STOCK("SHEET",<L>,<W>,<H>,<X>,<Y>,,<V>)

Brief description A new raw part is generated at or added to the selected tool holder as rectangular tip. It is intended for machining simulation of sheet machining (e.g. punching).

Overview

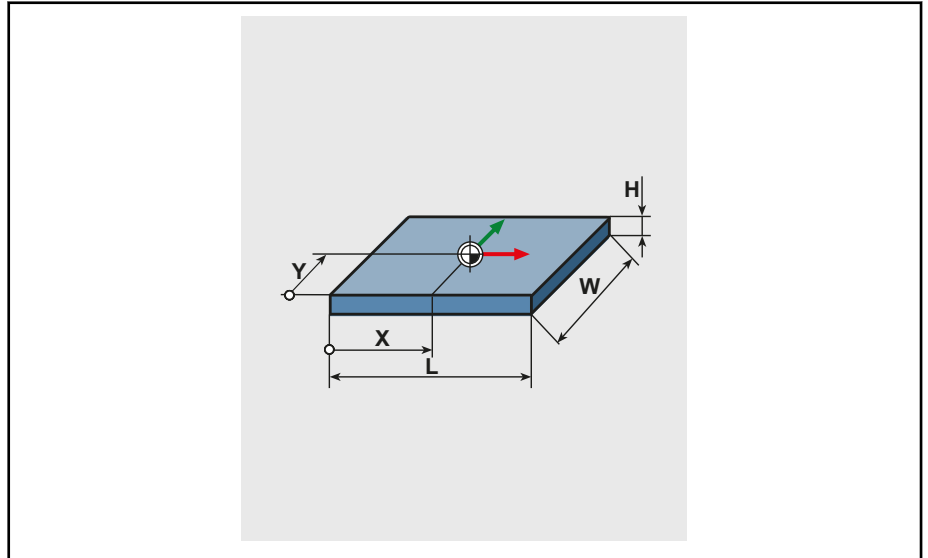
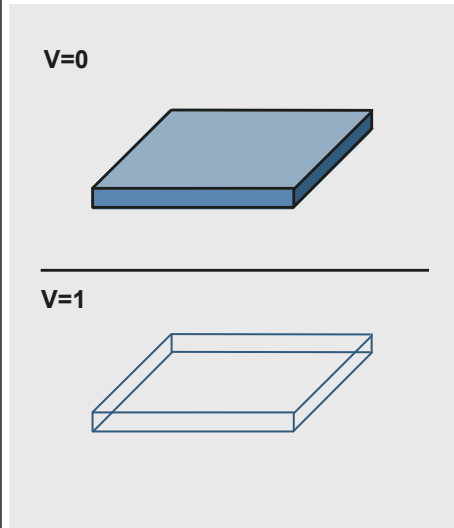


Fig. 18-5: Instruction STOCK/SHEET sheet

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
L	Length	Mandatory	>0	
W	Width	Mandatory	>0	
H	Thickness	Mandatory	>0	
X	Zero point in X	Optional	Incremental	0
Y	Zero point in Y	Optional	Incremental	0
V	Modeling	Optional		0

Tab. 18-10: Parameter call

V	Effect
0	Volume
1	Enveloping body
	

Tab. 18-11: Description of the parameter V - Modeling

Description	L length Defines the X-expansion of the sheet. W width Defines the Y-expansion of the sheet. H Thickness Defines the sheet thickness. X, Y zero point Specifies the distance of the current workpiece zero point from the front left corner. V modeling Specifies if a volume model for removal simulation or an enveloping body is to be drawn. The latter is an alternative to import a CAD model in a complex geometry.
Special features	The parameters are position parameters without an address.
Process/processing	The workpiece is displayed as compatible with the previously activated workpiece zero point. Toolpath is deleted.
Example	Plate in volumetric display with 1000 in length, 500 in width and 2.5 in thickness. The workpiece zero point (G54.5) has to be in the center. <i>Program:</i> <pre>N10 G54.5 STOCK("SHEET",1000,500,2.5,500,250)</pre>

18.6 STOCKIMP Import model

Syntax	STOCKIMP(P, TGT,X,Y,Z,RS,A1,A2,A3)
Brief description	Imports a CAD model displayed at a workpiece pick up.

Overview

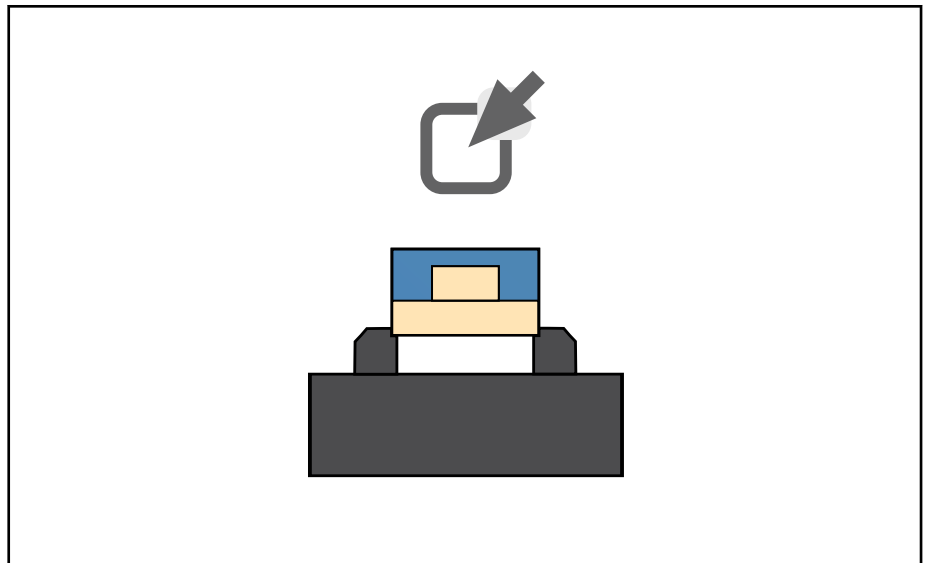


Fig. 18-6: Instruction STOCKIMP Import model

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
P	File name	Mandatory	Max. 255 characters	
TGT	Import target	Optional		1
X	Translation X	Optional	Incremental	0
Y	Translation Y	Optional	Incremental	0
Z	Translation Z	Optional	Incremental	0
RB	Position of the axes of rotation	Optional		1
RS	Order or rotation axes	Optional	max. 3 characters	ZYZ
A1	Angle of rotation first rotation	Optional	Degree	0
A2	Angle of rotation second rotation	Optional	Degree	0
A3	Angle of rotation third rotation	Optional	Degree	0

Tab. 18-12: Parameter call

TGT	Effect
1	Imports a raw part
2	Imports a finished part

TGT	Effect
3	Imports a clamping device
4	Imports a drawing

Tab. 18-13: Description of the parameter TGT - Import target

RB	Effect
1	Fixed in space The rotation axes contain their initial position
2	Subsequently: The rotary axes rotate synchronously with the model

Tab. 18-14: Description of the parameter RB - Position of rotary axes

Description

P File name

The extension ".stl" can be omitted. Path names of the local PC can also be used as path names of the MTX file system.

If only a file name is specified without path, the file is searched for according to the search strategy for subroutines, starting from the main program folder.

TGT Import target

Specifies the type of model use.

X, Y, Z translation

Specifies an offset after the rotation related to the coordinate jump of the model.

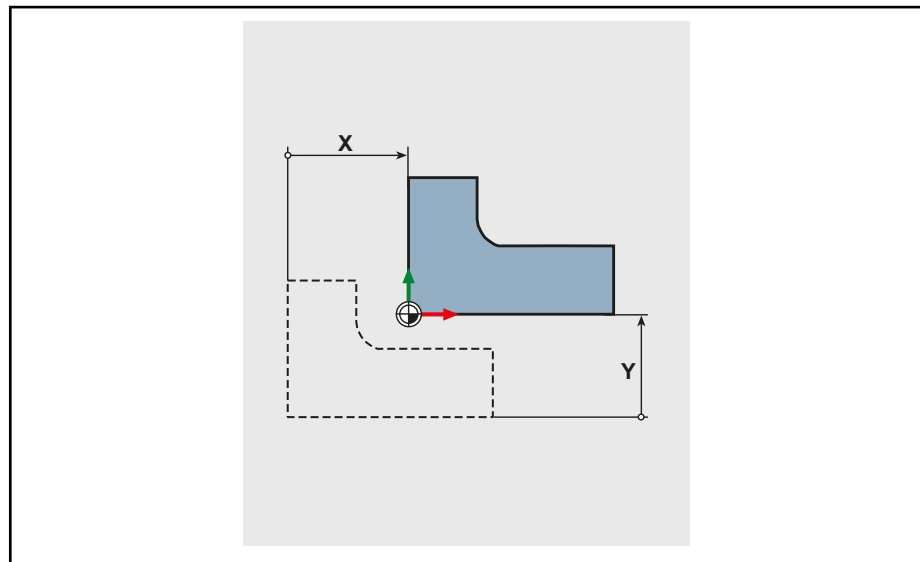


Fig. 18-7: Translation in x and y. The target position is displayed in color

RB Position of the rotary axes

Specifies if the subsequent rotations refer to the initial position of the rotary axes.

RS Order of the rotary axes

Specifies the rotary axes that are to be subsequently used in each rotation, starting with the first character of parameter RS. Only the characters x, y and z are permitted.

A1, A2, A3 angle of rotation

Angle of rotation for the first, second and third rotation, corresponding with RS.

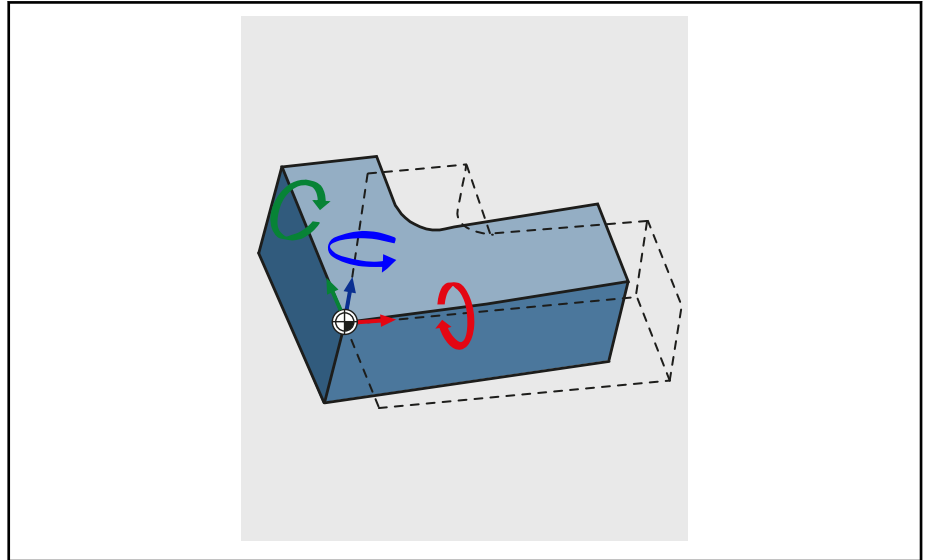


Fig. 18-8:

Special features	The parameters have addresses.
Process/processing	The model is displayed as compatible with the previously activated workpiece zero point. The tool path is deleted upon the import as workpiece.
Example	The "finalwp.stl" model contained in the folder of the main program is to be imported as finished model. Execute a rotation around the axes fixed in space: x at 180° and z at 90°. To subsequently position the model to the correct position in relation to the workpiece zero point (G55), an offset around Y-55 is required.

Program:

```
N10 G55 STOCKIMP(P finalwp, TGT2, Y-55, RS XZ, A1 180, A2 90)
```

18.7 STOCK/REMOVE Remove workpieces/models

Syntax	STOCK("REMOVE", <TGT> or STOCK()
Brief description	Removes a workpiece or another geometry from the selected tool holder.

Overview

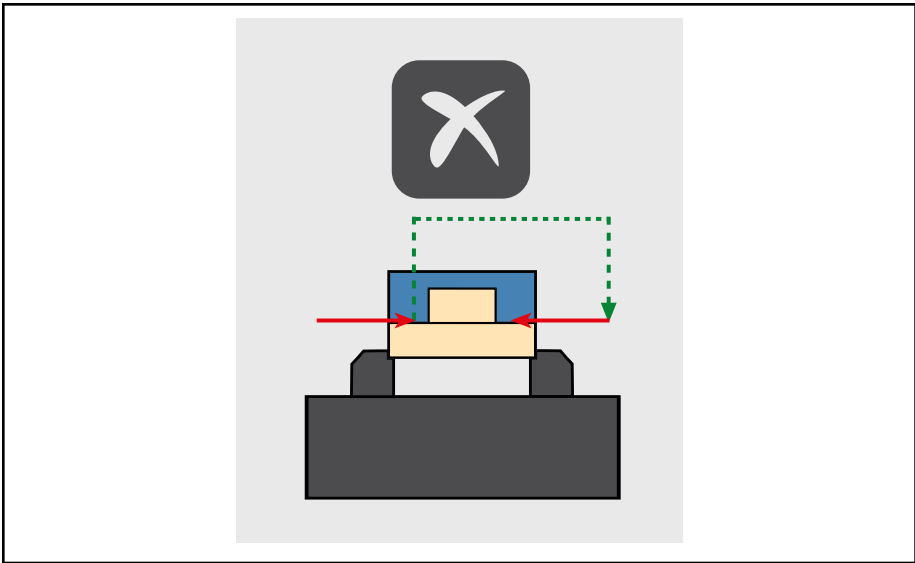


Fig. 18-9: Instruction STOCK/REMOVE Remove workpieces/models

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
TGT	Geometry to be removed	Optional		0

Tab. 18-15: Parameter call

TGT	Effect
0	Complete Removes all geometries of all tool holders
1	Removes workpiece/raw part and tool path from the selected tool holder
2	Removes finished part from the selected tool holder
3	Removes clamping device from the selected tool holder
4	Removes drawing from the selected tool holder
5	Removes tool path from the selected tool holder

Tab. 18-16: Description of parameter TGT - Geometry to be removed

Description

TGT Geometry to be removed

The entire parameter list is omitted in case of default value TGT=0.

Special features

The parameter does not have an address.

An automatically loaded workpiece look-ahead (TGT=0 or 2) cannot be deleted. It can only be overwritten by an import or removed by the dialog "Reset raw part", menu item "removing incl. finished part, clamping device and drawings".

Process/processing

The selected geometries are selected.

Example Delete workpiece, toolpath and finished part.

Program:

```
N10 STOCK("REMOVE",1) ;removes workpiece and tool path
N20 STOCK("REMOVE",2) ;removes finished part
```

18.8 STOCK/RENEW Renew workpiece

Syntax STOCK("RENEW",<TN>)

Brief description Renews the workpiece of the affected tool holder or all workpieces by replacing the workpiece(s) by the last defined raw part geometry. The function also deletes the tool path. It is not relevant whether the workpiece has previously been machined or removed.

Overview

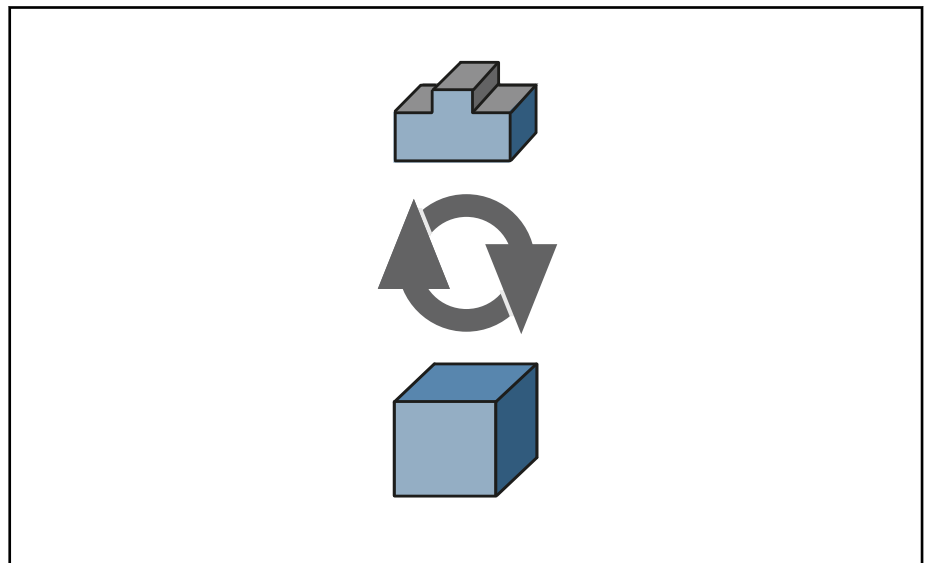


Fig. 18-10: Instruction STOCK/RENEW Renew workpiece

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
TN	Affected tool holder	Optional		0

Tab. 18-17: Parameter call

TN	Effect
0	Renews all workpieces
1	Renews the workpiece at the selected tool holder

Tab. 18-18: Description of the parameter TN - affected tool pick up

Description

TN Affected tool holder

Specifies if the renewal refers to the selected tool holder or to all.

Special features

The parameter does not have an address.



Generally, the result is the same, whether a new workpiece is defined each time in the program or if it is renewed. The renewal performance can be higher in case of complex, imported geometries.

There are settings in the simulation options that automatically executes renewal of the workpiece/workpieces machined by the relevant channel upon the program start.

Process/processing The workpiece geometry is restored as last defined.

Example All workpieces have to be renewed.

Program:

```
N10 STOCK("RENEW")
```

18.9

STOCKSEL Select tool holder

Syntax STOCKSEL(<NAM>)

Brief description Selects a tool holder via their names. All subsequent calls of the STOCK and STOCKIMP instructions in the same channel refer to this tool holder. This instruction is required if a tool machine has several tool holders.

Overview

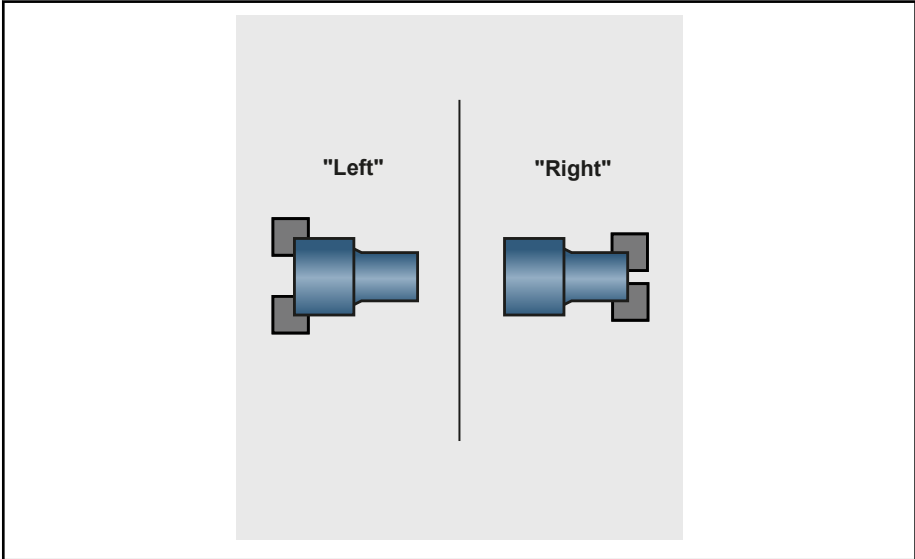


Fig. 18-11: Select instruction STOCKSEL tool holder

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
NAM	Tool holder name	Optional	Max. 32 characters	First tool holder of machine model

Tab. 18-19: Parameter call

Description

NAM Name of pick up tool

Defines the tool holder to be selected.

Special features The parameter does not have an address.

Process/processing The new selection is applied immediately in the current channel until cancellation of the selection.

Example Two tool holder are available in a machine model, whereas "Left" is the first tool holder in the machine model. In case of pick up "Left", delete the workpiece, in case of "Right", define a bar section.

Program:

```
N10 STOCKSEL()           ; Select "Left"
N20 STOCK("REMOVE",1)    ; Remove workpiece
N30 STOCKSEL("Right")    ; Select "Right"
N40 STOCK("BAR",61,50,, -60); Define bar section
```

18.10 STOCKADD Replace/add workpieces/models

Syntax STOCKADD(<ADD>)

Brief description The following raw part definitions and model imports replace the existing geometry or are added to the existing geometry.

Parameter

Parameter	Designation	Properties	Values	Pre-assignment
ADD	Effect of raw part definitions/imports	Optional		0

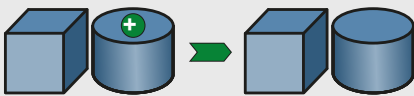
Tab. 18-20: Parameter call

ADD	Effect
0	Replace
1	Add

ADD=0



ADD=1



Tab. 18-21: Description of the parameter ADD

Description

ADD Effect of raw part definitions/imports

Defines the tool holder to be selected.

Special features

The parameter does not have an address.



In the active version, adding cannot neither be applied to workpieces in volume modeling nor to finished parts.

If "Adding" is active, a replacement can also be achieved by deleting the geometry.

Process/processing

The new selection is applied immediately in the current channel until cancellation of the selection.

Example

Import a clamping device consisting of two parts.

Program:

```
N10 STOCKADD()          ; subsequently results in replacement of
                        ; existing clamping device
N20 STOCKIMP("CLAMP1",3); loads CLAMP1.stl as clamping device
N30 STOCKADD(1)         ; switch to add
N40 STOCKIMP("CLAMP2",3); adds CLAMP2.stl to clamping devices
```


19 Service and support

Our worldwide service network provides an optimized and efficient support. Our experts offer you advice and assistance should you have any queries. You can contact us **24/7**.

Service Germany Our technology-oriented Competence Center in Lohr, Germany, is responsible for all your service-related queries for electric drive and controls.

Contact the **Service Hotline** and **Service Helpdesk** under:

Phone:	+49 9352 40 5060
Fax:	+49 9352 18 4941
E-mail:	service.svc@boschrexroth.de
Internet:	http://www.boschrexroth.com

Additional information on service, repair (e.g. delivery addresses) and training can be found on our internet sites.

Service worldwide Outside Germany, please contact your local service office first. For hotline numbers, refer to the sales office addresses on the internet.

Preparing information To be able to help you more quickly and efficiently, please have the following information ready:

- Detailed description of malfunction and circumstances
- Type plate specifications of the affected products, in particular type codes and serial numbers
- Your contact data (phone and fax number as well as your e-mail address)

Index

0 ... 9

2-point contour.....	276, 288
3-point contour.....	277, 290
4-point contour.....	278, 291

A

Abbreviations.....	7
About this documentation.....	1
Validity of the documentation.....	1
ANSI Z535.6-2006.....	6

B

Bar section (turning).....	365
Boring cycles	
G87 Back boring.....	45

C

Complaints.....	7
Contour drafts.....	275
Contour programming G31.....	275
General information.....	275
Contour programming G31 for milling.....	276
Contour programming G31 for turning.....	288
Intersection transitions.....	300
Point contours.....	288
Tangential transitions.....	293
Criticism.....	7
Cuboid raw piece (milling).....	361, 362
Customer Feedback.....	7

D

Defining the contour.....	147, 251
Documentation	
Change record.....	1
Drill cycles	
G85 Reaming.....	39
Drilling cycles.....	27
G81 Drilling.....	27
G82 Countersinking.....	29
G83 Deep hole drilling.....	32
G84 Tapping.....	36
G86 Fine boring.....	42
G88 Gun drilling.....	48
G89 Tapping with Chip Breaking.....	52

E

Engrave.....	181

F

Feedback.....	7
Fixed machine operator cycles.....	14

G

G-codes, assignment.....	11

G31.....	275
G81 Drilling.....	27
G82 Countersinking.....	29
G83 Deep hole drilling.....	32
G84 Tapping.....	36
G85 Reaming.....	39
G86 Fine boring.....	42
G87 Back boring.....	45
G88 Gun drilling.....	48
G89 Tapping with chip breaking.....	52
G111 Hole circle, rotary axis.....	57
G112 Hole circle, main spindle.....	59
G113 Hole circle, plane.....	63
G114 Hole pattern, line.....	68
G115 Hole pattern, matrix.....	72
G116 Hole pattern, frame.....	76
G160 Longitudinal turning.....	193
G161 Face turning.....	195
G162 Tapered turning.....	198
G165 Thread single step.....	200
G166 Thread sequence.....	203
G167 Thread turning.....	209
G168 Face thread.....	214
G169 Tapered thread.....	220
G170 Contour finishing.....	252
G171 Contour roughing, longitudinal.....	254
G172 Contour roughing, plane.....	259
G173 Contour roughing, parallel to contour.....	263
G174 Contour grooving.....	266
G191 Rounded groove turning.....	226
G192 Rounded groove turning.....	235
G196 Thread Undercut.....	240
G197 Undercut Turning.....	245
G201 Text engraving	
Linear.....	185
G202 Text engraving	
Circular.....	189
G213 Hole circle punching.....	342
G215 Hole pattern line punching.....	343
G215 Hole pattern matrix punchin.....	346
G219 Multiple part punching.....	348
G224 Nibble line.....	352
G226 Nibble rectangular.....	354
G230 Swiveling.....	305
G261.....	81
G262.....	87
G263.....	91
G266.....	95
G267 Face milling, circle.....	100
G271 Rectangular pocket milling.....	105
G272.....	109
G273 User-defined contour pocket milling.....	147
G274.....	112
G275.....	117
G276 Rectangular pin milling.....	124

Index

G277.....	130
G278 User-defined contour pivot milling.....	156
G279.....	134
G280.....	139
G281 Milling contour pocket with islands.....	172
G288 Defining island contour.....	168
G289 Defining starting points.....	170
G290 Path milling.....	162
G591 HSC settings.....	328
G592 Approximated free form surface mill- ing.....	331
GETBQ.....	337

H

Hazard warnings.....	5
Helpdesk.....	379
Hole circle	
G111.....	57
G111 Hole circle, rotary axis.....	57
G112.....	59
G112 Hole circle, main spindle.....	59
G113.....	63
Hole pattern	
G113 Hole circle, plane.....	63
G114 Hole pattern, line.....	68
G115 Hole pattern, matrix.....	72
G116 Hole pattern, frame.....	76
Hole patterns.....	57
Hotline.....	379

I

Import model.....	370
Information representation	
Names and abbreviations.....	7
INT_TO_STR.....	339
Intended use	
Areas of application.....	9
Introduction.....	9
Use cases.....	9
Intersection of circle-circle.....	287, 302
Intersection of circle-straight line.....	286, 301
Intersection of straight line-circle.....	285, 300
Intersection transitions.....	285, 300

M

MEAS.....	357
Measuring cycles	
MEAS.....	357
Milling contour pocket with islands cycle.....	168
Milling cycles.....	81
G261 Thread milling.....	81
G262 Thread milling.....	87
G263 Groove circular milling.....	91
G266 Face milling, rectangle.....	95
G267 Face milling, circle.....	100
G271 Rectangular pocket milling.....	105
G272 Circular pocket milling.....	109

G274 Straight slot milling.....	112
G275 Curved slot milling.....	117
G276 Rectangular pin milling.....	124
G277 Circular pin milling.....	130
G279 Land straight milling.....	134
G280 Land curved milling.....	139
Milling cycles, contour-dependent.....	147
G273 User-defined contour pocket mill- ing.....	147
G278 User-defined contour pivot milling.....	156
G281 Milling contour pocket with islands.....	172
G288 Defining island contour.....	168
G289 Defining starting points.....	170
MOD0.....	330
MOD1.....	329
MOD2.....	329
MOD3.....	329
MOD4.....	330
MOD31.....	329
MOD32.....	329
MOD33.....	329
MOD41.....	330
MOD42.....	330
MOD43.....	330
Modal cycle calls, rules.....	12
Mode machining MOD.....	329
Multi-edge (turning).....	367

O

Overview of NC cycles	
G-codes for measuring cycles.....	18
Overview on NC cycles.....	15
Cycles without G-codes.....	19
Modal G-codes for cycle calls.....	15
Non-modal G-codes for cycle calls.....	15
Specified end user cycles, local.....	17
Specified end user cycles, modal.....	18

P

Path milling	
G290 Path milling.....	162
Point contours.....	276, 288
2-point contour.....	276, 288
3-point contour.....	277, 290
4-point contour.....	278, 291
Intersection transitions.....	285
Tangential transitions.....	280
Program instructions for simulation.....	361
Programming.....	181, 193, 275
Drilling cycles.....	27
Engrave.....	181
G230 Swiveling.....	305
Hole patterns.....	57
Milling cycles.....	81
Punching cycle.....	341
Punching cycle.....	341

R

REAL_TO_STR.....	340
Remove workpieces/models.....	373
Renew workpiece.....	375
Replace/add workpieces/models.....	377

S

Safety instructions.....	5
Select tool holder.....	376
Service hotline.....	379
SETBQ.....	334
Sheet (punching).....	369
Signal alert symbol.....	6
Signal words.....	6
STOCK/BAR.....	365
STOCK/BLOCK.....	361
STOCK/CYLINDER.....	362
STOCK/POLYGON.....	367
STOCK/REMOVE.....	373
STOCK/RENEW.....	375
STOCK/SHEET.....	369
STOCKADD.....	377
STOCKIMP.....	370
STOCKSEL.....	376
STP.....	333
Suggestions.....	7
Support.....	379
Swiveling.....	305
Symbols.....	7

T

Tangential transition circle, straight line....	282, 295
Tangential transition of straight line-circle.....	280, 293
Tangential transitions.....	280, 293
Tangential transitions from circle-circle-circle.....	284, 298
Tangential transitions from circle-straight line-circle.....	283, 297
Target groups.....	1
TCH.....	359
Technology cycles.....	319, 333
G501 TFS Cycle turning.....	319
General information.....	319
GETBQ.....	337
INT_TO_STR.....	339
REAL_TO_STR.....	340
SETBQ.....	334
STP.....	333
TFS Cycle milling.....	324
Tool change	
TCH.....	359
Turning cycles.....	193
G160 Longitudinal turning.....	193
G161 Face turning.....	195
G162 Tapered turning.....	198
G165 Thread single step.....	200

G166 Thread sequence.....	203
G167 Thread turning.....	209
G168 Face thread.....	214
G169 Tapered thread.....	220
G191 Rounded groove turning.....	226
G192 Rounded groove turning.....	235
G196 Thread Undercut.....	240
G197 Undercut Turning.....	245
Turning cycles, contour-dependent.....	251
G170 Contour finishing.....	252
G171 Contour roughing, longitudinal.....	254
G174 Contour grooving.....	266
Turning, contour-dependent	
G173 Contour roughing, parallel to contour.....	263
Turning, cycles, contour roughing	
G172 Contour roughing, plane.....	259

U

Unintended use.....	10
Consequences, disclaimer.....	9
User cycles, rules.....	13
Cycles with addressed parameters.....	13
Cycles with position parameters.....	14
Reserved directories on the control.....	13
Reserved directories on the PLC or operator panel.....	13

W

Warnings.....	5

Notes

Notes

Bosch Rexroth AG

P.O. Box 13 57

97803 Lohr a.Main, Germany

Bgm.-Dr.-Nebel-Str. 2

97816 Lohr a.Main, Germany

Phone +49 9352 18 0

Fax +49 9352 18 8400

www.boschrexroth.com/electrics



R911394940