



HEIDENHAIN







Programming Manual
V600-02

MillPlus *IT*





NC Software
538 952-xx
538 953-xx
538 954-xx
538 955-xx
538 956-xx

English (en)
11/2008









Controls on the visual display unit

	Select window
	User keys
	Info key
	Function soft keys Machine-function soft keys







Keys for main operating modes

	Manual operating mode
	Automatic operating mode
	Programming operating mode
	Control (Setup) operating mode












Keys for machine functions

		Key freely assignable via IPLC
		Key freely assignable via IPLC
		Key freely assignable via IPLC
		Key freely assignable via IPLC




Keys for spindle functions

	Spindle speed 100%
	Increase spindle speed
	Decrease spindle speed
	Spindle on, CCW
	Spindle stop
	Spindle on, CW


Manual operation

		Axis-direction keys for three main axes
		Axis-direction keys for further axes
		Axis-direction keys for 4th axis
		Rapid traverse
		Rapid traverse override/feed rate override
		Emergency stop
		NC on






Start / stop keys

	START
	Feed rate STOP
	Feed rate and spindle STOP

Touchpad

	Positioning of the mouse cursor Selection and context buttons
--	--

ASCII keyboard (NC functionality)

	Menu key: Open a menu from the menu row
	ALT key: Open a menu from the menu row
	Apps key: Open context menus
	Windows key: Switch to Windows applications
	Clear key: Clear error messages

MillPlus V600, Software and Features

The MillPlus IT is designed for use with milling, drilling, boring and machining centers, as well as for use with mold machines. The MillPlus IT can also be traversed manually for simple machining operations.

Different types of aid are available to the programmer: dialog entry, Function Explorer, context-sensitive online help, graphic simulation, etc.

This Manual describes the programming language of the MillPlus and all G function that are available in MillPlus V600 as of NC software number 538 952-xx.

This Manual may include references to functions that are not yet available in this software version. These references are reserved for later software updates.



Some functions are not yet described completely in this version of the Programming Manual.

Machine configuration

The machine manufacturer adapts the features offered by the MillPlus to the capabilities of the specific machine via configuration data. Some of the functions described in this manual may therefore not be among the features provided by the MillPlus on your machine tool.

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

The machine manufacturer and HEIDENHAIN offer programming courses for the MillPlus. We recommend these courses as an effective way of improving your programming skill and sharing information and ideas with other MillPlus users.

Intended place of operation

The MillPlus V600 complies with the limits for a Class A device in accordance with the specifications in EN 55022, and is intended for use primarily in industrially-zoned areas.

Changes compared with V5xx

During the development of the new MillPlus version V600, care was taken to keep the MillPlus language compatible with earlier versions. With the new developments, however, some new features have been added and some G functions or their sequences have been changed. In this chapter you will find an overview of the most important changes.

New functions

- **G242** Contour advance calculation: ON
- **G251-G269** Contour programming
- **G270-G277** Limit level and zoning level
- **G280-G286** Contour milling cycles
- **High-level language** (If..then, While...)

Functions that are not available anymore

- **G5** Synchronize CNC and PLC (for OEM only).
- **G66** Selection of negative tool direction
- **G67** Selection of positive tool direction
- **G106, G108** Kinematic model
- **G241** Contour monitoring: ON

Functions that are not yet available

- **G8** Correct tool
- **G26** Feed rate and spindle override not effective
- **G52** Activate pallet datum shift
- **G63** Cancel geometry calculation
- **G125** Retract tool in the event of interruption: OFF
- **G126** Retract tool in the event of interruption: ON
- **G148** Query touch probe status
- **G180-G182** Cylinder interpolation
- **G195-G199** Definition of graphics
- **G606-G610** TT: Measure tool
- **G631-G642** Measure workpiece

Modified G functions

Some of the G functions were modified in respect of programming or sequence. For a list of the modifications, refer to Chapter “Changed G-functions” on page 491.

Contents

Introduction	1
Technology	2
Programming	3
Function Explorer	4
G0-G99 G Functions	5
G100-G199 G Functions	6
G200-G299 G Functions	7
G300-G399 G Functions for Macros	8
G600-G699 Measuring Cycles	9
G700-G799 Milling Cycles	10
G800-G899 Turning Cycles	11
G1000-G1999 Macro Functions	12
Modified G Functions	13

1 Introduction 17

- 1.1 Introduction 18
- 1.2 About These Instructions 19

2 Technology 21

- 2.1 F Functions 22
 - Description of the feed rate addresses 22
 - F, F3=, F4= Feed rate and direction of movement 22
 - F1=, Constant cutting feed rate for radius compensation of circles 23
 - F3=, F4= Plunging feed rate/feed rate in a plane 24
 - F5= Feed unit for rotary axes 24
 - F6= Blockwise feed rate 24
- 2.2 S Functions 25
 - Format 25
 - Application 25
- 2.3 M Functions 26
 - M0/M1 Program stop, optional program stop 26
 - M3/M4/M5 spindle ON clockwise/counterclockwise/spindle stop 27
 - M6 Automatic tool change 27
 - M66 Automatic tool change 29
 - M67 Changing the tool data 30
 - M7/M8/M9/M13/M14 Coolant supply on/off 31
 - M19 Oriented spindle stop 32
 - M30 End of part program 33
 - M41/M42/M43/M44 Selecting the spindle speed range 34
- 2.4 T Function Tool Table 35
 - Tool life monitoring 37

3 Programming 39

- 3.1 General Programming Information 40
 - Part programs 40
 - Program words 40
 - Program blocks 42
- 3.2 Creating a Part Program 43
 - Structure of a part program 43
 - Program editor 43
- 3.3 Datums 44
 - Machine datum (M0) 44
 - Pallet datum (M1) 45
 - Workpiece datum (W) 45
 - Program datum (W1) 45
- 3.4 Axis Configurations on Machine Tools 46
 - Axis configurations 46
 - Coordinate system 46
 - Cartesian coordinates 47
 - Polar coordinates 48
 - Mixture of coordinates 49
 - G7 coordinates 50
- 3.5 E Parameters 51
 - Format 51
 - Cancel 51
 - Quantity of parameters 51
 - Address 51
 - Parameter number (E) 51
 - Using a parameter in several programs 52
 - Parameter types 52
 - Input accuracy 52
 - Displaying the parameter table 52
- 3.6 String (ES) Parameters 53
 - Format 53
 - Cancellation 53
 - Quantity of parameters 53
- 3.7 Operators 54
 - Trigonometric functions 59
 - Relational operators 61
 - Logical operators 62
- 3.8 High-Level Language 67
 - Operators 67
 - Like 73
 - Call 78
 - GoTo 79
 - If...Then...Else 80

4 Function Explorer 85

4.1 Milling Functions 86

5 G0-G99 G Codes 93

5.1 G0 Rapid Traverse	94
5.2 G1 Linear Interpolation	97
5.3 G2 Circular CW	101
5.4 G3 Circular Counter-Clockwise	106
5.5 G4 Dwell Time	107
5.6 G7 Tilting Working Plane	108
5.7 G8 Tilting Tool Orientation	117
5.8 G9 Define Pole Position	122
5.9 G11 Linear Chamfer Rounding Cycle	125
5.10 G14 Repeat Function	131
5.11 G17 Main Plane XY, Tool Z	133
Turning	134
5.12 G18 Main Plane XZ, Tool Y	135
Turning	136
5.13 G19 Main Plane YZ, Tool X	137
5.14 G22 Subprogram Call	138
5.15 G23 Program Call	140
5.16 G25 Enable Feed/Speed Override	142
5.17 G26 Disable Feed/Speed Override	143
5.18 G27 Reset Positioning Functions	145
5.19 G28 Positioning Functions	146
5.20 G29 Jump Function	148
5.21 G31 Tapping with Chip Breaking	150
5.22 G37 Milling Operation	153
5.23 G39 Tool Offset Change	154
5.24 G40 Cancel Tool Radius Compensation	157
5.25 G41 Tool Radius Compensation, Left	158
5.26 G42 Tool Radius Compensation, Right	162
5.27 G43 Tool Radius Compensation to End Point	164
5.28 G44 Tool Radius Compensation Past End Point	166
5.29 G45 Measuring a Point	167
Measuring tool dimensions G45 + M25	169
5.30 G46 Measuring a Circle	170
G46 + M26 Calibrating the touch probe	172
5.31 G49 Checking on Tolerances	173
5.32 G50 Processing Measuring Results	175
5.33 G51 Cancel Pallet Zero Point Shift	180
5.34 G52 Activate Pallet Zero Point Shift	181
5.35 G53 Cancel G54-G59 Zero Point Shift	183
5.36 G54 - G59 Activate Zero Point Shift	184

5.37	G61 Tangential Approach	188
5.38	G62 Tangential Exit	191
5.39	G63 Cancel Geometric Calculations	193
5.40	G64 Activate Geometric Calculations	194
	Basic functions	194
	Straight line	196
	Chamfer	200
	Circles	201
	Rounding arcs	203
	Points of intersection	203
	Non-flowing transitions	205
5.41	G70 Inch Programming	208
5.42	G71 Metric Programming	209
5.43	G72 Cancel Mirror Image and Scaling	210
5.44	G73 Mirror Image and Scaling	211
5.45	G74 Absolute Position Approach	213
5.46	G77 Bolt Hole Circle	216
5.47	G78 Point Definition	219
5.48	G79 Cycle Call	221
5.49	G81 Drilling/Centering	223
5.50	G83 Deep-Hole Drilling	225
5.51	G84 Tapping	228
5.52	G85 Reaming	230
5.53	G86 Boring	232
5.54	G87 Pocket Milling	234
5.55	G88 Key-Way Milling	236
5.56	G89 Circular Pocket Milling	238
5.57	G90 Absolute Programming	240
5.58	G91 Incremental Programming	242
5.59	G92 Zero Point Shift Incr./Rotation	244
5.60	G93 Zero Point Shift Abs./Rotation	246
5.61	G94 Feed in mm/min (inch/min)	248
5.62	G95 Feed in mm/rev (inch/rev)	250
5.63	G97 Spindle Speed	251
5.64	G98 Graphic Window Definition	252
5.65	G99 Graphic Material Definition	253

6 G100-G199 G-Codes 255

- 6.1 G125 Lifting Tool on Intervention: OFF 256
- 6.2 G126 Lifting Tool on Intervention: ON 257
- 6.3 G141 3D Tool Correction 261
- 6.4 G145 Linear Measuring Movement 268
- 6.5 G148 Read Measure Probe Status 272
- 6.6 G149 Read Tool- or Zero Offset Values 274
 - Querying tool data 274
 - Querying Zero Offset Values 277
- 6.7 G150 Change Tool- or Zero Offset Values 280
 - Changing of tool data 280
 - Changing Zero Offset Values 282
- 6.8 G151 Cancel G152 283
- 6.9 G152 Limiting the Traverse Ranges 284
- 6.10 G153 Correct Workpiece Zero Point: OFF 286
- 6.11 G154 Correct Workpiece Zero Point: ON 287
- 6.12 G174 Tool Retract Movement 289
- 6.13 G179 ContourCycle Call 291
- 6.14 G180 Cancel Cylinder Interpolation 292
- 6.15 G182 Activate Cylinder Interpolation 294
- 6.16 G195 Graphic Window Definition 297
- 6.17 G196 End Graphic Model Description 298

7 G200-G299 G-Codes 299

7.1 G240 Contour Pre-Calculation: OFF	300
7.2 G242 Contour Pre-Calculation: On	301
7.3 G251-G269 Contour Programming	302
7.4 G251 Free Linear Movement	307
7.5 G252 Free Circular Movement, CW	308
7.6 G253 Free Circular Movement, CCW	310
7.7 G261 Free Linear Movement, Tangential	311
7.8 G262 Free Circular Movement, CW, Tangential	312
7.9 G263 Free Circular Movement, CCW, Tangential	313
7.10 G265 Free Chamfer	314
7.11 G266 Free Rounding	315
7.12 G269 Free Contour Selection	316
7.13 G270 Disables Limit Planes	317
7.14 G271 Enables Defined Limit Planes	318
7.15 G272 Definition of Lower Limit Plane	319
7.16 G273 Definition of Upper Limit Plane	321
7.17 G275 Zoning Planes: Disable	323
7.18 G276 Zoning Planes: Enable	324
7.19 G277 Zoning Planes: Define	325
7.20 G280-G286 Contour Milling Cycles	327
Entering a contour formula	328
Superimposed contours	329
Area of inclusion (joined with)	330
Area of intersection (intersected with)	331
Area of inclusion without intersection (joined with but without intersection)	331
7.21 G280 End Contour Milling	334
7.22 G281 Begin Contour Milling	335
7.23 G282 Contour Definition Program	336
7.24 G283 Contour Data Definition	337
7.25 G284 Contour Pilot Drilling	338
7.26 G285 Contour Roughing	340
7.27 G286 Contour Finishing	342

8 G300-G399 G-Codes for Macros 345

- 8.1 Specific G Codes for Macros 346
 - Overview of G codes for macros 346
 - Overview of G codes for installation purposes 346
- 8.2 G300 Program Error Call 347
- 8.3 G303 M19 with Programmable Direction 348
- 8.4 G305 Synchronize CNC and PLC 349
- 8.5 G319 Read Actual Technology Data 350
- 8.6 G320 Read Actual G Data 351
- 8.7 G321 Read Tool Data 354
- 8.8 G322 Read Machine Constant Memory 356
- 8.9 G323 Read Cycle Data 357
- 8.10 G324 Read G Group 358
- 8.11 G326 Read Actual Position 360
- 8.12 G327 Read Operation Mode 362
- 8.13 G328 Read IPLC Marker or I/O 363
- 8.14 G329 Read Offset from Kinematic Model 365
- 8.15 G331 Write Tool Data 368
- 8.16 G338 Write IPLC Marker or I/O 370
- 8.17 G339 Write Offset in Kinematic Model 371
- 8.18 G380 Protection Zones 373

9 G600-G699 Measuring Cycles 375

- 9.1 Tool Measuring Cycles for Laser Measurements 376
 - General notes and usage 376
 - Availability 376
 - Programming 376
 - Machine parameters 376
- 9.2 Tool Measuring Cycles for Tool Touch Probe Measuring Systems 377
 - General Notes on Tool Touch Probe Measuring Systems 377
- 9.3 Measuring Cycles 378
 - Introduction to measuring cycles 378
- 9.4 G620 Angle Measurement 381
- 9.5 G621 Position Measurement 384
- 9.6 G622 Corner Outside Measurement 386
- 9.7 G623 Corner Inside Measurement 388
- 9.8 G626 Datum Outside Rectangle 390
- 9.9 G627 Datum Inside Rectangle 392
- 9.10 G628 Circle Measurement Outside 394
- 9.11 G629 Circle Measurement Inside 397
- 9.12 G631 Measure Inclined Plane 400
- 9.13 G633 Angle Measurement 2 Holes 402
- 9.14 G634 Measurement Center 4 Holes 404
- 9.15 G636 Circle Measurement Inside (CP) 407
- 9.16 G638 Touch Probe Calibration on Ball 410
- 9.17 G639 Touch Probe Calibration 413

10 G700-G799 Milling Cycles 417

- 10.1 Machining and Positioning Cycles 418
 - Overview of machining and positioning cycles 418
 - Introduction 419
- 10.2 G700 Face Turning 421
- 10.3 G730 Multipass Milling 424
- 10.4 G740 Thread Milling Inside 426
- 10.5 G741 Thread Milling Outside 429
- 10.6 G771 Operation on Line 430
- 10.7 G772 Operation on Quadrangle 432
- 10.8 G773 Operation on Grid 434
- 10.9 G777 Operation on Circle 436
- 10.10 G781 Drilling/Centring 438
- 10.11 G782 Deep-Hole Drilling 440
- 10.12 G783 Deep-Hole Drill. Add Chip Break 443
- 10.13 G784 Tapping 445
- 10.14 G785 Reaming 447
- 10.15 G786 Boring 449
- 10.16 G787 Pocket Milling 451
- 10.17 G788 Key-Way Milling 453
- 10.18 G789 Circular Pocket Milling 456
- 10.19 G790 Back-Boring 458
- 10.20 G794 Tapping, Interpolated 461
- 10.21 G797 Pocket Finishing 463
- 10.22 G798 Key-Way Finishing 465
- 10.23 G799 Circular Pocket Finishing 467

11 G800-G899 Turning Cycles 469

- 11.1 Turning Cycles 470
 - Reserved for turning cycle extensions 470
 - These cycles will appear in a future version. 470

12 G1000-G1099 G-Codes for Macros 471

- 12.1 G1010 Edit Function for SQL tables 472
- 12.2 G1016 Export Formatted Text and E Parameter 476
- 12.3 G1017 Write NC System Data 479
- 12.4 G1018 Read NC System Data 483
- 12.5 G1019 Define up to Two PLC values 486
- 12.6 G1022 Activate Tool Exchange in PLC 487
- 12.7 G1029 Define up to eight PLC values 490

13 Changed G-functions 491

- 13.1 Description of changed G-functions with respect to version V500-V530 492

1

Introduction

1.1 Introduction

Dear customer,

These instructions are intended to support you while programming the MillPlus IT control.

The machine may only be operated—even if it is just temporarily—by properly trained personnel. The training can be provided by the company itself, institutes for advanced vocational training or by one of the training centers.

Please read the notes regarding proper use.

The control is interfaced with the machine via the machine configurations. Some of these configurations can be accessed by the operator. Caution! Before changing any configuration settings, be sure that you understand the meanings and functions thereof. Otherwise please contact the Customer Service.

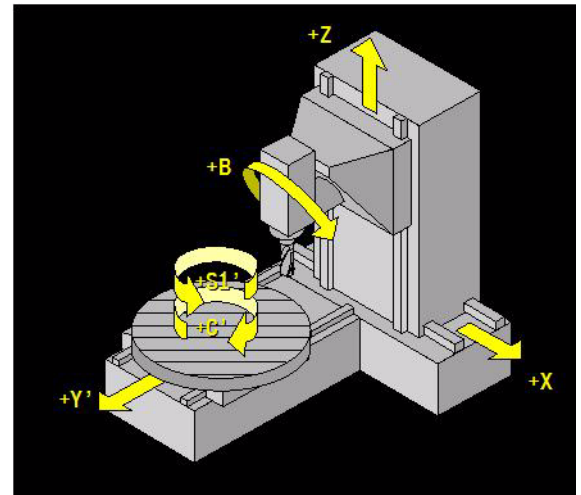
The user should always back up his programs and specific data (e.g. technology data, machine configurations etc.) on a PC or other memory space to prevent data from being lost irretrievably should the system be defective.

We reserve the right to make changes to the design, features and accessories as part of further development. Therefore, no claims may be derived from the data, descriptions or images. Errors and omissions excepted.

© Heidenhain Numeric B.V. Eindhoven, Netherlands 2008

The publisher accepts no liability with regard to specifications, based on the information contained in these instructions. Solely the data contained in the order and the corresponding technical specifications shall apply to the specifications of this numeric control.

All rights reserved. Reproduction of this material, in whole or in part, is not allowed without written permission from the copyright holder.



1.2 About These Instructions

These instructions provide comprehensive information on NC programming.

The core of the reference data contained in these instructions is described in the sections on the G, F, H, M S and T functions. Further information, such as mathematical operations, formulas and high-level language are explained.

G functions

These functions are used to prepare the CNC machine tool for the programming instructions. They are called "preparatory functions". The individual sections dealing with the G functions are structured as follows:

- **G number and brief description** Brief description of the G function and its application.
- **Address description** The address words that define the effective range of the function or the words that can be programmed when the function is active.
 - Address name (e.g. G0)
 - Brief description of address
 - Explanation of the address with a list of the entry options
- **Format** The applicable conventions:
 - Example: G... address..... {address.....}
 - Address..... = mandatory
 - {Address.....} = optional
 - E, F, S, T and M are not entered in the format.
 - Mutual dependencies are not shown.
- **Default** Basic values that are predefined in the CNC.
- **Application** Comments and notes on the use of functions and the circumstances.
- **Sequence** Description of the sequence of the individual steps of a function.
- **Example** Practical examples illustrating the use of a function.

F functions

This function specifies the feed rate types.

H functions

The machine tool builder assigns special tasks to these functions. For information on how to use these help functions, please refer to the respective documentation of the machine tool builder.

M functions

These functions have a direct effect on machine operation, e.g. coolant supply on/off.

S functions

This function specifies the spindle speed in rpm.

T functions

This function specifies the number for tool selection and storage of its dimensions in the CNC tool memory.

2

Technology

2.1 F Functions

Setting the feed rate in mm per minute (mm/min) or revolution (mm/rev). The feed rate actually used in practice is determined by different factors such as the material, machining method and tool.

Description of the feed rate addresses

- **F** Generally applicable feed rate for axis motions with G1/2/3, not for G0
- **F1=** Selecting the constant cutting feed rate for radius compensation of circles
- **F2=** Retraction feed rate with G85, infeed rate with G87-G89 or measuring feed rate with G145
- **F3=** Feed rate for a (negative) infeed movement (plunge)
- **F3=** Feed rate for movements in a plane
- **F5=** Feed unit for rotary axes
- **F6=** Blockwise feed rate.

Type of function

- **Modal:** F, F1=, F3=, F4=, F5=
- **Blockwise:** F2=, F6=

F, F3=, F4= Feed rate and direction of movement

For technological reasons it is necessary to adjust the feed rate carefully to the milling process when executing milling operations. The technological conditions for milling in radial direction are different from the conditions for milling in axial direction, for example.

The user can program modally and independently with two feed rate values, i.e. F3= and F4=.

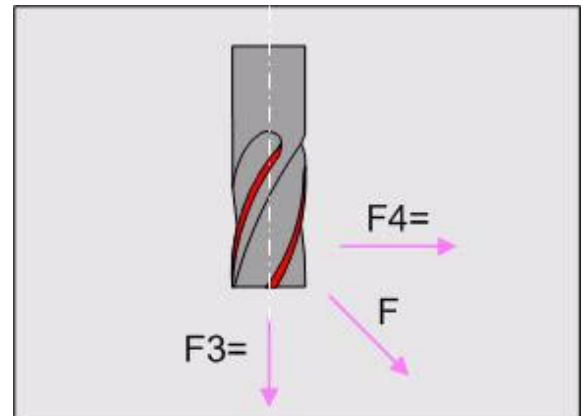
The feed rates F, F3= and F4= are modal.

- 0... 99999 [mm/min] for metric programming
- 0... 9999.9 [inch/min] for programming in inches.

If F, F3= and F4= are programmed in one block, F3= and F4= are of higher priority than F.

Default setting

F3=0, F4=0 and F = 0



Delete

After M30 or by pressing the **Reset CNC** or **Cancel Program** soft key, F F3= and F4= are set to zero.

Maximum feed rate

The maximum feed rate depends on the machine.
Refer to the machine configuration for the maximum feed rate that can be used on the machine tool.

F1=, Constant cutting feed rate for radius compensation of circles

The F1= parameter is used to keep the programmed feed rate constant on the workpiece contour, regardless of the milling radius and shape of the contour. This controlled speed is called constant cutting feed rate.

F1=0

No constant cutting feed rate (start-up condition, M30, **Cancel Program** soft key or **Reset CNC** soft key). The programmed feed rate should reflect the speed of the tool tip. (See Figure 2.)

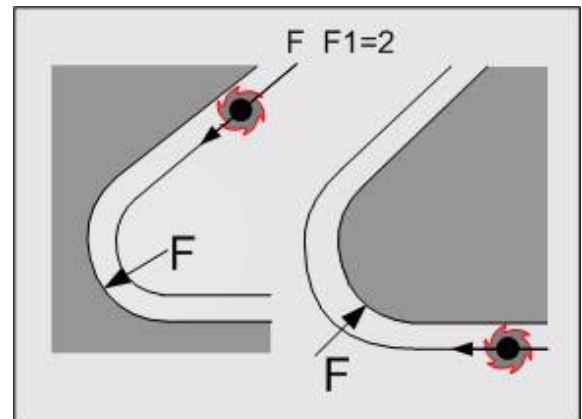
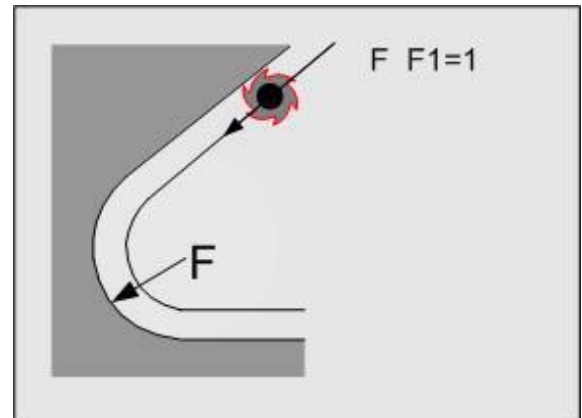
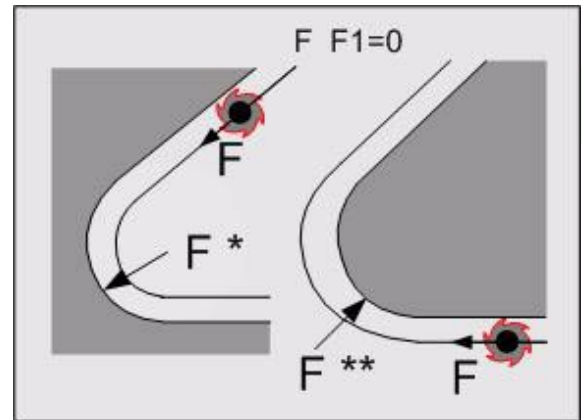
- * Cutting feed rate too high
- ** Cutting feed rate too low

F1=1

Constant cutting feed rate only on the inside of circular arcs. The programmed feed rate is reduced to ensure that the tool tip moves along the inside of a circular arc at the reduced speed. (See figure.)

F1=2

Constant cutting feed rate on the inside and outside of circular arcs. The programmed feed rate is reduced (inside of circular arc) or increased (outside of circular arc) to ensure that the tool tip moves at the recalculated speed. If the increased speed is higher than the maximum feed rate defined via the machine configuration, the maximum feed rate will be used. (See figure.)



F1=3

Constant cutting feed rate only on the outside of circular arcs. The programmed feed rate is increased to ensure that the tool tip moves along the outside of a circular arc at the increased speed. If the increased speed is higher than the maximum feed rate defined via the machine configuration, the maximum feed rate will be used. (See figure.)

F3=, F4= Plunging feed rate/feed rate in a plane

In Cycles G81, G83, G85 and G86, the movement in "axial" direction is not an infeed movement but a feed movement in a plane so that it is programmed with F/F4= and not with the feed rate F3=.

In Cycles G87, G83, and G89, the infeed movement can be programmed blockwise with F2=, for a modally active infeed rate with F3=.

In cycles, F3= is used as infeed rate.

If there is no feed rate or if the feed rate is 0 (e.g. F3= 0 or F4=0 or F 0), an error message is generated. No feed rate programmed.

F5= Feed unit for rotary axes

With G94 F5= you define the unit of the modal feed rate.

G94 F5=0 degree/min (default setting)

G94 F5=1 mm/min or in./min.

With F5=1 the speed on the current rotary axis radius is calculated. This is the distance from the tool to the center of the rotary axis.

Switching off G94 F5=1

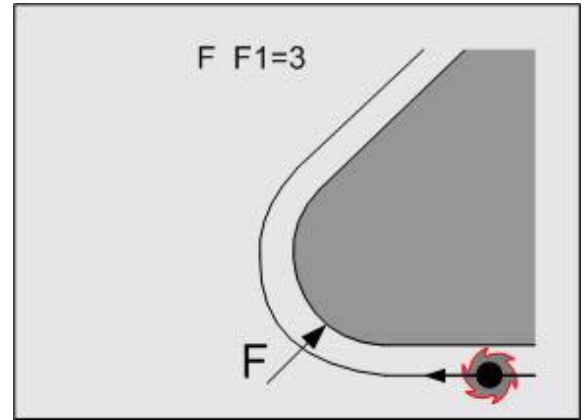
G94 F5=1 is canceled with G94 F5=0, G95, M30, the **Reset CNC** soft key or the **Cancel Program** soft key.

F6= Blockwise feed rate

F6= is a local feed rate and only active in a block in which this feed rate is programmed.

Movement at rapid traverse

F6= in a G0 block limits the movement at rapid traverse.



2.2 S Functions

Setting the speed of the main spindle (S) or the second spindle (or rotary table) (S1=) in revolutions per minute (rpm).

Format

{S...} {S1=...}

Application

Maximum speed

The maximum speeds of the first and second spindle are defined in the machine configuration.

Direction of spindle rotation

For programming the direction of spindle rotation, refer to the description of M3/M4.

Spindle speed ranges

For selecting a spindle speed range, refer to the description of M41/M42/M43/M44.

2.3 M Functions



Most of the available M functions depend on the PLC program. For a description of the M functions that are not, or only partly, described here, refer to the machine documentation.

M0/M1 Program stop, optional program stop

M0 Interruption of program execution

M1 Optional program stop

Application

The stop at M1 is only executed, however, if the **Selective Wait** soft key is active.

Activate

M0/M1 is activated when the current tool movement programmed in the same block has been executed.

Spindle speed and coolant supply

Depending on the PLC program, the spindle speed and coolant supply are also suppressed and/or switched off.

Suppressed means that the spindle and coolant supply are switched on again when the program execution is continued.

M3/M4/M5 spindle ON clockwise/ counterclockwise/spindle stop

M3	Spindle ON clockwise (CW)
M4	Spindle ON counterclockwise (CCW)
M5	Spindle OFF (spindle STOP).

Type of function

M3, M4 and M5 are modal.

Spindle ON (M3/M4)

The spindle is switched on before the tool movement programmed in the same block is executed. The spindle is only switched on when a spindle speed (S) has been programmed.

Spindle OFF (M5)

The spindle command M5 is activated when the current tool movement programmed in the same block has been executed.

The spindle is switched off with M30, the **Cancel Program** soft key or the **Reset CNC** soft key. For oriented spindle stop M19, see M19.

Program stop (M0/M1) or tool change (M6/M66)

Depending on the PLC program, the spindle rotation is suppressed or switched off at a program stop or tool change.

Suppressed means that the spindle is switched on again when the program execution is continued.

M6 Automatic tool change

Executing an automatic tool change



The execution depends on the PLC program. For a description, see the machine documentation.

Format

{T...} {T1=...} {T2=...} M6

T	Tool number
T1=	Switches cutting force monitoring on and off
T2=	Selects additional tool data

Executing the tool change

The tool is changed before the tool movement programmed in the same block is executed.

Tool change procedure

The M6 command leads to the following operating procedure:

- The tool first moves at rapid traverse to the change position.
- The old tool is then replaced by a new one and the compensation data of the new tool are activated.

Machine tool without automatic tool changer

On machines without an automatic tool changer, M6 is executed like M66 (manual tool change).

The M6 command leads to the following operating procedure:

- The tool first moves at rapid traverse to the change position.
- Program execution is interrupted so that the tool can be changed manually.

M6 without tool number T

If no T word is programmed in the M6 block, the tool that was last programmed is inserted and activated. It is recommended that you always program the tool number T for an M6 tool change.

Replacement tools

The tool table contains, for example, tool T100.00 with the replacement tools T100.01 and T100.02.

During an automatic tool change (M6), T100.00 is inserted (T100.00 M6). The replacement tool log is now active. If T100.00 is blocked, a replacement tool is automatically inserted. (T100.01).

During an automatic tool change (M6), T100.01 inserted (T100.01 M6). The replacement tool log is not active now. If T100.01 is blocked, no replacement tool is inserted. An error message is generated.

Note:

If tool T100.01 is measured last during tool measurement, the operator must return the tool to the tool magazine again. If this is not done and T100.00 M6 is programmed in the new program, tool T100.00 will not be changed.

Spindle speed and coolant supply

Depending on the PLC program, the spindle rotation and coolant supply are also suppressed or switched off.

Suppressed means that the spindle and coolant supply are switched on again when the program execution is continued.

Tool change position

It is advisable to reprogram all linear axes in the block after a tool change command. This ensures that Manual Block Search and Restart are always executed in the same way after a program interruption.

Incremental programming after a tool change

With incremental programming the increments refer to the last programmed position. A tool change position is not regarded as a programmed position.

Example: Automatic tool change

T12 M6

M6 Automatic tool change: The new tool is inserted and activated.

M66 Automatic tool change

Interrupt the program execution for a manual tool change.



The execution depends on the PLC program. For a description, see the machine documentation.

Format

{T ...} {T1=...} {T2=...} M66

T Tool number
T1= Switches cutting force monitoring on and off
T2= Selects additional tool data

Using M66

The M66 function is used for tools that are not in the tool magazines.

Executing manual tool changes

The tool is changed before the tool movement programmed in the same block is executed.

Machine tool without automatic tool changer

M66 is used for tools that are not in the tool magazine

A tool may first have to be removed from the spindle (with T0 M6) and returned to its place in the magazine.

It may also be necessary to program a return to a position at which the tool can be inserted.

Manual tool change procedure

The M6 command leads to the following operating procedure:

- The tool first moves at rapid traverse to the change position.
- Program execution is interrupted so that the tool can be changed manually.

M66 interrupts the program execution so that the tool can be changed manually. The execution depends on the PLC program.

When the tool change is completed, the program execution is continued by pressing the start button.

Spindle speed and coolant supply

Depending on the PLC program, the spindle speed and coolant supply are also suppressed or switched off.

Suppressed means that the spindle and coolant supply are switched on again when the program execution is continued.

Example: Automatic tool change

T24 M66

M66 Interruption of program execution and manual tool change. The dimensions of tool T24 are activated.

M67 Changing the tool data

Activating tool data without a tool change.

Format

{T...} {T1=...} {T2=...} M67

- T Tool number
- T1= Switches cutting force monitoring on and off
- T2= Selects additional tool data

Tools with more than one cutting edge

If a tool with more than one cutting edge, e.g. a boring bar, is inserted, each cutting edge has its own length and radius, which are stored as additional compensations for the same tool in the tool table.

Activating the tool data

The new tool dimensions are activated before the tool movement programmed in the same block is executed.

Example: Changing the tool data

The boring bar identified as tool T12 in the illustration has two cutting edges.

- Cutting edge 1 with the tool length XS1 is stored in the tool table as L=XS1,
- cutting edge 2 with the tool length XS2 as L1=XS2.

T12 M6
T12 T2=1 M67

- M6 The boring bar is inserted and the tool data of T12 are activated.
- M67 Changing the tool data from XS1 to XS2. The boring bar is not changed.

M7/M8/M9/M13/M14 Coolant supply on/off

M7	Coolant no. 2 ON, internal coolant supply
M8	Coolant no. 1 ON, external coolant supply
M9	Coolant no. 1 and/or 2 OFF
M13	Coolant no. 1 together with spindle ON clockwise M13=M3+M8
M14	Coolant no. 1 together with spindle ON counterclockwise M14=M4+M8

Format

{M7/M8/M9/M13/M14}

Switch-on (M7/M8/M13/M14)

The coolant supply is switched on before the tool movement programmed in the same block is executed.

Switch-off (M9)

The coolant supply is switched off before the tool movement programmed in the same block is executed.

M30 Cancel program

The coolant supply is switched off with M30, the Cancel Program soft key or **Reset CNC** soft key.

Program stop (M0/M1), spindle stop (M5) or tool change (M6/M66)

Depending on the PLC program, the coolant supply is suppressed or switched off at a program stop, spindle stop or tool change.

Suppressed means that the coolant supply is switched on again when the program execution is continued.

Machine parameters for activation of M13/M14

In order to use the M13 and M14 functions, the CfgPlcMStrobe machine parameter must be set.

Example: Switching the coolant supply on and off

M7
...
M9
...
M13

M7	Coolant no. 2 ON
M9	Coolant OFF
M13	Coolant no. 1 and spindle ON clockwise

M19 Oriented spindle stop

Stopping the spindle at a programmed angular position.

See also G303 M19 D... I2=...

Format

M19 {D...}

Angular position (D)

The angular position is measured from a fixed position that is defined via a machine parameter (CfgReferencing/ref Position).

Spindle speed and direction of rotation

The spindle always moves to the nominal position in a fixed direction that is defined via a machine parameter (CfgReferencing/refDirection).

- + Spindle speed in positive direction of rotation (M3 or CW)
- Spindle speed in negative direction of rotation (M4 or CCW)

Activation

The M19 command is activated when all movements programmed in the same block have been executed.

The spindle position remains unchanged until M3, M4, M13, M14, M41, M42, M43, M44 or M19 is programmed.

Example: Oriented spindle stop

M19 D30

M19 Spindle stops at +30° from the fixed angular position

M30 End of part program

Completing the part program execution with return to the beginning of the program.

Format

M30

Activate

The M30 command is activated when the current tool movement programmed in the same block has been executed.

Spindle rotation and coolant supply

A block with M30 switches off the spindle and the coolant supply.

On-position

When executing M30, the on-position that is applicable for a specific group of G functions becomes effective automatically, if intended.

Other functions with an on-position are reset, as well.

Example: Program structure

N9000
M30

N9000	Program name
	Part program instructions
M30	End of program and return to beginning of program

M41/M42/M43/M44 Selecting the spindle speed range

Format

S M41/M42/M43/M44

Speed range selection

The speed range can be selected automatically by the CNC (the corresponding M function is generated by the CNC) or by programming the appropriate M function (useful with overlapping speed ranges).

Limit values for the speed ranges

The limit values for the speed ranges are stored in the machine parameter memory of the CNC.

Speed range types

Speed ranges can overlap more or less. If no M function for the range selection has been programmed and the spindle speed occurs in two ranges, the maximum range is selected automatically.

Overlapping speed ranges

M41	10 - 250 rpm
M42	200 - 550 rpm
M43	500 - 750 rpm
M44	700 - 1000 rpm

Example:Spindle speed range

M41 S50

M41 The above speed ranges are assumed to be applicable. The required spindle speed is 50 rpm. M41 is programmed so that Range 1 is selected. The automatic range selection is not used.

2.4 T Function Tool Table

Using tools in the program

You can define a tool call in the program with:

Program line	Format	Description
Tnnnn nnnn	[0-9999 9999]	Tool
T*.nn	[00-99]	Explicitly programmed replacement tool
T="ssss....."	40 [char]	Tool name
T1=nnnn	[0-9999]	Cutting force monitoring
T2=n	[0-9]	Activate additional data of an indexed tool
T3=nnnn.n	[0-9999.9 min]	Select tool with sufficient remaining tool life
M6		Automatic tool change
M46		Automatic tool change without safety clearance
M66		Manual tool change
M67		Activate tool data without tool change
G39 L... R...	[mm]	Temporarily activate oversize in length or radius
G50 Tnn		Write the measured tool dimensions into the table
G141 R=... R1=...	[mm]	Temporarily activate oversize in length or radius
[G149 G150] Tnnnn.nn T2=n L1=nn R1=nn M1=nn En		Read and write tool data (length, radius, tool life, status)
G302 Oxx		Temporarily activate the tool orientation (turning)
[G321 G331] Tnnnn.nn T2=n I1=n		Read and write tool data

Tool change

Tool change with index number (T2=)

With the index number IDX you can assign other tool data to a tool. This index number is called in the NC program with T2=.

Example: Tool compensations

```
T1234 T2=3 M6
```

T2= Tool number 1234 is inserted with index number 3

Removing the tool from the spindle

With T0 M6 the tool is removed from the spindle and returned to the magazine.

The tool must be removed from the spindle:

- Before a manual tool change
- For oversized tools.

Tool pre-selection

The next tool can already be pre-selected in the magazine during program run so that it can be inserted immediately with the next tool change. The block includes only the tool number T (without M function).

Replacement tools

A replacement tool is inserted if the tool life of the current machining tool expires or if the lowest performance limit of cutting force monitoring is exceeded.

The replacement tool is identified by the two-digit number after the decimal point.

The replacement tool with the lowest number is selected, unless it is blocked. Otherwise the replacement tool with the next higher number is used.

Tool life monitoring

Each tool is assigned a certain tool life. Whenever a tool is in use, its life is reduced by the cutting time. When the tool life has expired a warning is generated.

Tool change with sufficient tool life (T3=)

A tool can have several replacement tools with different remaining tool lives. Programming T3= selects the tool whose remaining life is sufficient.

If no tool with sufficient remaining life can be found, the MillPlus generates an error message.

Example: Tool life monitoring

T1234 T3=12 M6

T3=12 A program block T3=12 leads to the insertion of tool number 1234, where the remaining tool life has to be at least 12 minutes.

3

Programming

3.1 General Programming Information

Part programs

A part program is the entirety of data and instructions required for the production of a workpiece on a numerically controlled machine tool.

The instructions can contain different operations, e.g. milling, drilling, tapping etc. Each individual operation forms a unit and can be divided into subinstructions. A program block is a complete operation and the words in a program block define the subinstructions.

The correct machining sequence and all subinstructions must be defined in the part program. Examples of subinstructions are tool movements, machine functions and technology data.

A part program can be generated and stored in several ways:

- By interactive contour programming (ICP) for complex contours.
- By manual program entry via the control panel.
- By downloading via a network (e.g. Ethernet or external PC).

Program words

The CNC control uses the standard CNC word address system. A word defined according to this system consists of two parts:

- 1 The address, i.e. an individual address (a letter) or an indexed address. An indexed address consists of a letter followed by an index (number) and the equal sign (=).
- 2 A multi-digit number.

Leading zeroes can be omitted in all words. If, however, the value of one word equals zero, then at least one zero must be written.

Example words:

X-21.43 "X" is the address, "-21.43" the value

X1=-21.43 "X1=" is the address, "-21.43" the value

Format of words with path or angle information

Words that give path or angle information may have an algebraic sign (+ or -). If no algebraic sign has been programmed, a positive value is assumed. Negative values must always have a minus sign.

Path or angle commands can be written with a decimal point. The number of digits after the decimal point depends on the machine configuration: 3 digits (accuracy 1 μm or 1 mdeg.) or 4 digits (accuracy 0.1 μm or 0.1 mdeg.). Any zeroes that follow may therefore be omitted. If no decimal point has been programmed, the CNC assumes that it comes after the last digit.

The total length is always 9 digits. This means that either 123456.789 or 12345.6789 is programmed.

Metric or inches

If G70 is programmed at the beginning of a program, mm is switched to inch. A path command is then programmed 12345.6789 or 1234.56789 (accuracy 0.0001 or 0.00001 inches).

Program blocks

A program block can consist of several words that form a unit, and contain all information required for the execution of a complete operation. This operation may be a tool movement or a machine function or a combination of both.

The control uses a variable block format, i.e. the block length may be different for every block, depending on the difference in the number or length of the words.

If the program block is numbered, the N word always has to come first. The order of the other words is freely selectable. The example shows the recommended order of frequently used words.

Each word can only be used once in a specific block. Words like E1= and E2= have different addresses and can therefore be used in the same block.

Technological and machine data like spindle speed (S), feed rate (F), tool selection (T) and direction of spindle rotation (M3/M4) can also be contained in the block.

Example of a program block

T12 M6
F300 S200 M3
G1 X14 Z62.5

G1	G function Linear Interpolation
X14 and Z62.5	Path information
F	Feed rate
S	Spindle speed
T	Tool number
M	Machine function

Block number N

Block number N is not mandatory in the MillPlus IT as of version V600. A block number is only mandatory if a certain block is to be jumped to. The block numbers range from N0 to N9999999.

It is common practice to use a specific block number only once in the same program. The order of the block numbers is freely selectable. The blocks are executed in the programmed order.

3.2 Creating a Part Program

Program identifier

Each part program or subprogram has its own name. The name consists of letters and/or numbers.

Structure of a part program

To create a part program, the programmer needs the following information:

- The workpiece clamping position and the chucking equipment
- The machining sequence
- The tools that are required for machining
- The applicable technological data for each tool
- The workpiece dimensions and the required movements.

The traverse motions on the machine are a combination of tool and workpiece motions. To simplify programming it is assumed that all motions are made by the tool. The actual sequence of motions depends on the configuration of machine tool and control.

An imaginary coordinate system whose datum is used as the reference point for the programmed motion is placed on the workpiece. The position of this datum is chosen such by the programmer that the programming calculations are reduced to a minimum. For the direction of the coordinate axes, see "Axis Configurations on MachineTools".

Program editor

Several programs can be opened simultaneously in the editor. The maximum number is specified in the settings menu. You can easily switch between the opened part and subprograms and copy data.

The stored programs can be protected against unauthorized editing by using the "Locking" function.

3.3 Datums

In order to determine the machine datum, a reference run must first be performed after switch-on. The machine datum is specified, since the offset data between the machine datum (M0) and the machine-based reference point (R) are saved via machine configuration data.

- R = machine reference point
- M0 = machine datum
- W = program datum

The operator specifies a program datum (W) for the workpiece, relative to which the workpiece dimensions are measured. This program datum must also refer to the machine datum, which can be specified with the G52 and/or G54...G59 functions. (See figure.)

Machine-based reference point (R)

Every traversable axis of a machine tool has a stationary point designated as the reference point. The reference points of all axes together form the machine-based reference point (R) (See figure.)

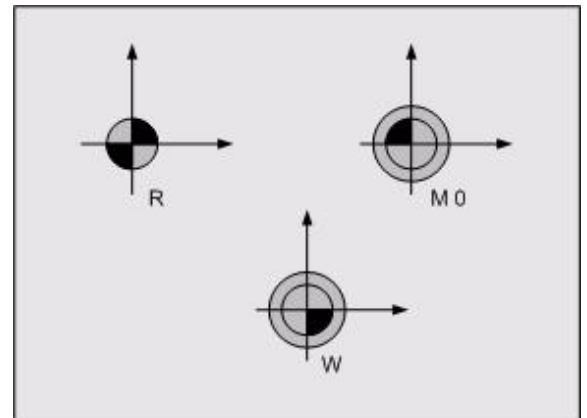
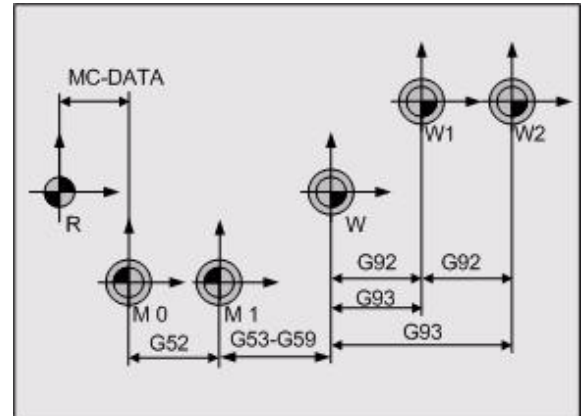
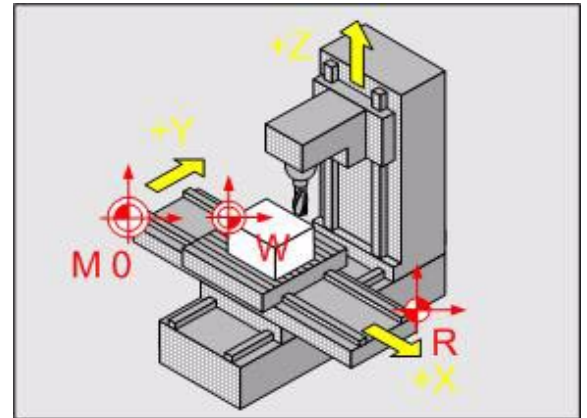
During the reference run (see operating instructions), the tool traverses to the reference point of each respective axis. Once this point is reached, the CNC automatically zeroes the axis. The positions of the software limit switches are specified.

Machine datum (M0)

The machine datum is also a stationary point on the machine.

When the CNC system is put into service, the distances in the axes from the machine-based reference point (R) to the machine datum (M0) are measured and saved as machine parameters. Each axis has its own machine parameter value.

When the machine-based reference point has been defined with the reference run procedure, the CNC reads out the relevant dimensions saved as machine parameters. The machine datum (M0) is defined as the origin of the coordinate system. The displayed positions are referenced to the machine datum.



Pallet datum (M1)

If the machine has several fixtures (e.g. pallets), each fixture has its own datum. This stationary fixture is called pallet datum (M1).

The distances in the axes from the machine datum (M0) to the pallet datums (M1) are stored in the datum shift memory. The functions G52 or G52 I[0...99] permit storage of 99 pallet datums.

Workpiece datum (W)

When a pallet datum (M1) has been defined, the datum of the workpiece must be determined. The workpiece datum can be the same as the active datum M1 or be entered in the table by direct programming or be defined with the F54 function "Preset axes"

If there is an external program call with shift data, the control defines the C datum automatically.

The distances in the axes from the pallet datum (M1) to the workpiece datums (W) are stored in the datum shift memory. The functions G53...G59 or G54 I[0...99] permit storage of up to 99 workpiece datums.

Program datum (W1)

The program datum W1 is the datum from where the axis coordinates in part programs are measured. The programmer can select any position for the W1 datum. It is advisable to select the position such that any additional calculations required for workpiece programming are limited to a minimum.

3.4 Axis Configurations on Machine Tools

Axis configurations

A machine has three linear principal axes (X, Y, Z), which are mutually perpendicular. The orientation of these axes is determined by the Z axis, which always runs parallel to the machine tool's main spindle. The X axis is that with the greatest traverse path perpendicular to the Z axis. A rotary axis and a linear auxiliary axis can be assigned to each principal axis. They are shown in the top figure.

Coordinate system

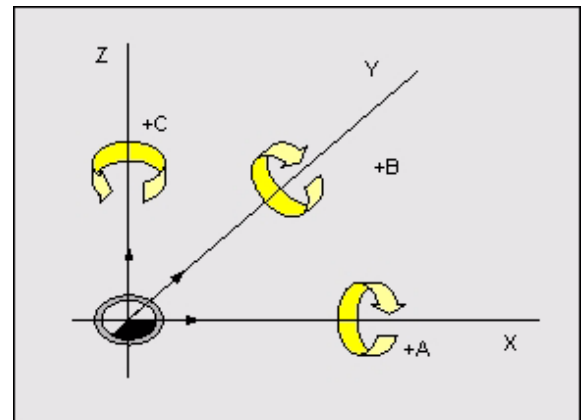
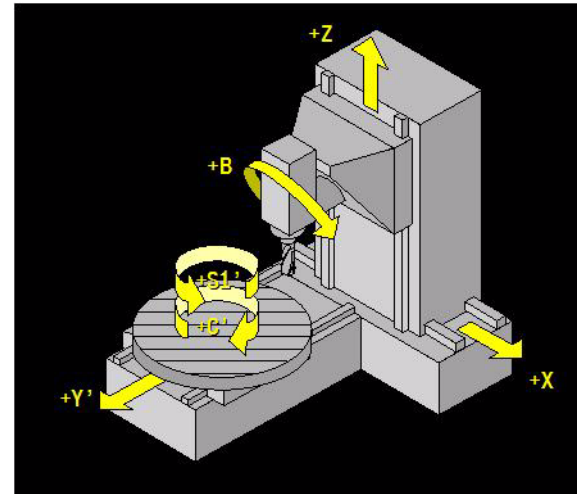
The CNC can connect points by linear and circular paths of traverse (interpolations). Workpiece machining is programmed by entering the coordinates for a succession of points and connecting the points by linear or circular paths of traverse.

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

The positions of the CNC machine tool's axes are defined by the following standards: ISO 841, DIN 66217 and EIA RS-267-A. The right-hand rule defined in these standards is used to indicate the orientation of all axes on CNC machine tools. (Center figure)

Defining coordinates

The coordinates of points in space (3-D) define traverse paths along the axes. The axis coordinates are in one of three planes (XY plane, ZX plane, YZ plane).



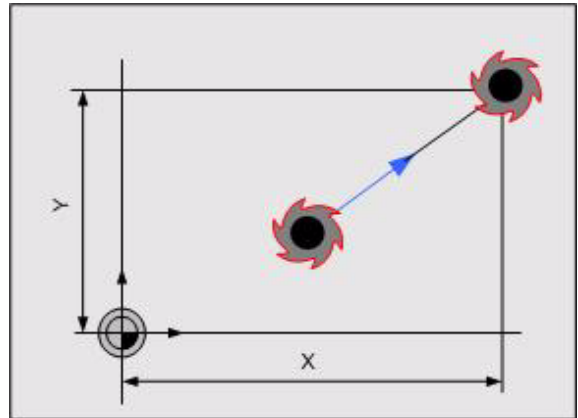
Cartesian coordinates

Absolute coordinates

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

Movement with absolute Cartesian coordinates

G0 X40 Y30 Z30 'X, Y, Z is the distance to the datum



Incremental coordinates

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

Movement with incremental Cartesian coordinates

G0 X91=40 Y91=30 X91. Y91 incremental distances to the current position

or

G91 Incremental programming active

G0 X40 Y30 Z30 X, Y, Z are incremental distances to the current position

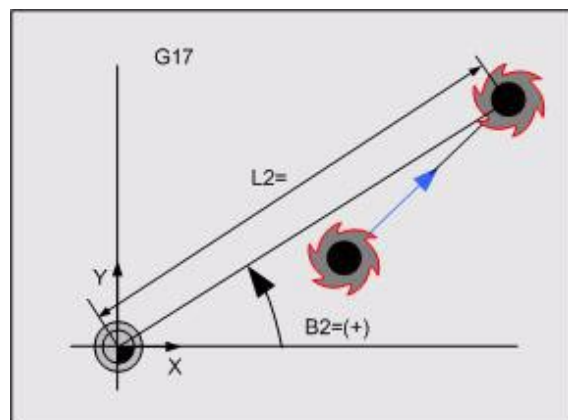
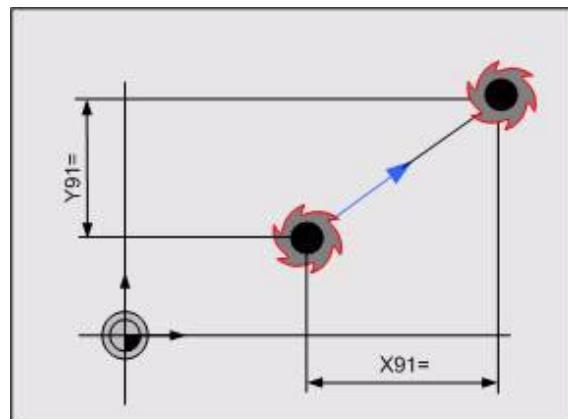
Polar coordinates

When programming with polar coordinates, a position on the workpiece is clearly defined by the entries for polar length and angle.

Movement with absolute polar coordinates

G0 L2=20 B2=45 Z30

- L2= distance to the datum
- B2= angle formed with first principal axis



Movement with incremental polar coordinates

G0 L1=20 B1=45 Z30

- L1= distance between the actual and the nominal position
- B1= angle formed with first principal axis

When programming with incremental polar coordinates, a position on the workpiece is clearly defined by the entries for length and angle.



If a pole has been programmed (see G9), the program blocks with polar programming (angle and length) are no longer referenced to the datum but to the last programmed pole.

Polar coordinates in the XZ plane (G18)
and YZ plane (G19).

Mixture of coordinates

Mixing different coordinates is permitted. Absolute, incremental and polar coordinates are possible.

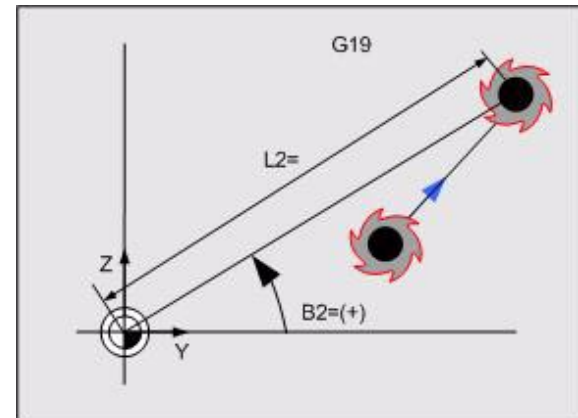
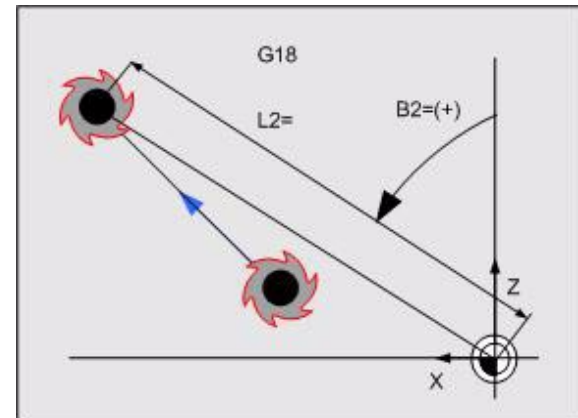
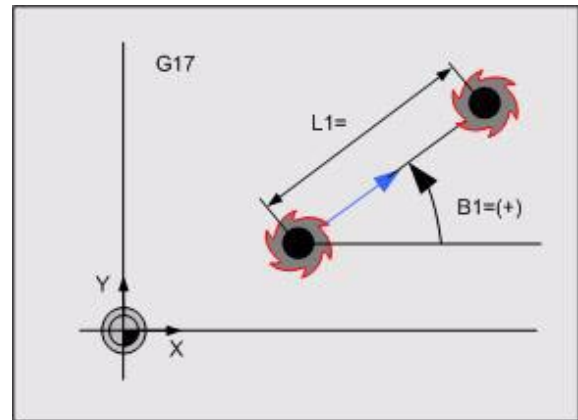
Movement with mixed coordinates

G1 X30 Y91=40 'Absolute and incremental

G1 X30 B1=45 'Absolute and absolute angle

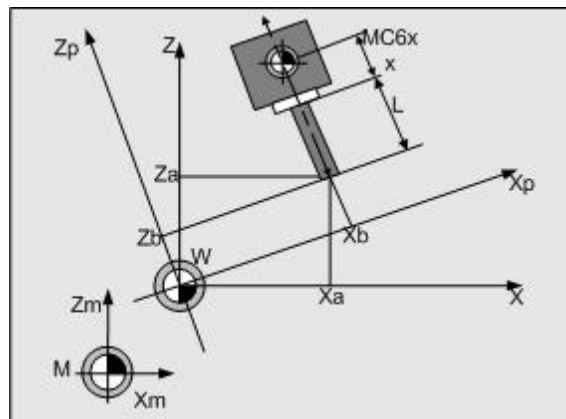
G1 X30 B2=45 'Absolute and incremental angle

G1 X30 L2=45 'Absolute and incremental length



G7 coordinates

The position display on the screen can toggle between the position in the G7 plane (X_p , Z_p) or the machine coordinates (X , Y). Both refer to the active datum $G52 + G54 + G92/G93$.



3.5 E Parameters

E parameters permit a more flexible use of the programs. With a single program you can manufacture different workpieces by changing the parameter data contained in the CNC's parameter memory.

With the help of macros, high-level language and E parameters, a problem can be solved in a general way, e. g. measuring a hole with three or four points. The parameters receive their current values during execution, and the subprogram is adjusted to the special program requirements.

Format

Parameter definition:

- E...=[value or arithmetic expression]

Parameter assignment

- [Address]=(+/-) E...

Parameter assignment and calculation:

- Address = [arithmetic expression]

Cancel

The parameter values are modal, unless they are changed by conversion, entry via the control panel, entry from another data medium or assignment of new values in the part program. By pressing a soft key you can delete a parameter value or the entire table. Pressing the **Cancel Program** soft key, or M30 do not delete parameter values.

Quantity of parameters

You can save a maximum of 1,000 parameters. This quantity can be changed with a machine parameter (numberOfReals). (Default setting: 600 parameters).

For system cycles (PLC and cycles) you can use parameters 1000 to 1400.

Address

Every existing address, except for the address N. Address N generates an error message. Impermissible word: N=.

Parameter number (E)

This number indicates where in the parameter memory of the control the numerical value is stored.

Using a parameter in several programs

A parameter can be used in different programs. If a parameter that was already assigned a value in a previous program is used in a new program, the parameter must be assigned a new value. Otherwise the old value will be used again.

If a parameter for which there is no value contained in the parameter memory is programmed, an error message is generated (parameter not defined).

Parameter types

Parameters can be used in every MillPlus. The following parameter types are possible:

A floating point number consists of a fixed point number (mantissa) multiplied by an exponent. 1.965e5, for example, means $1.965(10^5)$, which equals 196 500.

1	Integral (no decimal point)	E1=20
2	Fixed point number	E1=200.105
3	Floating point number	E1=1.965e5
4	Character sequence	ES1="example.mm"

Input accuracy

The input accuracy of the parameter types is as follows:

1	Integer	A 15-digit number.
2	Fixed point number	At least 6 decimal places, not more than 15 decimal places
3	Floating point number	The mantissa is programmed as a fixed point number, the exponent is an integer between -99 and +99.
4	Character sequence	-

Displaying the parameter table

The parameter values saved in the parameter memory can be listed on the screen.

They are rounded values consisting of several decimal places. They are either displayed as fixed point number or in the so-called scientific notation, i.e. with an exponent.

The fact that the accuracy of the calculated, saved parameter values is equal or greater than the displayed values can cause differences.

Example of a saved value: 99.99999999999999 (more than 16 places) – displayed value is 100

3.6 String (ES) Parameters

The string parameters (or ES parameters) permit a more flexible use of the programs. With a single program you can manufacture different workpieces by changing the parameter data contained in the CNC's parameter memory.

Format

Parameter definition:

- ES...=[string expression]

String expression:

- Character sequence

Parameter assignment

- [Address]=(+/-) ES...

Cancelation

The parameter values are modal, unless they are changed by conversion, entry via the control panel, entry from another data medium or assignment of new values in the part program. By pressing a soft key you can delete a parameter value or the entire table. Pressing the **Cancel Program** soft key or M30 do not delete parameter values.

Quantity of parameters

You can save a maximum of 200 parameters. This quantity can be changed with a machine parameter (numberOfStrings). (Default setting: 200 parameters).

For system cycles (PLC and cycles) you can use up to 400 parameters.

3.7 Operators

Arithmetic operators

Mathematical operators

Description	Operators
Exponential calculation	$E1 = E2 \wedge E3$
Multiplication	$E1 = E2 * E3$
Floating-point division	$E1 = E2 / E3$
Integer division	$E1 = E2 \backslash E3$
Remainder of division	$E1 = E2 \text{ Mod } E3$
Addition	$E1 = E2 + E3$
Subtraction	$E1 = E2 - E3$
Assign	$E1 = E2$

Exponential calculation

$E1 = E2^2$ or $E1 = E2^{E3}$ (with $E3 = 2$)

The two operations have the effect that the $E1$ parameter is equal to the square value of $E2$.

Exponential calculations follow a fixed order. First you do the exponential calculation, then you consider the algebraic sign. For example, in the equation $E1 = -3^2$ you first do the exponential calculation (3^2), then you consider the algebraic sign, which results in a negative number (-9).

If you want to raise a negative number to a power, you have to place it in parentheses, e. g. $E1 = (-3)^{E3}$. Another method is to assign the negative number to a parameter and then raise the parameter value to a power, e. g. $E2 = -3$ and then $E1 = E2^2$.

The following exponential calculations are impermissible:

- 1 0^0 .
- 2 $E2^{E3}$, if $E2 < 0$ and $E3$ have a real value.



In all cases it is permissible to replace the E parameters in parentheses by an arithmetic expression. Example:
 $E1 = \text{Sqrt}(E2^2 + E3^4)$.

Notes

If the relational expression is true, then $E1=1$. If the expression is not true, then $E1=0$. This parameter can be used with the G29 function (conditional jump) or with high-level language.

In the format description, the parameters E2 and E3 are arbitrary parameters or expressions.

Functions and arithmetic expressions can also be used without parameters, e.g. $X=(10+12*\sin(23))$.

The E parameter with the result of the calculation or the mathematical function does not have the required accuracy but can be saved in two different ways.

$E1=99.9999999$ and $E1=100.0000001$ are, for example, equally accurate but differ in their numerical value.

There could be problems if the "Int" function or a relational expression comparing *all* numbers is used.

Translating the calculated numerical values into program words

The parameter values (or the calculated numerical values) are rounded automatically by the CNC and converted into the fixed number of decimal places that is appropriate for the program word.

When programming, for example, $E1=101.74e-3$ and $X=E1$, the number is rounded so that the result is $X0.102$. The numerical value is rounded to three decimal places.



There must be no space between the characters of an arithmetic expression. $E1=E2$ is not permitted, for example. $E1=E2$ is correct. Arithmetic operators must be placed between arithmetic values. $E1=E2 E3$ is not permitted, for example.



Successive arithmetic operators are not permitted. Example: $E1=E2*/E3$. Exception: $E1=E2*-E3$. An expression may contain only one mathematical operation.

Mathematical functions

Description	Operators
Absolute value	$E1 = \text{Abs}(E2)$
Rounding	$E1 = \text{Round}(E2, n)$ (n = decimal places)
Remainder of division	$E1 = E2 \text{ Mod } E3$
Sign	$E1 = \text{Abs}(E2)$
Square root	$E1 = \text{Sqrt}(E2)$
Conversion of integer	$E1 = \text{Int}(E2)$
Pi value (=3.141 592)	Pi
Conversion of high-value integer	$E1 = \text{Ceiling}(E2)$
Conversion of small-value integer	$E1 = \text{Floor}(E2)$
Maximum	$E1 = \text{Max}(E2, E3)$
Minimum	$E1 = \text{Min}(E2, E3)$

Reciprocal values

The reciprocal value of E2 is calculated with $E1 = 1/E2$ or $E1 = E2^{-1}$

Absolute values

With an absolute function, a negative value is converted into a positive value. Positive values remain unchanged. $E1 = \text{Abs}(E2)$.

Square

The square value of E2 is calculated with $E1 = E2 * E2$ or $E1 = E2^2$

Square root

The square root of E2 is calculated with $E1 = \text{Sqrt}(E2)$ or $E1 = E2^{0.5}$

$E1 = \text{Sqrt}(\dots)$: An arithmetical expression in parenthesis is permissible, e.g. $E1 = \text{Sqrt}(E2^2 + E3^4)$.

To extract the square root (Sqrt), the parameter must be positive or zero.

Integer

When the integer function is used, the numerical value is truncated, e.g. all decimal places are ignored. $E1 = \text{Int}(E2)$

Example: $E2=8.9$ is shown as 8, $E2=-8.9$ is shown as -8

Pi constant

The value of the pi constant is saved in the control with an accuracy of 15 digits. Pi can be used at any place at which a value or E parameter is permitted, e.g. for conversion of angles of radians in decimal degrees or vice versa.

Integers with minimum value greater or equal to the argument

When using the function with minimum value, the numerical value is rounded to the biggest argument. $E1 = \text{Ceiling}(E2)$

Example: $E2=8.9$ is shown as 9, $E2=-8.9$ is shown as -8, $E2=8$ is shown as 8

Integers with maximum value smaller or equal to the argument

When using the function with maximum value, the numerical value is rounded to the smallest argument. $E1 = \text{Floor}(E2)$.

Example: $E2=8.9$ is shown as 8, $E2=-8.9$ is shown as -9, $E2=8$ is shown as 8

Rounding

When using the rounding function, the numerical value is rounded based on the number of decimal places. $E1 = \text{Round}(E2, n)$ (n is decimal places)

Note: If the number of decimal places has not been entered, it is automatically assumed to be zero.

Example: $n=1$ and $E2=8.94$ results in 8.9, $n=1$ and $E2=-8.94$ results in -8.9 $n=1$ and $E2=8.96$ results in 9.0, $n=1$ and $E2=-8.96$ results in -9.0

Remainder of division

When using the remainder function, the remainder of the argument is returned.

Note: $E1 = (E2 \text{ Mod } E3)$:

- If $E3$ is 0, $E2$ is returned
- If $E3$ is not entered, 1 is assumed.
- The sign is the same as that of $E1$.

Example: $E2=5$ and $E3=3$ results in 2, $E2=-5$ and $E3=3$ results in -2

Sign

When using the sign function the sign is returned. $E1 = \text{Sign}(E2)$

Example: $E2=8.9$ results in 1, $E2=0$ results in 0, $E2=-8.9$ results in -1

Notes

The E parameter is saved with maximum accuracy. Nevertheless, its value can be entered in different ways.

Example: $E1=99.9999999$ $E3=100.0000001$

$E2=\text{Int}(E1)$ results in $E2=99$, $E2=\text{Int}(E3)$ results in $E2=100$

The E1 and E3 parameters are saved with the same accuracy. In both cases, the value 100 is displayed on the screen. The result of the "Int" function is different, however.

The E parameter with the result of the calculation or the mathematical function does have the required accuracy but can be saved in two different ways.

It is advisable to assign a small value to the parameter whose integer is to be determined, e.g. the required accuracy of the calculations.

Example: If $E1=99.9999999$ or $E1=100.0000001$, the expression $E2=\text{Int}(E1 + 0.0000001)$ yields the value $E2=100$, regardless of the value of E1

Maximum

The Max() function returns the maximum value of the two arguments.
 $E1=\text{Max}(E2,E3)$

Example: $E1=\text{Max}(16,-10)$ results in $E1=16$

Minimum

The Min() function returns the minimum value of the two arguments.
 $E1=\text{Min}(E2,E3)$

Example: $E1=\text{Min}(16,-10)$ results in $E1=-10$

Angle in decimal degrees

An angle is usually programmed in degrees and fractions of degrees. The value can be entered directly in the trigonometric functions, arithmetical expressions or relational expressions.

Example: $E1=\text{Sin}(44.209303)$

Angle in radians

For angle calculations it can sometimes be useful to express angles in radians. 360° corresponds to 2π radians.

Consequently, an angle of 44.209303° equals 0.7715979 radians.

If in a trigonometric function the angle is expressed in radians, the numerical value must be followed by the addition "rad". If in a trigonometric function the angle is expressed in radians, the numerical value must be followed by the addition "rad".

Example: $E1=\text{Sin}(0.7715979\text{rad})$

Trigonometric functions

Description	Operators
Sine	$E1 = \text{Sin}(E2)$
Cosine	$E1 = \text{Cos}(E2)$
Tangent	$E1 = \text{Tan}(E2)$
Arc sine	$E1 = \text{Asin}(E2)$
Arc cosine	$E1 = \text{Acos}(E2)$
Arc tangent	$E1 = \text{Atan}(E2)$
Arc sine	$E1 = \text{Asin2}(E2, E3)$
Arc cosine	$E1 = \text{Acos2}(E2, E3)$
Arc tangent	$E1 = \text{Atan2}(E2, E3)$

Trigonometric functions

The following trigonometric functions are available.
Sine (sin), cosine (cos), tangent (tan)

They are written as follows: $E1 = \text{Sin}(E2)$ $E1 = \text{Cos}(E2)$ $E1 = \text{Tan}(E2)$

The sine of an angle of 44.209303' can, for example, be programmed as follows: $E1 = \text{Sin}(44.209303)$ or $E1 = \text{Sin}(0.7715979\text{rad})$

Notes

The E2 parameter represents any mathematical expression.

Using an odd multiple of 90° in connection with the tan-function is not permissible. Otherwise, an error message is issued.

Using an odd multiple of 90° in connection with the tan-function is not permissible. Otherwise, an error message is issued.

Inverse trigonometric functions

The following inverse trigonometric functions are available: arc sine (asin), arc cosine (acos), arctan (atan)

They are written as follows: $E1 = \text{Asin}(E2)$ $E1 = \text{Acos}(E2)$ $E1 = \text{Atan}(E2)$

Also possible: $E1 = \text{Asin2}(E3, E4)$ $E1 = \text{Acos2}(E3, E4)$ $E1 = \text{Atan2}(E3, E4)$
where $E2 = E3/E4$

Notes

The E2 parameter represents any mathematical expression.

The values of the inverse functions asin and acos should be between -1 and +1; atan can have any numerical value.

The E2 parameter represents any mathematical expression.

The angles created by these functions are expressed in decimal degrees.

The angle created by asin and atan is between -90° and $+90^\circ$.

The angle created by acos is between 0° and 180° .

Remarks:

- For acos and asin, $\text{abs}(E2)$ must be smaller or equal to 1
- The angle created is between 0° and $+360^\circ$.

Relational operators

A relational expression is used to assign the value 1 to the E parameter if certain conditions are fulfilled.

As long as these conditions are not fulfilled, the value of the parameter is 0.

With G29 or high-level language, this parameter enables you to jump within the program.

The following relations can be used:

Description	Type	Operators
Equal to	=	E2=E3
Not equal to	<>	E2<>E3
Greater than	>	E2>E3
Greater than or equal to	>=	E2>=E3
Less than	<	E2<E3
Less than or equal to	<=	E2<=E3

Example: G29 E1=E2>E3 E1 N=400 or If E2>E3 Then GoTo M400

This block means: If parameter E2 is greater than E3, the relation is true and the value 1 is assigned to parameter E1. Parameter E1 is used as a jump condition in the G29 block. If E2>E3, a jump to label M400 is executed.

Notes

Parameters E2 and E3 represent any mathematical expression.

To satisfy a relational expression, all numbers are compared to see if they are equal. Problems may arise if the parameter values are obtained from calculations. In this case, limit values must be set and it must be tested whether the respective value is within the limits. Smaller < E1=E2<E3, smaller or equal <= E1=E2<=E3.



If the relational expression is true, then E1=1. If the expression is false, E1=0. This parameter can be used with the G29 function (conditional jump) or with high-level language.

Logical operators

Logical operators compare Boolean expressions and return a Boolean result.

The following relations can be used:

Description	Type	Operators
Conjunction	And	E2 And E3
Short-circuit conjunction	AndAlso	E2 AndAlso E3
Compares two object reference variables	Is	E2 Is E3
Compares two object reference variables	IsNot	E2 IsNot E3
Compares character string with a sample	Like	E2 Like E3
Negation	Not	Not E2
Disjunction	Or	E2 Or E3
Short-circuit disjunction	OrElse	E2 OrElse E3



In all cases, both of the parameters must be Boolean expressions.

Example: G29 E1=E2 And E3 E1 N=400

This block means: If parameters E2 and E3 are true and E1 is true, the relation is true and the value True is assigned to parameter E1. Parameter E1 is used as a jump condition in the G29 block. With E2 And E3, a jump to N400 is therefore executed.

Sequence of operators in the evaluation

The CNC evaluates mathematical and relational operations in the following order:

Order of priority	Description
1	Calculation of the reciprocal values ($\wedge -1$) and/or exponential calculations (")
2	Multiplication (*) and/or floating-point division (/)
3	Integer division (\)
4	Remainder of division (Mod)
5	Addition (+) and/or subtraction (-)
6	Linking (&)
7	Evaluation of the relational expressions (=, <>, >, >=, <, <=, Like, Is, IsNot)
8	Negation (Not)
9	Conjunction (And, AndAlso)
10	Disjunction (Or, OrElse)

If a block contains operations of the same priority, they are evaluated from the beginning of the block to the end of the block (from the left to the right).

The block $E1=3+7/2-4^2+5*6$ is evaluated in the following order:

- 1 $4^2=16$
- 2 $7/2=3.5$
- 3 $5*6=30$
- 4 $3+3.5=6.5$
- 5 $6.5-16=-9.5$
- 6 $-9.5+30=20.5$

Use of parentheses ()

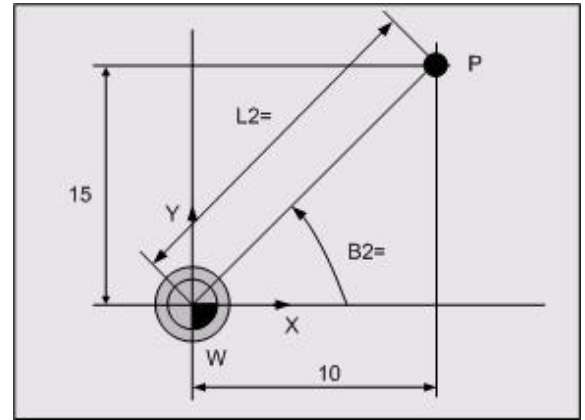
With parentheses () you can group operations and thus change the order of evaluation of an expression. The expression in parentheses is evaluated in the usual order of priority. One pair of parentheses can be placed between another pair of parentheses. This is referred to as "nesting". The evaluation of the expressions between the individual pairs of parentheses is from the inside towards the outside.

Calculating polar coordinates (see figure)

B2=ATAN (15/10) L2=SQRT(10^2+15^2)

B2= For B2= the sequence is:
Calculate 15/10,
determine angle in decimal degrees.

L2= For L2= the sequence is:
Calculate 10^2,
calculate 15^2,
add 10^2 and 15^2
extract the square root.



Calculating the point of intersection of two straight lines (see figure)

Input parameters

E1 First coordinate of the first point on the first straight line.

E2 Second coordinate of the first point on the first straight line.

E3 First coordinate of the second point on the first straight line.

E4 Second coordinate of the second point on the first straight line.

E5 First coordinate of the first point on the second straight line.

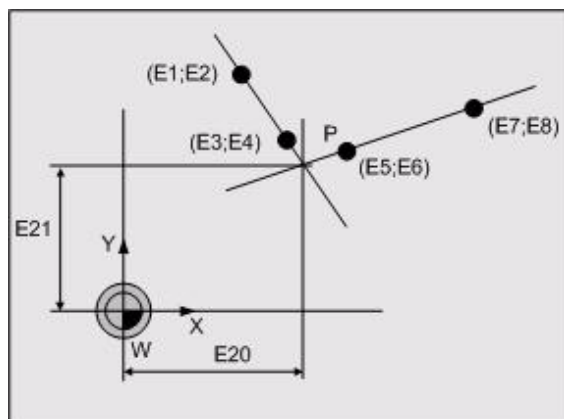
E6 Second coordinate of the first point on the second straight line.

E7 First coordinate of the second point on the second straight line.

E8 Second coordinate of the second point on the second straight line.

Output parameters

E20	First coordinate of the point of intersection.
E21	Second coordinate of the point of intersection.
E79=1	Error found in macro.
E79=0	No error.



N99401 'MACRO FOR CALCULATION OF THE POINT OF INTERSECTION OF TWO STRAIGHT LINES	Macro program
N1 E11=E3-E1 E12=E4-E2 E79=0	Calculation of the unit vector of the first straight line
N2 E13=SQRT(E11^2+E12^2)	
N3 E11=E11/E13 E12=E12/E13	
N4 E13=E7-E5 E14=E8-E6	Calculation of the unit vector of the second straight line
N5 E15=SQRT(E13^2+E14^2)	
N6 E13=E13/E15 E14=E14/E15	
N7 E16=E11*E13+E12*E14	Tests whether the unit vectors run in parallel or not
N8 G29 E15=ABS(E16)<.99995 N=12 E15	
N9 E79=1	If the straight lines run in parallel, parameter E79 is set and there is an error message with program stop. No calculations are carried out after the start.
N10 M0 'STRAIGHT LINES PARALLEL	
N11 G29 E79 N=17	
N12 E15=E1-E7 E16=E2-E8	Calculation of the vector factor
N13 E17=(E15*E12-E16*E11)	
N14 E17=E17/(E13*E12-E11*E14)	
N15 E20=E7+E17*E13	Calculation of the coordinates of the point of intersection
N16 E21=E8+E17*E14	
N17	



Parameter E79 can be used for error processing in the calling program or macro

Example of macro application

- 1 First straight line through the points (30,50) and (60,30)
- 2 Second straight line through the points (100,50) and (50,10)
- 3 The calculation of the point of intersection could be programmed as follows:

	Example of macro application First straight line through the points (30,50) and (60,30) Second straight line through the points (100,50) and (50,10) The calculation of the point of intersection could be programmed as follows:
N100 E1=30 E2=50 E3=60 E4=30	The points on the first straight line
N101 E5=100 E6=50 E7=50 E8=10	The points on the second straight line
N102 G22 N=99401	Calculation of the point of intersection
N103 G29 E79 K0 N=...	If an error is detected, a jump to the block with M30 is carried out.
N104 G0 X=E20 Y=E21	At rapid traverse to the point of intersection

3.8 High-Level Language

Operators

Operators indicate the type of calculation you want to execute on the elements of a formula. If a formula includes more than one operation, they are carried out following a specific order.

You can use parentheses to override the order of priority so that certain parts of an expression are calculated first. Operations in parentheses always have priority over the ones outside the parentheses. Within the parentheses, the normal priority applies.

Operators	Type	Description
&	Linking	Creates one character string out of several character strings.
And	Logical	Performs logical conjunction on two expressions.
AndAlso	Logical	Performs short-circuiting logical conjunction between two expressions.
Is	Logical	Compares two object reference variables.
IsNot	Logical	Performs short-circuiting logical conjunction between two expressions and compares two object reference variables.
Like	Comparison	Compares two character strings
Not	Logical	Performs logical negation on an expression.
Or	Logical	Performs logical disjunction on two expressions.
OrElse	Logical	Performs logical short-circuiting disjunction on two expressions.

&

The linking operator (&) performs logical conjunction of several character strings into one character string.

Syntax

<Result> = <text 1> & <text 2>

Parameters

Text, numbers and string parameters may be used as <Text> expression. They must be programmed as a character string enclosed in quotation marks.

Result:

A character string

Example

ES1="V:\NC_PROG"	
G22 N=ES1 & "\" & "MACRO.MM"	Subprogram call: v:\nc_prog\macro.mm

And

The And operator performs logical conjunction on two expressions.

Syntax

<Result> = <expression 1> And <expression 2>

Parameter

The <expression> is a Boolean expression (True or False)

Result:

True (1)	If both expressions are true
False (0)	If one of the expressions or both of them are false

Example

E1=10	
E2=3	
IF E1<20 AND E2>0 THEN	Result = True: execute instructions after Then
.....	
END IF	

AndAlso

The AndAlso operator performs short-circuiting logical conjunction on two expressions.

Syntax

<Result> = <expression 1> AndAlso <expression 2>

Parameter

The <expression> is a Boolean expression (True or False)

Result:

If the result of <expression 1> is false, <expression 2> is not evaluated. The second expression is not evaluated as it cannot change the final result.

True (1)	If both expressions are true
False (0)	If one of the expressions or both of them are false

Example

E1=10	
E2=3	
IF E1ISNOT NOTHING ANDALSO E2>E1 THEN	If E1 does not have a value, E2>E1 is not evaluated. Result = False: do not execute instructions after Then
.....	
END IF	

Is

The Is operator determines whether two object references refer to the same object.

Syntax

<Result> = <expression 1> Is <expression 2>

Parameter

The <expression> is an object variable.

Result:

True (1)	If both expressions refer to the same object
False (0)	If the two expressions refer to different objects

Example

E1=	
ES2=""	
IF E1 IS NOTHING THEN	Result = True: execute instructions after Then
....	
END IF	

IsNot

The IsNot operator determines whether two object references refer to different objects.

Syntax

<Result> = <expression 1> IsNot <expression 2>

Parameter

The <result> is a Boolean expression (True or False). The <expression> is an object variable.

Result:

True (1)	If the two expressions refer to different objects
False (0)	If both expressions refer to the same object

Example

E1=10	
ES2="MENU"	
IF E1 ISNOT NOTHING THEN	Result = True: execute instructions after Then
.....	
END IF	

Like

The Like operator compares two character strings

Syntax

<Result> = <text 1> Like <text 2>

Parameter

<Text> strings

Text and string parameters (ES) may be used as a <text> expression. They must be programmed as a characters string enclosed in quotation marks.

Result:

True (1)	If the character strings match
False (0)	If the character strings do not match

Example

ES1="WORD"	
ES2="PROGRAM"	
IF ES1 LIKE ES2 THEN	Result=False: program jumps to End If
.....	
.....	
END IF	

Not

The Not operator performs logical negation on an expression. It reverses the value of an argument.

Syntax

<Result > = Not <expression>

Parameter

The <expression> is a Boolean expression (True or False).

Result

True (1)	If the expression is false.
False (0)	If the expression is true.

Example

E2=E3-E5	
IF NOT E2=0 THEN	Result = True: execute instructions after Then
.....	
END IF	

Or

The Or operator performs a logical disjunction between two expressions.

Syntax

<Result> = <expression 1> Or <expression 2>

Parameter

The <expression> is a Boolean expression (True or False).

Result

True (1)	If one of the expressions or both of them are true
False (0)	If both expressions are false

Example

E1=10	
E2=3	
IF E1=10 OR E2>10 THEN	Result = True: execute instructions after Then
....	
END IF	
....	
IF E1=9 OR E2>10 THEN	Result=False: program jumps to End If
....	
END IF	

OrElse

The OrElse operator performs short-circuiting logical disjunction between two expressions.

Syntax

<Result> = <expression 1> OrElse <expression 2>

Parameter

The <expression> is a Boolean expression. If the result of <expression 1> is true, <expression 2> is not evaluated. The second expression is not evaluated as it cannot change the final result.

Result

True (1)	If one of the expressions or both of them are true
False (0)	If both expressions are false

Example

E1=10	
E2=3	
IF E1=10 ORELSE E3=0 THEN	If the result of <expression 1> is true, <expression 2> is not evaluated because it cannot change the final result. Result = True: execute instructions after Then
.....	
END IF	

Instructions

Below is an overview of the available instructions:

Instruction	Description
Call	Executes program run within a program on a subprocedure (Sub).
GoTo	Executes program run without restriction at another point within the program.
If...Then...Else	Executes one or several instruction blocks that are to be executed only if a specific condition is true
Select Case	Executes a command that depends on the result of a specific expression
While...End While	Executes a number of instructions as long as a given condition is true

Call

Executes program run within a program on a subprocedure (Sub). This subprocedure must be in the same DIN program as the Call instruction and be defined with a subinstruction. The subprocedure must start with sub and end with sub. After completion of the subprocedure, program run returns to the block following the Call instruction.

Syntax

```
Call      < name> ()
          .....
Sub       < name> ()
          < instructions >
End       Sub
```

Parameters

<Name>	Name of the subprocedure
<Instructions>	One or more instructions to be executed

The <name> starts with a letter followed by a combination of letters, “_” and numbers.


All subprocedures are written at the end of a program or subprogram. NC blocks that do not follow a subprocedure are not executed. M30 is written before the first sub-procedure.

Example

CALL ENDSEARCH ()	Jumps to subprocedure: EndSearch
.....	
M30	
.....	
SUB ENDSEARCH ()	Beginning of subprocedure: EndSearch
.....	
END SUB	

GoTo

Branches program run without restriction to a defined point ahead in the program. This point must be identified with a label. Furthermore, the label must be at the beginning of a line.



The GoTo instruction is not permitted within While and Case instructions. GoTo is also not permitted within a subprocedure (called with the Call instruction).

Syntax

```
GoTo      <name>
          .....
<Name>:
          <instructions >
```

Parameters

<name>	Name of the label to be jumped to
<Instructions>	One or more instructions

The <name> starts with a letter followed by a combination of letters, “_” and numbers. Only for GoTo can the <name> be a number with 1 to 9 digits.

Example

E1=10	
GOTO ENDSEARCH	Jumps to the point with the EndSearch label
.....	
ENDSEARCH:	
.....	

If...Then...Else

Executes one or more instructions if certain conditions are fulfilled. The instruction consists of If, Then and End If. The Elself and Else instructions are both optional.

If the condition following the If instruction is true, the instruction after Then is executed. Otherwise, program run must skip the instruction and continue immediately with the EndIf instruction. If an Else instruction exists and the If instruction is false , the instruction after the Else instruction must be executed. If Elself instructions have been programmed, each Elself instruction must be evaluated and be executed only if it is true.

Syntax

```
If          < condition > Then
    < instructions >
Elself      < condition > Then
    < instructions >
Else
    < instructions >
End If
```

Parameters

<Condition>	Logical value
<Instructions>	One or more instructions to be executed

The <condition> is a logical value (True or False), consisting of a numerical expression or character string, possibly with logical operators.

Example

E1=10	
IF E1>1 THEN	Result = True: execute instructions after Then
E1=E1-1	
.....	
ELSE	Result = False: Program run continues
E1=10	
.....	
END IF	

Select Case

Executes one or more instructions that depend on the result of a specific test expression. The instruction consists of Select Case, Case and End Select. The Case Else instruction is optional.

If the test expression after Select Case matches a value of the Case instruction, only the instruction following this case is executed. If no matching value is found, the Case Else instruction, if there is one, is executed; otherwise, the instruction immediately after the End Select.

Syntax

```
Select Case  < test expression >
              Case < value >
                                < instructions >
              Case Else
                                < instructions >
End          Select
```

Parameters

<Test expression>	Numerical expression or character string
<Value>	Numerical expression or character string
<Instructions>	One or more instructions to be executed

Example

E1=10	
SELECT CASE E1	Test expression is E1
CASE 1	If E1=1, execute instructions here
....	
CASE 10	If E1=10, execute instructions here
....	
CASE ELSE	If E1 is neither 1 nor 10, execute instructions here
....	
END SELECT	

While...End While

Executes a number of instructions as long as a defined condition is true. The instruction consists of While and End While. The repeat is stopped when the condition after the While instruction is no longer true (False).

Syntax

While < condition >
 < instructions >

End While

Parameters

<Condition>	Logical value
<Instructions>	One or more instructions to be executed

The <condition> is a logical value (True or False), consisting of a numerical expression or character string, possibly with logical operators.

Example

E1=10	
WHILE E1<10	
E1=E1+1	
....	
END WHILE	

Additional Functions

Operators	Description
'	An apostrophe (') marks the beginning of a comment text.
Nothing	The E or ES parameter does not have any value.

'

The ' operator marks the beginning of the user's comment.

Syntax

' Comment text

Example

' COMMENT TEXT	

Nothing

The Nothing operator represents the standard value of any E parameter or ES parameter.

Syntax

<E parameter> = Nothing

<ES parameter> = Nothing

Example

E1=NOTHING	
ES2=NOTHING	

4

Function Explorer

4.1 Milling Functions

Basic functions	Path commands
	Working planes
	Program linkage
	Contouring behavior
	Path compensation
	Zero point shifts
	Geometric functions
	Free-form surfaces/coordinate transformation
	Graphical simulations
	Special functions
Contour programming	Contour programming
Drilling cycles	Drilling cycles
Milling cycles	Milling cycles
Threaded cycles	Threaded cycles
Cycle call	Cycle call
Contour milling cycles	Contour milling cycles
Measuring cycles	Measuring cycles
Tool measurement	
Workpiece measurement	Workpiece measurement
Kinematic measurement	
Flexible NC programming	Operators
	High-level language

Milling Functions

Path commands

G0 Rapid Traverse
G1 Linear Interpolation
G2 Circular CW
G3 Circular Counter-Clockwise
G11 Linear Chamfer Rounding Cycle
G37 Milling Operation
G61 Tangential Approach
G62 Tangential Exit
G74 Absolute Position Approach
G174 Tool Retract Movement

Working planes

G17 Main Plane XY, Tool Z
G18 Main Plane XZ, Tool Y
G19 Main Plane YZ, Tool X

Program linkage

G14 Repeat Function
G22 Subprogram Call
G23 Program Call
G29 Jump Function

Contouring behavior

G4 Dwell Time
G25 Enable Feed/Speed Override
G26 Disable Feed/Speed Override
G27 Reset Positioning Functions
G28 Positioning Functions
G94 Feed in mm/min (inch/min)
G95 Feed in mm/rev (inch/rev)
G97 Spindle Speed
G125 Lifting Tool on Intervention: OFF
G126 Lifting Tool on Intervention: ON

Path compensation

G39 Tool Offset Change
G40 Cancel Tool Radius Compensation
G41 Tool Radius Compensation, Left
G42 Tool Radius Compensation, Right
G43 Tool Radius Compensation to End Point
G44 Tool Radius Compensation Past End Point

Zero point shifts

G51 Cancel Pallet Zero Point Shift
G52 Activate Pallet Zero Point Shift
G53 Cancel G54-G59 Zero Point Shift
G54 - G59 Activate Zero Point Shift
G92 Zero Point Shift Incr./Rotation
G93 Zero Point Shift Abs./Rotation
G153 Correct Workpiece Zero Point: OFF
G154 Correct Workpiece Zero Point: ON

Geometric functions

G9 Define Pole Position
G63 Cancel Geometric Calculations
G64 Activate Geometric Calculations
G70 Inch Programming
G71 Metric Programming
G72 Cancel Mirror Image and Scaling
G73 Mirror Image and Scaling
G78 Point Definition
G90 Absolute Programming
G91 Incremental Programming
G240 Contour Pre-Calculation: OFF
G242 Contour Pre-Calculation: On

Free-form surfaces/coordinate transformation

G7 Tilting Working Plane
G8 Tilting Tool Orientation
G141 3D Tool Correction
G180 Cancel Cylinder Interpolation
G270 Disables Limit Planes
G271 Enables Defined Limit Planes
G272 Definition of Lower Limit Plane
G273 Definition of Upper Limit Plane
G275 Zoning Planes: Disable
G276 Zoning Planes: Enable
G277 Zoning Planes: Define

Graphical simulations

G98 Graphic Window Definition
G99 Graphic Material Definition
G195 Graphic Window Definition
G196 End Graphic Model Description

Special functions

G300 Program Error Call
G303 M19 with Programmable Direction
G305 Synchronize CNC and PLC
G319 Read Actual Technology Data
G320 Read Actual G Data
G321 Read Tool Data
G322 Read Machine Constant Memory
G323 Read Cycle Data
G324 Read G Group
G326 Read Actual Position
G327 Read Operation Mode
G328 Read IPLC Marker or I/O
G331 Write Tool Data
G338 Write IPLC Marker or I/O

Contour programming

G9 Define Pole Position
G251 Free Linear Movement
G252 Free Circular Movement, CW
G253 Free Circular Movement, CCW
G261 Free Linear Movement, Tangential
G262 Free Circular Movement, CW, Tangential
G263 Free Circular Movement, CCW, Tangential
G265 Free Chamfer
G266 Free Rounding
G269 Free Contour Selection

Drilling cycles

G81 Drilling/Centering
G83 Deep-Hole Drilling
G85 Reaming
G86 Boring
G700 Face Turning
G781 Drilling/Centring
G782 Deep-Hole Drilling
G783 Deep-Hole Drill. Add Chip Break
G785 Reaming
G786 Boring
G790 Back-Boring

Milling cycles

- G87 Pocket Milling
- G88 Key-Way Milling
- G89 Circular Pocket Milling
- G730 Multipass Milling
- G787 Pocket Milling
- G788 Key-Way Milling
- G789 Circular Pocket Milling
- G797 Pocket Finishing
- G798 Key-Way Finishing
- G799 Circular Pocket Finishing

Threaded cycles

- G84 Tapping
- G740 Thread Milling Inside
- G741 Thread Milling Outside
- G784 Tapping
- G794 Tapping, Interpolated

Cycle call

- G77 Bolt Hole Circle
- G79 Cycle Call
- G179 ContourCycle Call
- G771 Operation on Line
- G772 Operation on Quadrangle
- G773 Operation on Grid
- G777 Operation on Circle

Contour milling cycles

- G280 End Contour Milling
- G281 Begin Contour Milling
- G282 Contour Definition Program
- G283 Contour Data Definition
- G284 Contour Pilot Drilling
- G285 Contour Roughing
- G286 Contour Finishing

Measuring cycles

- G49 Checking on Tolerances
- G50 Processing Measuring Results
- G148 Read Measure Probe Status
- G149 Read Tool- or Zero Offset Values
- G150 Change Tool- or Zero Offset Values

Workpiece measurement

G45 Measuring a Point
 G46 Measuring a Circle
 G145 Linear Measuring Movement
 G620 Angle Measurement
 G621 Position Measurement
 G622 Corner Outside Measurement
 G623 Corner Inside Measurement
 G626 Datum Outside Rectangle
 G627 Datum Inside Rectangle
 G628 Circle Measurement Outside
 G629 Circle Measurement Inside
 G631 Measure Inclined Plane
 G633 Angle Measurement 2 Holes
 G634 Measurement Center 4 Holes
 G636 Circle Measurement Inside (CP)

Operators

Arithmetic operators
 Mathematical functions
 Trigonometric functions
 Relational operators

High-level language

&
 And
 Like
 Not
 Or
 GoTo
 Call
 If...Then...Else
 Select Case
 While...End While

5

G0-G99 G Codes

5.1 G0 Rapid Traverse

Execution of traverse movements in rapid traverse. G0 is used mainly to position a tool before and after an operation. The sequence of the traverse movements is determined by the positioning logic. All converging axes are interpolated on a linear basis and reach the final position at the same time.

Address description

- ▶ X, Y, Z end point coordinates
- ▶ B, C end point angles
- ▶ B1= angle
- ▶ B2= polar angle
- ▶ L1= path length
- ▶ L2= polar length
- ▶ ?90= end point abs. (X,Y,Z..)
- ▶ ?91= end point incr. (X,Y,Z..)
- ▶ P1= point definition number

Format

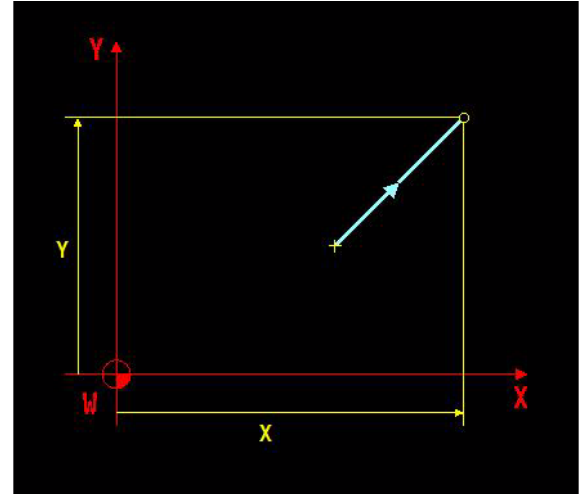
G0 [axis coordinates]



See coordinate systems in the Programming chapter for an explanation of the possible coordinate systems (Cartesian, polar, absolute, and incremental) and definitions.

Default setting

The modal function G0 is automatically effective when the program starts, after CNC reset, after Cancel program, or after executing G37, G77, or G79.



Application

Point definition

A G0 block can contain up to four predefined points (Pn or P1=, P2=, P3=, P4=).

The procedure is determined by:

- The sequence: G0 P10 P1 P7 P11 or
- The point definition: G0 P1=10 P2=1 P4=11 P3=7.

The combination of Pn and P1...4=n is not permitted.

Positioning logic

Every active axis must be programmed in a program block for traverse movements at the start of a program and after a tool or swivel head is replaced. This means that every axis is in the initial position. The positioning logic determines the sequence of the traverse movements in the rapid traverse.

Tool movement towards the workpiece:

	G17	G18	G19
1. Axis movement	A + B	A + B	A + B
2. Axis movement	X + Y	X + Z	Y + Z
3. Axis movement	Z	Y	X

Tool movement away from the workpiece:

	G17	G18	G19
1. Axis movement	Z	Y	X
2. Axis movement	X + Y	X + Z	Y + Z
3. Axis movement	A + B	A + B	A + B

Changes to V5xx

- See "G0..G3_G91" on page 492.

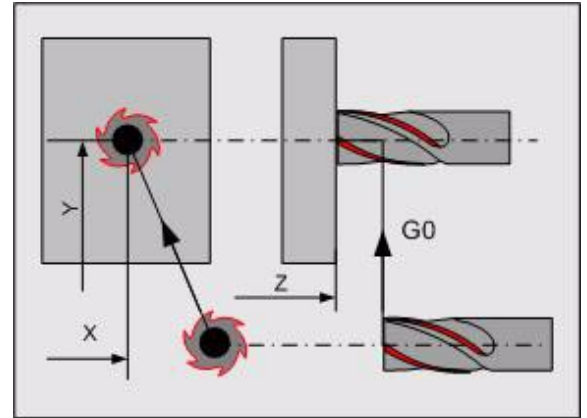
Example

Positioning with rapid traverse

G0 X25 Y15 Z30

G0 P1=80 P2=P1

- G0 Simultaneous movement in the main plane XY, then in the tool axis Z
- G0 The predefined point 80 is approached with rapid traverse followed by point 1.



5.2 G1 Linear Interpolation

Traverse movements are executed on a linear interpolated basis with the specified feed rate.

Address description

- ▶ X, Y, Z end point coordinates
- ▶ B, C end point angles
- ▶ B1= angle
- ▶ B2= polar angle
- ▶ L1= path length
- ▶ L2= polar length
- ▶ ?90= end point abs. (X,Y,Z..)
- ▶ ?91= end point incr. (X,Y,Z..)
- ▶ P1= .. P4= point definition number

Format

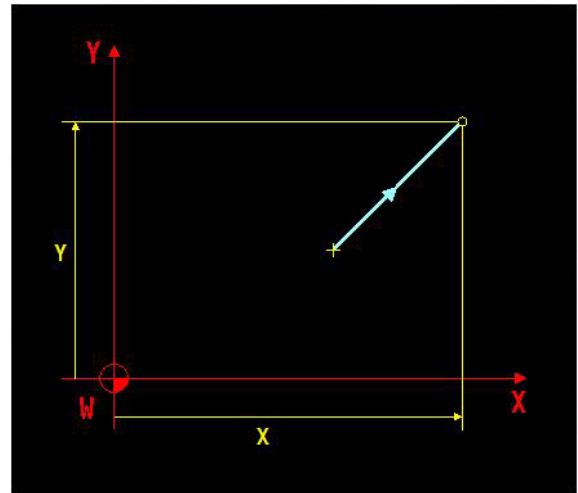
- Linear interpolation in the main plane: G1 {X...} {Y...} {Z...} {F...}
- 3D-interpolation: G1 X... Y... Z... {F...}
- One rotary axis: G1 {A...} {B...} {C...} {F...}
- Movements in more than one axis: G1 {X...} {Y...} {Z...} {A...} {B...} {C...} {F...}



See coordinate systems in the Programming chapter for an explanation of the possible coordinate systems (Cartesian, polar, absolute, and incremental) and definitions.

Default setting

The modal function G1 is deleted by G0, G2, G3, G6, End of program (M30), Cancel program, and CNC reset. G1 is automatically effective after G36 is executed.



Application

Point definition

A G1 block can contain up to four predefined points (Pn or P1=, P2=, P3=, P4=).

The procedure is determined by:

- The sequence: G1 P10 P1 P7 P11 or
- The point definition: G1 P1=10 P2=1 P4=11 P3=7.

The combination of Pn and P1...4=n is not permitted.

Rotary axis and the kinematic model

Every machine is equipped with a kinematic model. This means that the rounding radius between the center point of the rotary axis and the tool is automatically calculated if G94 F5=1 is active.

A40=, B40=, or C40= therefore no longer have to be programmed but are still available for old programs.

Changes to V5xx

- See "G0..G3_G91" on page 492.
- See "G1, G41 und G64" on page 493.

Example

3D interpolation (see figure)

```
G1 X20 Y10 Z40
```

Programming rotary axes with a linear axis (see figure)

```
G0 X10 Y0 Z4 C0
```

```
G1 Z-10 F600
```

```
G94 F5=1
```

```
G1 C360 F1000
```

```
G0 Z20
```

One rotary axis and one linear axis (see figure)

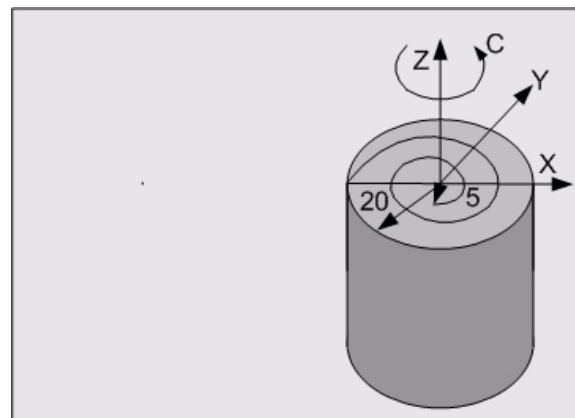
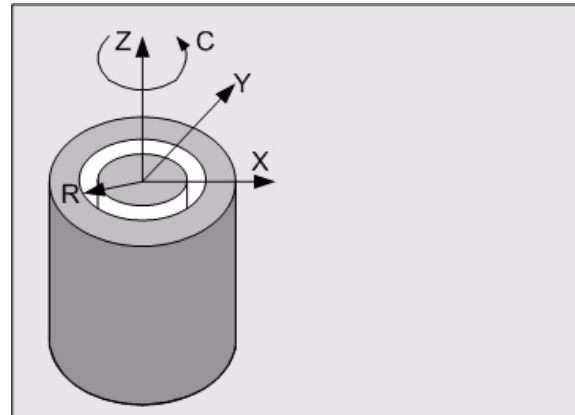
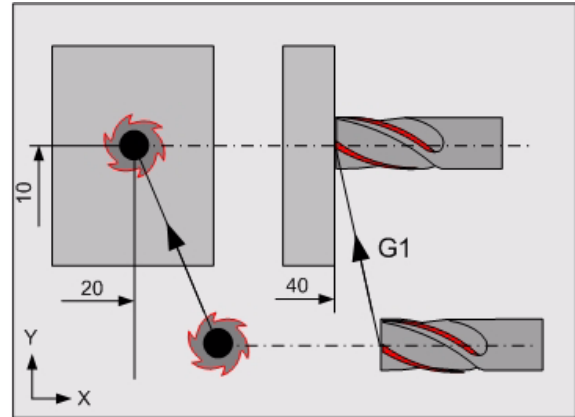
```
G0 X20 Y0 Z4 C0
```

```
G1 Z-10 F600
```

```
G94 F5=1
```

```
G1 X5 C=360+270 F1000
```

```
G0 Z20
```



5.2 G1 Linear Interpolation

Thread on a cylindrical surface (see figure)

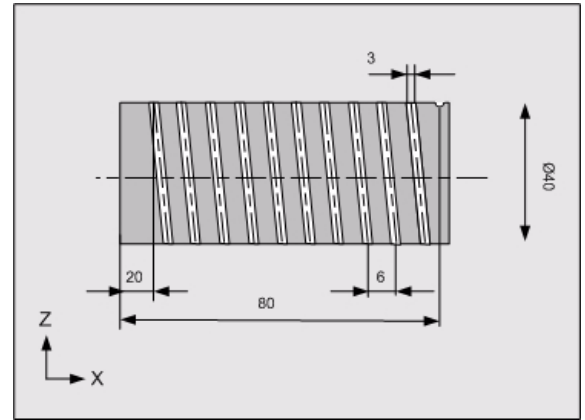
```
G0 X80 Y0 Z22 A0
```

```
G1 Z18 F600
```

```
G94 F5=1
```

```
G1 X20 A=360*10 F1000
```

```
G0 Z30
```



5.3 G2 Circular CW

Execution of a circular, clockwise movement with a programmed feed rate.

Address description

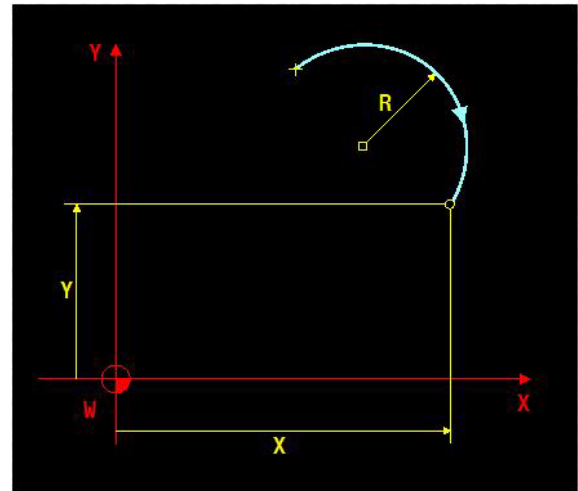
- ▶ X, Y, Z end point coordinates
- ▶ B, C end point angles
- ▶ I center point in X/pitch in X
- ▶ J center point in Y/pitch in Y
- ▶ K center point in Z/pitch in Z
- ▶ R circle radius
- ▶ B1= angle
- ▶ B2= polar angle
- ▶ B3= polar angle for center
- ▶ B5= angle of arc
- ▶ L1= path length
- ▶ L2= polar length
- ▶ L3= polar length for center
- ▶ ?90= absolute center point (X,Y,Z..I,J,K)
- ▶ ?91= incremental center point(X,Y,Z..I,J,K)
- ▶ P1= point definition number

Format

- Full circle: G2/G3 [center point]
- Arc less than or equal to 180: G2/G3 [end point] R...
- More than one arc with the same radius using preprogrammed points, where the arc is less than or equal to 180: G2/G3 P1=... P2=... P3=... P4=... R...
- Arc less than or greater than 180: G2/G3 [center point] [end point] G2/G3 [center point] B5=..
- 2.5D- interpolation G2/G3 [center point] [end point of arc] [end point of linear- or rotary axis].
- Spiral: G2/G3 center point] [end point of arc] [end point of linear- or rotary axis] [pitch] G2/G3 [center point] [pitch] B5=...



See coordinate systems in the Programming chapter for an explanation of the possible coordinate systems (Cartesian, polar, absolute, and incremental) and definitions.



Default setting

The modal function G2 is deleted by G0, G1, G3, G6, End of program (M30), **Cancel program**, and **CNC reset**.

Application

Arc greater than 180°

Center point coordinates

G17 G2/G3 I... J...
G18 G2/G3 I... K...
G19 G2/G3 J... K...

- Absolute center point coordinates (G90): center point coordinates relative to the program zero point (see Figure A)
- Incremental center point coordinates (G91): center point coordinates relative to the starting point (see Figure B)
- Polar center point coordinates: G2/G3 L3=... B3=... (G17/G18/G19) (see Figure C)

End point coordinates

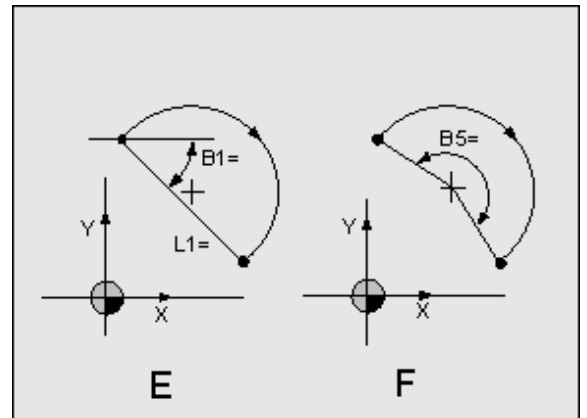
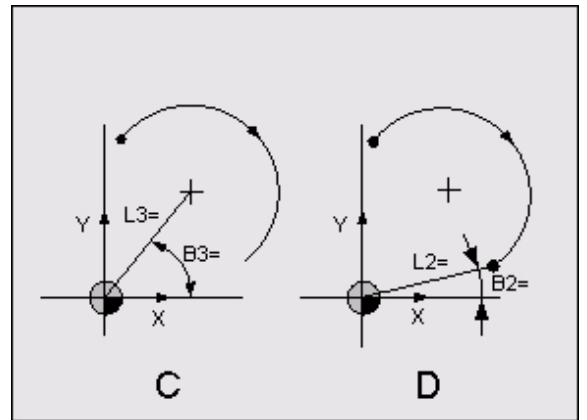
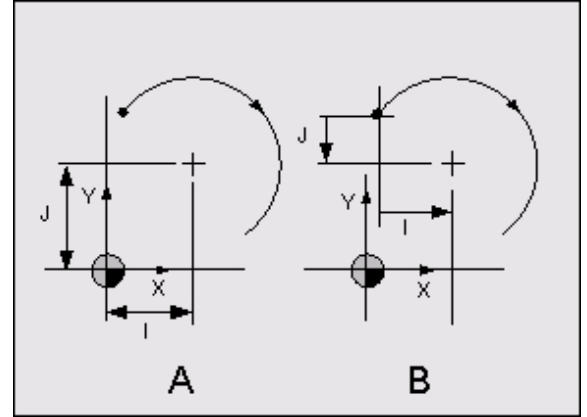
- Cartesian end point coordinates

G17 G2/G3 X...Y...
G18 G2/G3 X... Z...
G19 G2/G3 Y... Z...

- Absolute end point coordinates (G90): end point coordinates relative to the program zero point.
- Incremental end point coordinates (G91): end point coordinates relative to the starting point.

Polar end point coordinates

- End point coordinates relative to the program zero point. G2/G3 L2=... B2=... (G17/G18/G19) (see Figure D)
- End point coordinates relative to the starting point G2/G3 L1=... B1=... (G17/G18/G19) (see Figure E)
- Angle from arc: G2/G3 B5+... (G17/G18/G19). (see Figure F)



Circular movement not in the main plane

Arc to 180°

- G2/G3 [end point coordinates for the linear axes] R..
- G2/G3 [Cartesian coordinates of the circle center point]

Arc greater than 180°

- G2/G3 [Cartesian coordinates of the end point and the circle center point]

Radius compensation cannot be applied. (see figure).

Circular movement with simultaneous movement in the third axis (2.5D)

Circle in the main plane

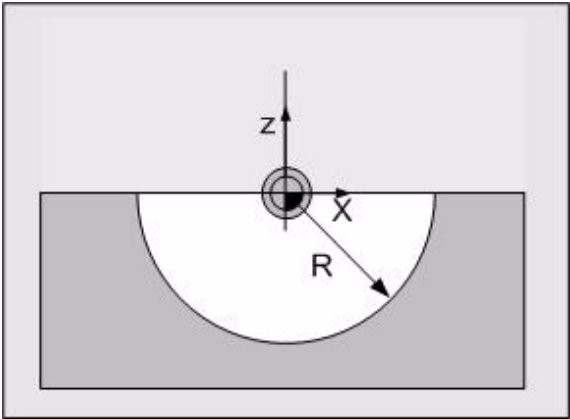
- G2/G3 [circle definition] [tool axis]

Plane	G17	G18	G19
Tool axis	Z	Y	X

Circle not in the main plane

- G2/G3 [Cartesian coordinates of the end point and the circle center point] [tool axis].

Plane	G17	G18	G19
End point	X and Y	X and Z	Y and Z
Center point	I and J	I and K	J and K
Tool axis	Z	Y	X



Spiral interpolation

Plane	G17	G18	G19
Tool axis	Z	Y	X
Center point	I and J B3= and L3=	I and K B3= and L3=	J and K B3= and L3=
Arc angle	B5=	B5=	B5=
Spiral pitch	K	J	I

The value of (B5=) can lie between 0 and 999999 degrees (approx. 2777 revolutions).

Plane	G17	G18	G19
Tool axis	Z	Y	X
Arc end point	X and Y	X and Z	Y and Z
Center point	I and J	I and K	J and K
Spiral pitch	K	J	I

Changes to V5xx

- See "G0..G3_G91" on page 492.
- See "G2" on page 496.

Example

Arc to 180° (see figure)

G1 X40 Y35

G2 X55 Y20 R15

G1 Linear movement

G2 Circular CW

Programming a spiral (see figure)

G0 X40 Y40 Z1.5

G1

G43 Y61

G42

G2 I40 J40 K1.5 B5=4320

G40

G1 Y40

G0

G1

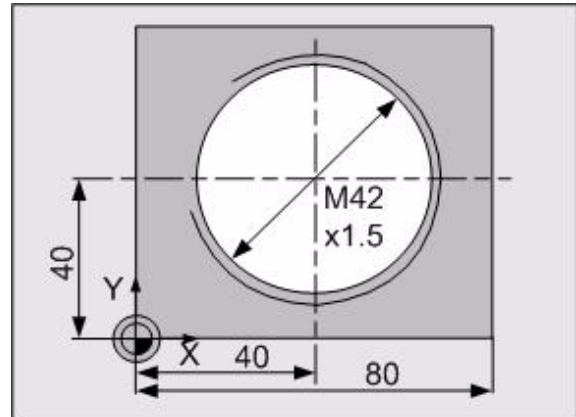
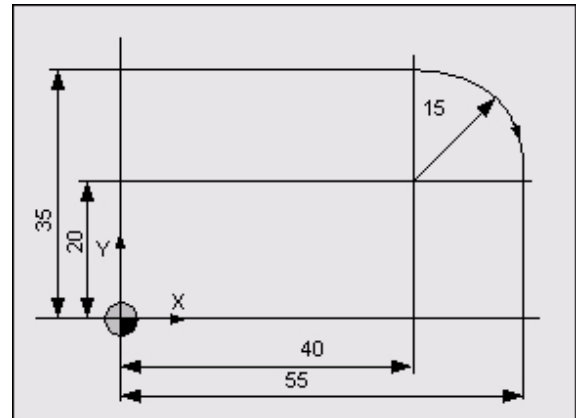
G43 Tool radius compensation to workpiece (G43).

G42 Tool radius compensation right (G42).

G2 Circular CW (thread)

G40 Delete tool radius compensation (G40) .

G1



5.4 G3 Circular Counter-Clockwise

Execution of a circular, counter-clockwise movement with a programmed feed rate.

Address description

- ▶ X, Y, Z end point coordinates
- ▶ B, C end point angles
- ▶ I center point in X/pitch in X
- ▶ J center point in Y/pitch in Y
- ▶ K center point in Z/pitch in Z
- ▶ R circle radius
- ▶ B1= angle
- ▶ B2= polar angle
- ▶ B3= polar angle for center
- ▶ B5= angle of arc
- ▶ L1= path length
- ▶ L2= polar length
- ▶ L3= polar length for center
- ▶ ?90= absolute center point (X,Y,Z..I,J,K)
- ▶ ?91= incremental center point(X,Y,Z..I,J,K)
- ▶ P1= point definition number

Format

See G2

Default setting

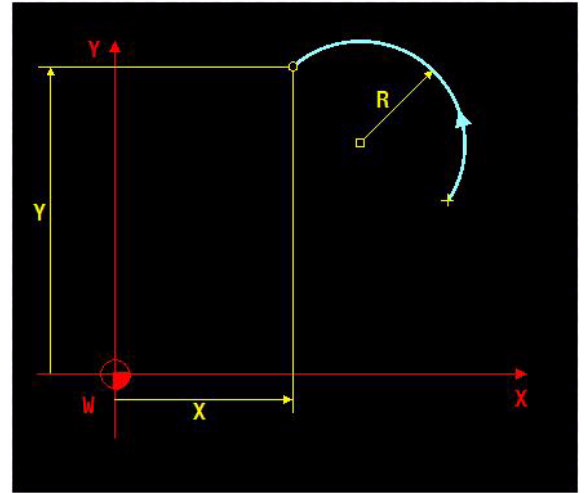
The modal function G3 is deleted by G0, G1, G2, G6, End of program (M30), Cancel program, and CNC reset.

Application

See G2.

Changes to V5xx

- See "G0..G3_G91" on page 492.



5.5 G4 Dwell Time

Insertion of a dwell time (seconds or number of revolutions) in the execution of a program.

Address description

- ▶ X dwell time in sec.
- ▶ D dwell time in revolutions of S
- ▶ D1= dwell time in revolutions of S1

Format

G4 X... or D or D1=

- Minimum dwell time: 0.1 seconds
- Maximum dwell time: 983 seconds (approx. 16 minutes)

Application

Input values

- Dwell time (X) 0.1-983 seconds
- Revolutions (D or D1=) 0–9.9

Example

Dwell time

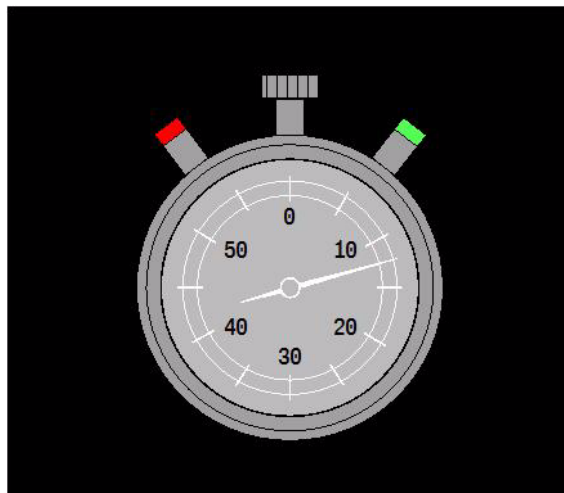
G4 X2.5

G4	Dwell time
X2.5	This block effects a dwell time between two operations of 2.5 seconds.

Revolutions

G4 D2

G4	Dwell time
D2	This block effects a dwell time between two operations that lasts for 2 spindle revolutions.



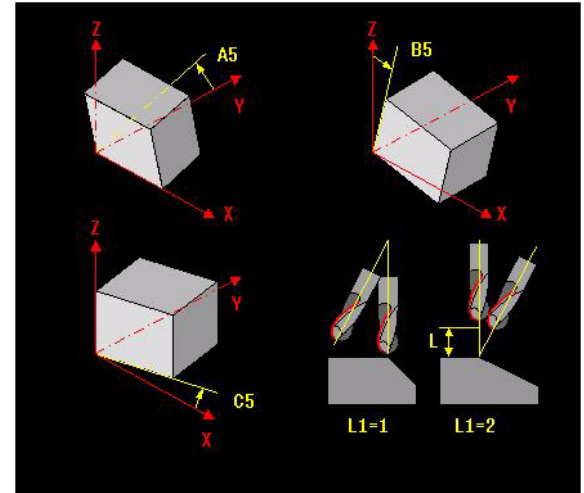
5.6 G7 Tilting Working Plane

The G7 function is used to define and execute the rotation of the working plane for four or five-axis machines.

The machining programmed on the main plane (G17) is then carried out on the tilted working plane. The tool axis orients itself perpendicular to the new plane.

Address description

- ▶ **A5=, B5=, C5= angle of rotation absolute** This is used to define the absolute rotations around the relevant positive axes. The rotations are calculated as follows:
 - The active G7 rotation is canceled
 - C5= Rotation around the machine-based positive Z axis
 - B5= Rotation around the current positive Y axis
 - A5= Rotation around the current positive X axis
- ▶ **A51=, B51=, C51= incremental spatial angle** The new working plane is defined by adding the incremental angle to the active angles. This means that the incremental angles are defined in the machine-based coordinate system.
- ▶ **A7=, B7=, C7= E par. for position in A, B, C** Reading of calculated rotary axis position. Contains the number of an E parameter. The calculated position of the corresponding rotary axis is stored in this E parameter.
- ▶ **L tool length offset** If the tilting movement takes place around the tool tip (L1=2), L defines an allowance in the tool direction between the programmed end point and the tool tip.
- ▶ **L1= 0=no move., 1=rot.axis, 2=tool tip** The G7 tilting movement is carried out on an interpolating basis with rapid traverse. It tilts the tool axis onto the defined plane. Address L1= determines which axes move.
 - **L1=0** The axes do not move (default position). The tilting movement can be carried out in a subsequent block using the E parameters that are loaded with A7=, B7=, or C7=.
 - **L1=1** Only the rotary axes interpolate, the linear axes do not move.
 - **L1=2** The rotary axes and the linear axes interpolate. This means that the tool tip remains in the same position relative to the workpiece.



- **I1= switch off temporarily** The plane currently being tilted can be temporarily suppressed and re-activated without re-programming the spatial angle. **Note:** This function is used via the soft key whereby the tilted plane is temporarily switched off in manual operation.
 - **I1=0:** Tilted plane is activated
 - **I1=1:** Tilted plane is suppressed.
- **B47= E par. for rotation main plane** Reading of main plane. Contains the number of an E parameter. The calculated angle of the main plane is set in this E parameter.

Format

G7 {A5=..} {B5=..} {C5=..} {A51=..} {B51=..} {C51=..} {A7=..} {B7=..} {C7=..} {L1=..} {L..} {L2=..} {I1=...} {B47=..}

Default setting

The modal function G7 is only canceled after programming of G7 only (without angle parameters) or after Advance to reference point or CNC reset. G7 is NOT canceled after Program end (M30) or Cancel program.

Default setting L1=0.



G7 remains active after the control is switched off and on. You can then traverse the G7 plane.

Application

Spatial angles

The programming is independent of the machine configuration. The rotation of the planes is calculated relative to the current zero point. The movement depends on the machine configuration.



A G7 block must contain either absolute or incremental programming.

G codes that are not allowed when G7 is switched on

The following (modal) G codes must not be active if G7 is switched on:

■ G6, G9, G19, G41, G42, G43, G44, G61, G64, G73, G141, G182, G197, G198, G199, G280, G281, G282, G283, G284, G285, G286

G codes that are not allowed within G7

The following G codes are not permitted if G7 is active:

■ G6, G19, G66, G67, G182, G339.

G codes that are not allowed when G7 is switched off

The following (modal) G codes must not be active if G7 is switched off:

■ G9, G41, G42, G43, G44, G61, G64, G73, G141, G197, G198, G199, G280, G281, G282, G283, G284, G285, G286

Switching off the G7 function

The effect of G7 is canceled by programming G7 without angle parameters.



We recommend that you program a G7 without parameters at the start of every program with G7. This means that the plane is always reset when the program is started (cancellation within the tilted plane and restart). Without this G7 at the start, the first part of the program would be executed in the tilted plane instead of the plane that is not tilted.

This programming is similar to the programming with G17/G18 - various zero points or various tools.

M functions that are not allowed when G7 is switched on

The following M functions must not be active if G7 is switched on: M53, M54

M functions that are not allowed within G7

The following M functions are not allowed if G7 is active: M6, M46, M53, M54, M60, M61, M62, M63, M66

Alternative tilting options in the machine's range of traverse

The CNC checks which tilting options are possible in the range of traverse of the rotary axes (to the left or to the right).

- If there is no tilting option, an error message is issued
- If there is only one tilting option, then this is used
- If there are two tilting options, then the one (L2=0 or not programmed) with the shortest traverse path is used. The shortest traverse path is not always possible

The L2= address can be used to control which tilting option has to be used. L2=1/2/3 means that the A/B/C axis is positioned so that it adopts a positive angle. A negative L2= means that a negative angle is adopted.

Rotary axes

The rotary axes can be programmed as normal on the tilted plane. The programmer is responsible for ensuring that the position of the rotary axes conforms with the G7 rotation.

Display

A yellow symbol is shown in the display if G7 is active. A small "p" to the right of the "axis letters" shows whether the position is shown in the tilted machining plane or in machine coordinates.

Changing the tool, pallet, or adapter spindle

No pallet, swivel head, or tool change can be performed if G7 is active. An error is issued and the program must be terminated. G7 must be deactivated before these changes can be made.

Tilting the working plane with M53/M54

The swivel head positioning M53/M54 must be deselected with M55 before programming G7 in mixed operation with G7 and M53/M54. This may involve deselecting active head removal.

Messages

Message	Plane not clearly defined. The G7 plane is defined with a mixture of absolute angles (A5=, B5=, C5=) and incremental angles (A51=, B51=, C51=).
Solution	Only use absolute or incremental angles. If necessary, several G7 definitions with incremental angles can be defined in succession.
Message	Program level not accessible. The defined G7 tilted position is not accessible due to a restricted area of the rotary axes.
Solution	The head can be tilted for certain machine types, which can make the level accessible.

The relevant rotary axis is rotated if the desired rotation of the working plane corresponds to the rotation of this rotary axis. For example, the programming G7 C5=30 on a machine with a (actual) C axis produces a 30° rotation of the C axis.

Changes to V5xx

- See "G7" on page 497.

Procedure

The new plane is activated with the current zero point.

- 1 The tool axis orients itself perpendicular to the new plane. The machine configuration and the programming determine which axes actually move
- 2 The display shows the coordinates on the new (tilted) plane. The manual operation orients itself to the new plane

Example

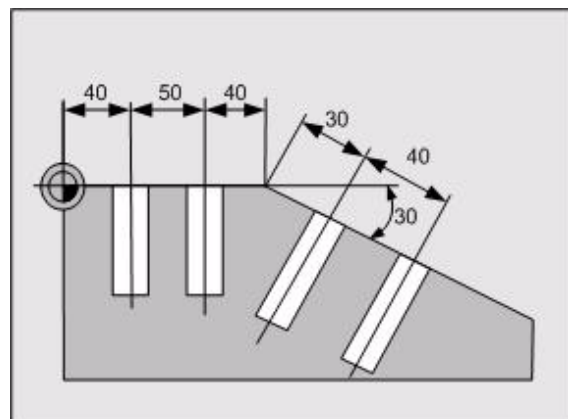
Tool change

G7 B5=45 L1=1
T14
...
G0 Z200
G7 B5=0 L1=1
M6 T14
G0 X... Y... Z...
G7 B5=45 L1=1

G7	Plane is set
T14	Tool preselection
...	
G0	The tool axis is retracted
G7	Deselect G7
M6	Tool change
G0	Rapid traverse to the new starting position
G7	Head is turned back to the G7 plane

Workpiece with tilted working plane

Program example:



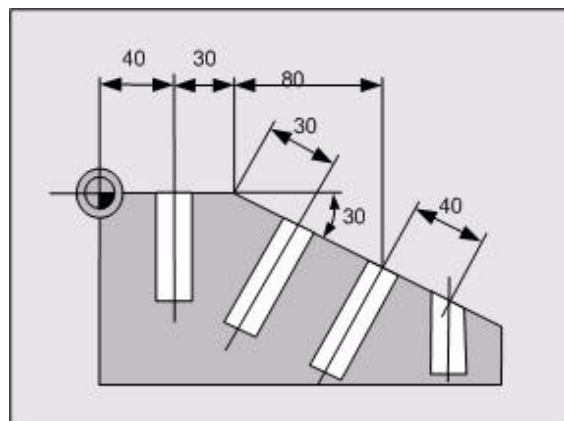
G7 L1=1	Reset G7
G81 Y1 Z-30	Drilling cycle definition
G79 X40 Z0	Drill the first hole in the horizontal plane
G79 X90	Drill the second hole in the horizontal plane
	Other movements in the horizontal plane
G0 X130 Z50	The tool is set to the safety clearance
G93 X130	The zero point is set to the start of the tilted working plane
G7 B5=30 L1=2 L50 OR B5=30 L1=1	G7 Definition of a new working plane B5=30 angle of rotation L1=2 tool/table rotates around the tool tip L50 extra allowance in the tool direction. This means that the tool rotates around the zero point. The distance from the tool tip to the zero point is 50 mm
G79 X30 Z0	Drill the first hole in the tilted working plane
G79 X70	Drill the second hole in the tilted working plane
	Other movements in the tilted working plane
G7 L1=2 L50 OR L1=1	Turn back to the horizontal plane.

Determining the zero point with G7 and G54 I {no.}

Procedure:

- 1 G54 I{no.} can be active, only B4= must be zero
- 2 Pivot G7 with free input (e.g. B5=45 C5=-45 L1=1 (only rotate rotary axes))
- 3 Position manually with the touch probe in the center of a hole.
- 3 Start program N54

Program example:



E1=35	E1=zero point number.
E2=20	E2=hole radius.
G54 I=E1 X0 Y0 Z0 A0 B0 C0 B4=0	Set zero point to zero.
G51	
G53	All zero point shifts are stopped.
G326 X7=50 Y7=51 Z7=52	Read and store the current position of the touch probe. E50= X, E51=Y, E52=Z.
M27	
MEASURE IN G7 PLANE, FIRST MEASUREMENT IN POSITIVE X DIRECTION	
G0 X=E50+(E2-5) Y=E51 Z=E52	To starting position. 5 mm before edge of hole. Collision if E2=<5.
G145 X=E50+(E2+10) Y=E51 Z=E52 L0 X7=49 F2=50 E40 I3=0	X, Y, Z end position, X is edge+10 (verification distance is 10 mm). L0 measure on contact. X7=49 measuring position in E49. F2=50 measurement feed. E40 measuring status in E40. I3=0 status monitoring on.
G29 E41 E40<>1 N=24	Jump to program end if no measuring point is determined.
G0 X=E50-(E2-5) Y=E51 Z=E52	To the starting position of the second measurement in negative X direction.
G145 X=E50-(E2+10) Y=E51 Z=E52 L0 X7=48 F2=50 E40 I3=0	
G29 E41 E40<>1 N=24	Jump to program end if no measuring point is determined.
G0 X=E50 Y=E51+(E2-5) Z=E52	To the starting position of the third measurement in positive Y direction.
G145 X=E50 Y=E51+(E2+10) Z=E52 L0 Y7=47 F2=50 E40 I3=0	
G29 E41 E40<>1 N=24	Jump to program end if no measuring point is determined.
G0 X=E50 Y=E51-(E2-5) Z=E52	To the starting position of the third measurement in negative Y direction.

G145 X=E50 Y=E51-(E2+10) Z=E52 L0 Y7=46 F3=50 E40 I3=0	
G29 E41 E40<>1 N=24	Jump to program end if no measuring point is determined.
MEASURE PERPENDICULAR TO THE G7 PLANE, FIFTH MEASUREMENT IN NEGATIVE Z DIRECTION	
G0 X=E49+5 Y=E51 Z=E52+5	To starting position above material
G145 X=E49+5 Y=E51 Z=E52-10 L0 Z7=45 F2=50 E40 I3=0	
G29 E41 E40<>1 N=24	Jump to program end if no measuring point is determined.
G54 I=E1 X=(E49+E48):2 Y=(E47+E46):2 Z=E45	Set zero point. X, Y, and Z must be entered. These coordinates are converted and then stored in the original machine coordinate system.
G0 X0 Y0 Z0	Center of hole. Display of coordinates, all are zero.
N24 M28	M function for switching off the touch probe.

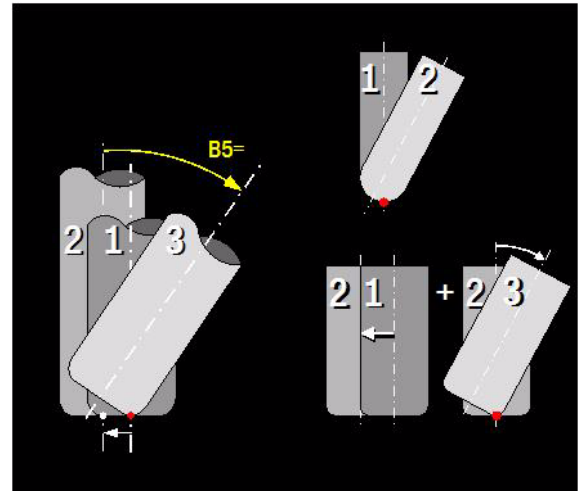
5.7 G8 Tilting Tool Orientation

Programming of a tilted tool for four or five axis machines.

The "Tilt tool" function allows you to tilt the tool direction relative to the working plane. This enables inclined-tool machining and significantly improves the cutting conditions for milling and thus the surface definition.

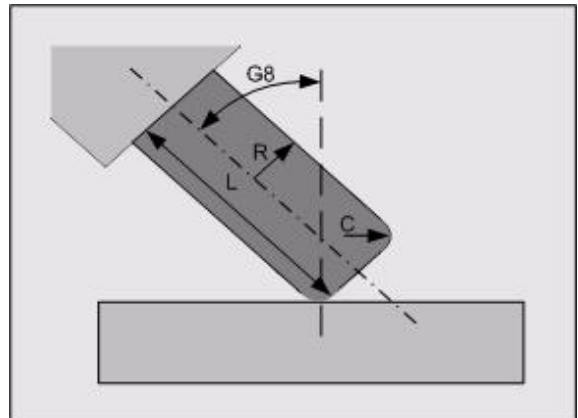
See also G7 Tilting working plane.

L, R, and C from the tool table. (see figure)



Address description

- ▶ **L tool length offset**
- ▶ **A5=, B5=, C5= angle of rotation absolute** Defines the absolute angle by which the working plane rotates around the corresponding positive axis.
- ▶ **A6=, B6=, C6= angle of rotation incremental** Defines the incremental angle by which the working plane rotates around the corresponding positive axis. The value lies between -359.999 and 359.999 [degrees].
- ▶ **B7=, C7= E par. for position in B, C**
- ▶ **L1= 0=no move., 1=rot.axes, 2=tool tip**
- ▶ **L2= -/+1,2,3 = Neg/Pos A,B,C angle**
- ▶ **L3= radius compensation (0=on, 1=off)**
- ▶ **F6= block feed**



Format

G8 {A5=... | A6=...} {B5=... | B6=...} {C5=... | C6=...} {A7=...} {B7=...} {C7=...} {L...} {L1=...} {F6=...}

Default setting

The modal function G8 is only canceled after programming of G8 only (without angle parameters) or after Advance to reference point or CNC reset. G8 is NOT canceled after Program end (M30) or Cancel program.

Default setting L1=0, L3=0.

Application

G codes that are not allowed within G8

The following G codes are not permitted if G8 is active:
G6, G19, G40, G41, G42, G43, G44, G141, G180, G182

Redefining the tool direction

The rotation of the tool direction can be defined in two ways:

Programming with A5=, B5=, or C5= parameters. This means that the absolute rotations are defined around the corresponding positive axes. The rotations are calculated as follows:

- 1 The active G8 rotation is canceled.
- 2 C5= rotation around the machine-based positive Z axis.
- 3 B5= rotation around the positive Y axis.
- 4 A5= rotation around the positive X axis.

Programming with A6=, B6=, or C6= parameters. This means that the incremental rotations are defined around the corresponding positive axes. The rotations are calculated as follows:

- 1 C6= rotation around the current G8 positive Z axis.
- 2 B6= rotation around the current G8 positive Y axis.
- 3 A6= rotation around the current G8 positive X axis.

The programming is independent of the machine configuration. The rotation of the planes is calculated relative to the current zero point. The movement depends on the machine configuration.

Querying a calculated angle position

- **A7=, B7=, C7= E parameter** Contains the number of the E parameter in which the calculated angle for the relevant rotary axis is set.

Alternative tilting options in the machine's range of traverse

The CNC checks which tilting options are possible in the range of traverse of the rotary axes (to the left or to the right).

- If there is no tilting option, an error message is issued.
- If there is only one tilting option, then this is used.
- If there are two tilting options, then the one (L2=0 or not programmed) with the shortest traverse path is used. The shortest traverse path is not always possible.

The L2= address can be used to control which tilting option has to be used. L2=1/2/3 means that the A/B/C axis is positioned so that it adopts a positive angle. A negative L2= means that a negative angle is adopted.

Feed rate

F6= is a local feed rate that is only active in the block in which it was programmed. In this case, this relates to the tilting of the tool. F is the normal feed rate and also applies for the subsequent blocks.

Tilting movement

The G8 tilting movement is performed on an interpolating basis with feed rate (F6=). It tilts the tool axis onto the defined plane. Which axes move depends on the movement type L1=:

- L1=0 The axes do not move (default setting)
- L1=1 Only the rotary axes tilt, the linear axes do not move
- L1=2 The rotary axes tilt and the linear axes execute a "compensating movement". This means that the contact point position remains X, Y, Z

Note: The tilting movement can be programmed or carried out manually using the E parameters that are loaded with A7=, B7= or C7=. The axes do not move.

The movement is just a rotation if the contact point lies on the tool corner radius.

If the contact point is the tool tip and the corner radius (C) is smaller than the tool radius (R), then a compensatory movement is carried out so that the contact point moves from the tool tip to the corner radius.

In the case of cylinder milling (with corner radius $C < \text{milling radius } R$), the following anomaly applies: During tilting from the perpendicular (1) to a tilted position (2 → 3) or vice versa, the contact point moves from the milling center to the corner radius and vice versa. A compensatory movement at the tool tip ensures that the current contact position X,Y,Z remains unchanged.



The movements when starting/canceling the tool compensation within G8 can result in a risk of collisions. The programmer (user) is responsible for avoiding this.

Tool length allowance (L)

If the tilting movement takes place around the tool contact point (L1=2), L defines an extra allowance in the tool direction between the center of rotation and the tool tip.

Tool radius compensation (L3=)

The values L, R, and C are corrected for the tool during the "Tilt tool" function (G8).

This G8 tool compensation is independent of G40, G41, G42, G43, G44 and is always effective.

If the corner radius (C) is less than the tool radius (R), then a compensatory movement is carried out at the beginning and end of the tool compensation, so that the contact point moves from the tool tip to the corner radius.

The current position of the linear axes is recalculated if the tool dimensions (L, R, C) change during active G8.

Tool compensation

The values L, R, and C are corrected for the tool during the "Tilt tool" function (G8), depending on the tool radius compensation (L3=). This G8 tool compensation is independent of G40, G41, G42, G43, G44 and is always effective. If the corner radius (C) is less than the tool radius (R), then a compensatory movement is carried out at the beginning and end of the tool compensation, so that the contact point moves from the tool tip to the corner radius. The current position of the linear axes is recalculated if the tool dimensions (L, R, C) change during active G8.

Switching off the G8 function

The effect of G8 is canceled by programming G8 without angle parameters.



We recommend that you program a G8 without parameters at the start of every program with G8. This means that the tool direction is always reset when the program is started (cancellation for a tilted tool and restart). Without this G8 at the start, the first part of the program would be executed in the tilted plane instead of the plane that is not tilted.

The programming is similar to the programming with G7/G17/G18 - various zero points or various tools.

Graphic

G8 has no influence on the graphic.

Display

A yellow symbol is shown after the tool number in the display if G8 is active. A small "p" to the right of the "axis letters" indicates whether the position of the tool tip is shown or the position in machine coordinates.

Changes to V5xx

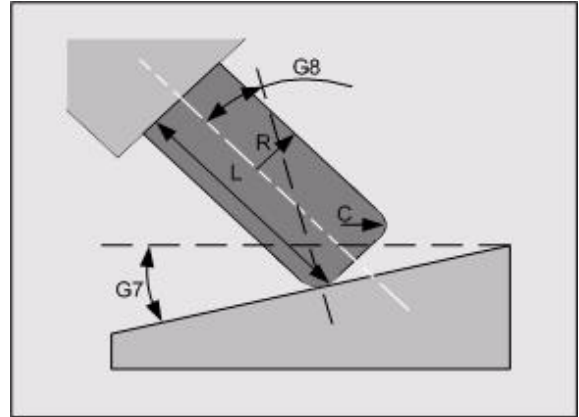
■ See "G8" on page 499.

Example

Workpiece with tilted working plane and tilted tool

The regular hexagon should be milled on the outside of the workpiece surface. One-point-geometry with an angle is used. Sides 2 and 4 are programmed as chamfers (see figure).

Program example:



G7 L1=1	Reset G7
G8 L1=1	Reset G8
G0 X130 Z50	The tool is set to the safety clearance.
G93 X130	The zero point is set to the start of the tilted working plane.
G7 B5=-30 L1=2	Define a new working plane. B5=-30 angle of rotation L1=2 tool/table rotates around the tool tip.
G8 B5=30 L1=2	Define a new tilted position for the tool. B5=30 angle of rotation L1=2 tool rotates around the tool tip and a compensatory movement is carried out
	Other movements in the horizontal plane
G8	Rotate the tool so it is perpendicular to the working plane again (rotary and compensatory movement).
G7 L1=2	Turn back to the horizontal plane.

5.8 G9 Define Pole Position

Programming a pole. If a pole was programmed, the program blocks with polar programming (angle and length) now relate to the last programmed pole instead of to the zero point.

The pole is programmed depending on the modal valid measuring system G90/G91. It can also be programmed word-oriented on an absolute, incremental, or mixed absolute/incremental basis.

Address description

- ▶ X, Y, Z pole coordinates
- ▶ B1= angle
- ▶ B2= polar angle
- ▶ L1= path length
- ▶ L2= polar length
- ▶ ?90= pole coordinate abs. (X,Y,Z..)
- ▶ ?91= pole coordinate incr. (X,Y,Z..)

Format

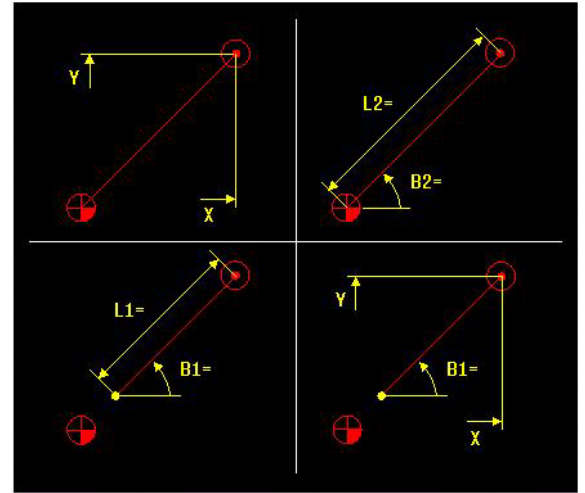
- G17 active: G9 X... Y... {X90=...} {X91=...} {Y90=...} {Y91=...}
- G18 active: G9 X... Z... {X90=...} {X91=...} {Z90=...} {Z91=...}
- G19 active: G9 Y... Z... {Y90=...} {Y91=...} {Z90=...} {Z91=...}
- Deactivate pole (equal to workpiece zero point) G9 X0 Y0
- Pole in polar coordinates (G17, G18, G19 active):
Absolute: G9 B2=... L2=...
Incremental: G9 B1=... L1=...



See coordinate systems in the Programming chapter for an explanation of the possible coordinate systems (Cartesian, polar, absolute, and incremental) and definitions.



G9 "Define pole position" is also used by the contour programming G codes G251-G269.



Application

Pole definitions

Pole definitions are only allowed in the active working plane. The pole is set to 0 (zero) when the plane is changed with G17, G18, G19. The pole is at the workpiece zero point before the G9 block is called. (Pole = 0). The pole is effective modally.

Deactivating

This function is modal and is deactivated by programming a pole with coordinates (0.0).

Defining the end point on a polar basis

In the case of absolute polar programming, the pole lengths $L1=$ or $L2=$ and polar angles $B1=$ or $B1=$ relate to the pole instead of the zero point.

If no pole was defined, the pole=0 (zero) and is thus equal to the active zero point.

Polar point definitions

Polar point definitions with poles are possible in the following G codes: G0, G1, G40, G44, G61, G62, G77, G78, G79, G145.

Polar circle definitions

The center and end point in G2 and G3 blocks can be programmed on a polar basis with a pole.

Pole in absolute coordinates (see figure)

The programmed coordinates relate to the workpiece zero point.

■ B= pole

G9 X... Y...

Pole in incremental coordinates (see figure)

The programmed coordinates relate to the actual position:

■ A= existing pole

■ B= new pole

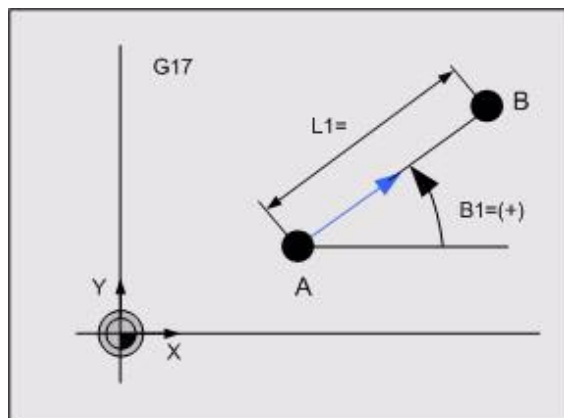
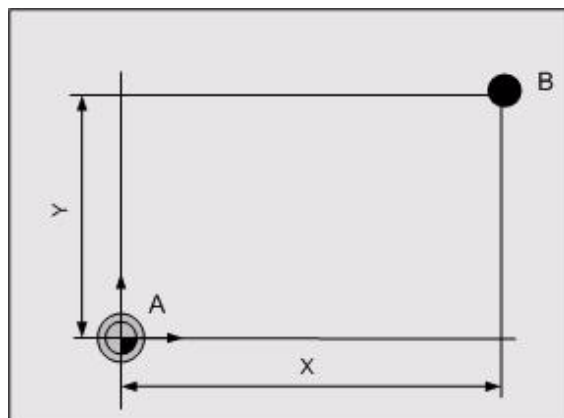
G9 B1=... L1=...

Mixture of coordinates

A mixture of different coordinates is also allowed. Absolute, incremental, and polar are possible.

Changes to V5xx

■ See "G9_B2" on page 499.



Example

Possible movements with different pole coordinates

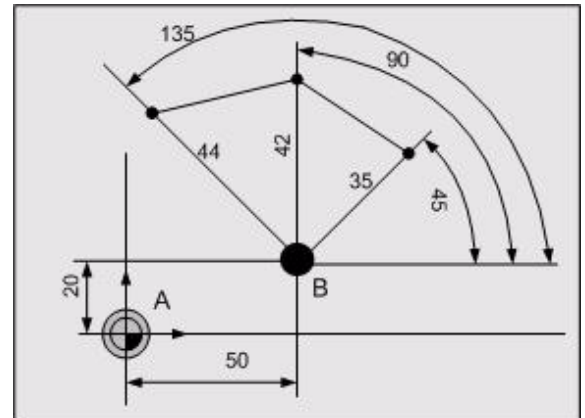
G9 X30 Y40
 G9 X91 Y91=40
 G9 X30 Y91=40
 G9 L2=30 B2=45
 G9 L1=30 B1=45
 G9 L2=30 B1=45
 G9 X30 B1=45
 G9 X30 B2=45
 G9 X30 L1=45

- G9 Absolute Cartesian coordinates
- G9 Incremental Cartesian coordinates
- G9 Mixed absolute and incremental Cartesian coordinates.
- G9 Absolute polar coordinates.
- G9 Incremental polar coordinates.
- G9 Mixed absolute and incremental polar coordinates.
- G9 Absolute and incremental angle.
- G9 Absolute and absolute angle.
- G9 Absolute and incremental length.

Polar programming (see figure)

G9 X48 Y39
 G1 B2=135 L2=44
 G1 B2=90 L2=42
 G1 B2=45 L2=35

- G9 Definition of a new pole
A = new pole
- G1 Definition of the end point coordinates, relative to the new pole
- G1 Definition of the end point coordinates, relative to the new pole
- G1 Definition of the end point coordinates, relative to the new pole



5.9 G11 Linear Chamfer Rounding Cycle

Programming in a block of one or two linear movements with chamfer or rounding. The application of the function is restricted to existing programs. Please do not use it for new programs.

Address description

- ▶ X, Y, Z end point coordinates
- ▶ B first angle
- ▶ K first chamfer length
- ▶ L first length
- ▶ R first rounding radius
- ▶ X1=, Y1= end point of second element
- ▶ B1= second angle
- ▶ K1= second chamfer length
- ▶ L1= second length
- ▶ P1= point definition number
- ▶ R1= second rounding radius

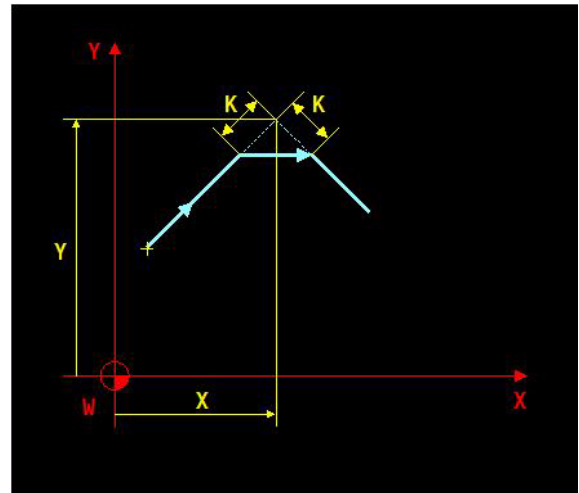
Format

- One-point-geometry(XY-plane)
 - G11** X... Y... {K...} {R...}
 - G11** B... L... {K...} {R...}
- Two-point-geometry(XY-plane)
 - G11** X... Y... X1=... Y1=... {K...} {R...} {K1=...} {R1=...}
 - G11** B... L... X1=... Y1=... {K...} {R...} {K1=...} {R1=...}
 - G11** X... Y... B1=... L1=... {K...} {R...} {K1=...} {R1=...}
 - G11** B... L... B1=... L1=... {K...} {R...} {K1=...} {R1=...}
- Two-line-geometry(XY-plane)
 - G11** B... X... Y... B1=... {K...} {R...} {K1=...} {R1=...}

The angles B and B1= must be programmed in the correct direction.



See coordinate systems in the Programming chapter for an explanation of the possible coordinate systems (Cartesian, polar, absolute, and incremental) and definitions.



Basic functions

One-point geometry

Programming in a block (see figure)

- Of the end point (P1) of a linear movement.
- Of a symmetrical chamfer (K) or rounding (R) between this and the next linear movement (if necessary).

Two-point geometry

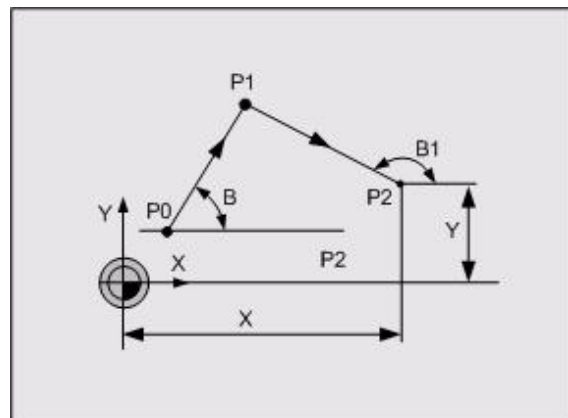
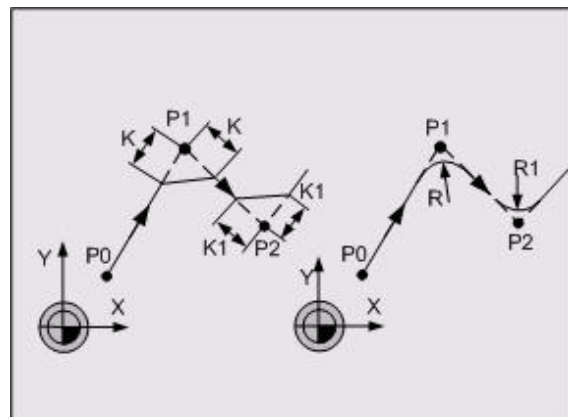
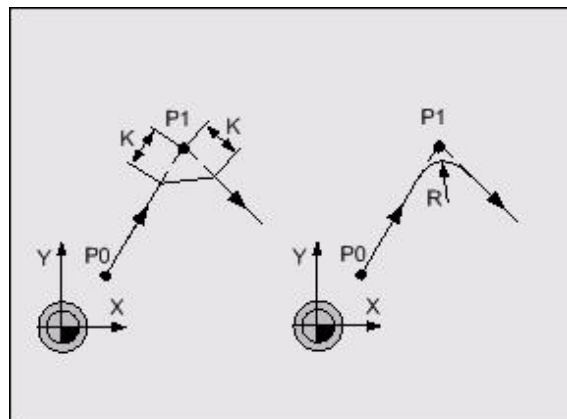
Programming in a block (see figure)

- Of the end points (P1 and P2) of two separate linear movements.
- Of a symmetrical chamfer (K) or rounding (R) between these movements (if necessary).
- Of a symmetrical chamfer (K1=) or rounding (R1=) between the last and the next linear movement (if necessary).

Two-line geometry

Programming in a block of two separate linear movements: (see figure)

- The first linear movement with the angle with the principal axis.
- The second linear movement with the end point and the angle with the principal axis.
- A symmetrical chamfer or rounding between these movements (if necessary).
- A symmetrical chamfer or rounding between the last and the next linear movement (if necessary).



Application

Feed rate

All traverse movements programmed in a G11-block are carried out with the same feed rate.

Single block

Every contour train is carried out separately in a single block.

Incremental next block

If the next block is programmed on an incremental basis, then this block is incremental from the end point of the last chamfer or rounding.

The movement immediately after a G11-block

If a second chamfer (K1=) or a second rounding (R1=) is programmed, then the block immediately after the G11-block must contain either the function G1 or G11. If a G11-block is programmed immediately after the G11-block, then both end point coordinates (e.g. X... and Y...) must be specified.

Limitation

- 1 The G11- function must not be used for geometry calculations (G64 active).
- 2 The G11- function must not be used to define a pocket or island contour for the universal pocket cycle (G200 to G208).
- 3 A tool axis must not be programmed for G11.

Changes to V5xx

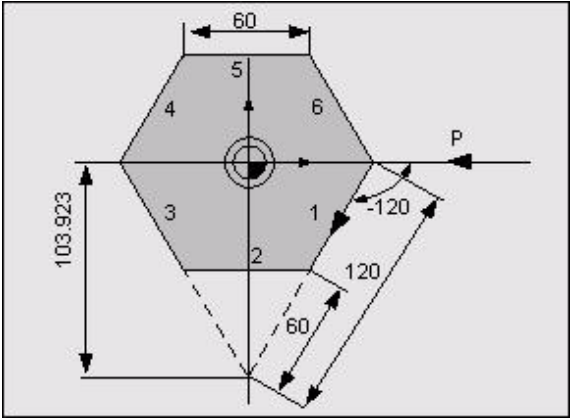
- See "G11" on page 499.

Example

One-point geometry

The regular hexagon should be milled on the outside of the workpiece surface. One-point-geometry with an angle is used. Sides 2 and 4 are programmed as chamfers.

Program example:



N9010	
G17 T1 M6	Activate main plane. Insert tool
G0 X100 Y10 Z-10 S1000 M3	Switch spindle on. Move the tool to point P and then to the working depth
G1 F300	Define the feed rate as 300 mm/min
G43 X60	Move the tool to the corner of the hexagon
G41 Y0	Call the radius compensation LEFT
G11 B-90 L103.923 K60	Sides 1 and 2 are milled The following have been programmed: - The point of intersection for sides 1 and 3 - The chamfer (K expression) around this point
G11 B150 L103.923 K60	Sides 3 and 4 are milled The following have been programmed: - The point of intersection for sides 3 and 5 - The chamfer (K expression) around this point
G11 B60 L60	Side 5 is milled
G11 B0 L60	Side 6 is milled
G40	Delete radius compensation
G1 X100 Y10	Move the tool away from the workpiece
G0 Z100 M30	Tool retraction and end of program

Two-point geometry

The regular hexagon should be milled on the outside of the workpiece surface. Two-point-geometry with angles and increments is used. Sides 2 and 5 are programmed as chamfers. (Same figure as for example 1).

Program example:

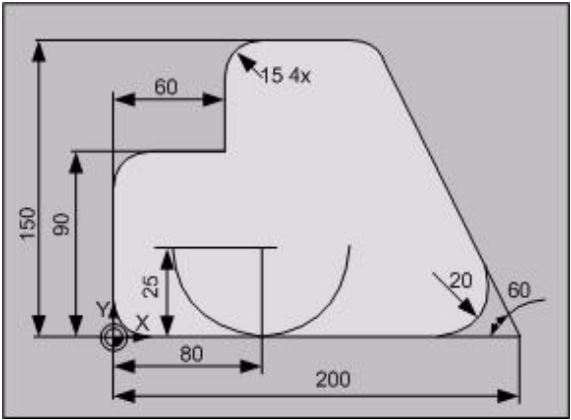
N9011	
G17 T1 M6	Activate main plane. Insert tool
G0 X100 Y10 Z-10 S1000 M3	Switch spindle on. Move the tool to point P and then to the working depth.
G1 F300	Enter the linear movement and specify the feed rate.
G43 X60	Move the tool to the corner of the hexagon,
G41 Y0	Call the radius compensation LEFT.
G91	Switch to incremental measurement programming. The length values in the next blocks are measured from the previous tool position.
G11 B-120 L120 K60 B1=-120 L1=120	Sides 1, 2, and 3 are milled. The following have been programmed: - The point of intersection for sides 1 and 3 (B and L) - The chamfer (K-expression) around this point - The end point of side 3 (B1= and L1=).
G11 B60 L120 K60 B1=-60 L1=120	Sides 4, 5, and 6 are milled. The following have been programmed: - The point of intersection for sides 4 and 6 (B and L) - The chamfer (K-expression) around this point - The end point of side 6 (B1= and L1=).
G40	Delete radius compensation.
G90	Switch back to absolute measurement programming.
G1 X100 Y10	Move the tool away from the workpiece
G0 Z10 M30	End of program

Two-line geometry

F = milling path
R = radius compensation
W= tool radius

The inner pocket can be programmed in two-line geometry operation using the G11 function.

Two-line geometry



N9012	
G17 T1 M6	Activate main plane. Insert tool (milling tool diameter 10 mm).
G0 X80 Y25 Z0 S1000 M3	Switch spindle on. Move the tool to point B and then above the workpiece.
G1 Z-10 F300	Move the tool to working depth.
G43 X105	Move the tool to the starting point for the entry circle.
G42	Call the radius compensation RIGHT.
G2 X80 Y0 R25 F300	Advance to the contour over the entry circle.
G11 X0 Y90 B180 B1=90 R15 R1=15	Mill- Along the X axis (B180) - Along the radius (R15) - Along the Y-axis (B1=90) - Along the second radius (R1=15).
G11 X60 Y150 B0 B1=90 R1=15	Mill- Parallel to the X-axis (B0) - Parallel to the Y-axis (B1=90) - Along the second radius (R1=15).
G11 X200 Y0 B0 B1=120 R15 R1=20	Mill -parallel to the X-axis (B0) - Follow the first radius (R15) - Along the 60-ramp (B1=120) - Follow the second radius (R20).
G1 X80 Y0	Mill along the X-axis up to the starting point of the circular movement that leaves the contour.
G2 X55 Y25 R25	Leave the contour with a circular movement.
G40	Delete radius compensation.
G0 Z200 M30	Tool retraction. End of program.

5.10 G14 Repeat Function

Repetition of a specific number of blocks in a part program or subprogram.

Address description

- ▶ **J** number of repeats
- ▶ **K** repeat decrement
- ▶ **N1=** repeater begin block
- ▶ **N2=** repeater end block

Format

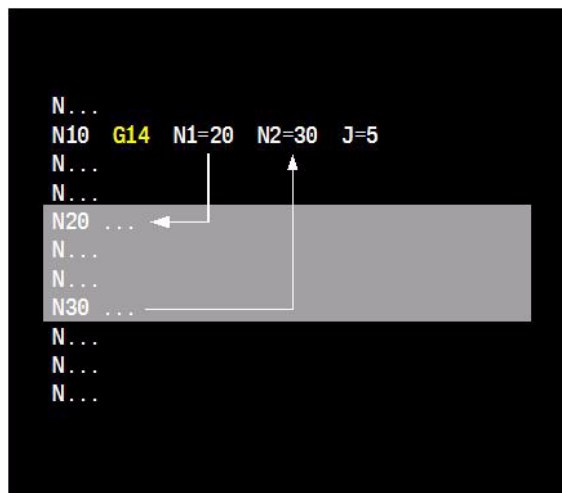
G14 N1=... {N2=...} {J...} {K...}

Application

- The block numbers N1= and N2= must both be in the same part program or subprogram.
- If N2= is not programmed, then only the block identified with N1= is repeated in accordance with specifications.
- If the parameter for the number of repetitions J is not programmed, the program run is only repeated once. J does not have to be a whole number. The number of repetitions is determined by the value before the decimal point
- A repeating program run can be incorporated into another repeating program run. The nesting depth can be adjusted in the configuration file "CfgNestingLevels(repeatlevels:=4)"
- Only one repetition is carried out in a G14 block, if J>0. The CNC uses the standard value K1 if the K parameter is not programmed.

Changes to V5xx

- See "G14_E" on page 500.



Example

Programming a repeat function

N10

N13

N14 G14 N1=10 N2=13 J4 'REPEAT BLOCKS N10-N13

N15 E2=4

N17

N19

N20 G14 N1=17 N2=19 J=E2 'REPEAT BLOCKS N17-N19

N27

N29

N30 G14 N1=17 N2=19 J4 K2 'REPEAT BLOCKS N27-N29

5.11 G17 Main Plane XY, Tool Z

The position of the tool axis is determined by the main spindle of the tool machine. G17 is used to specify that the main plane for the milling work is the XY-plane and that the tool axis is the Z axis.

Address description

No specific addresses.

Application

Modality

G17 is modal with G18 and G19.

Default setting

The last active plane always takes effect after switch-on.

Operations in the plane

The calculations required for the radius compensation, geometry (G64), polar coordinates, milling cycles, pocket cycles etc. are executed in the current plane, i.e. in the XY-plane in the case of G17.

Operations in the tool axis.

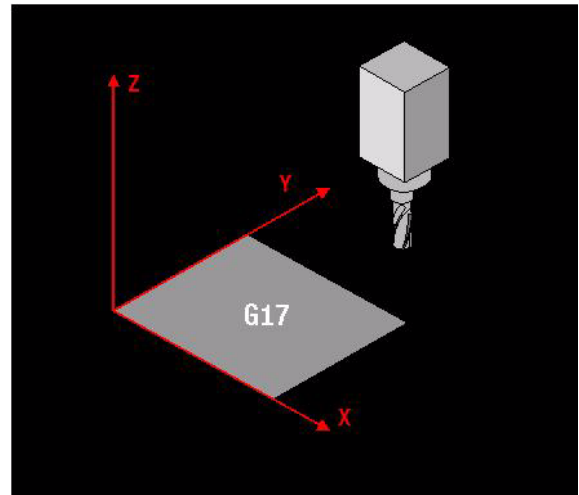
The tool length compensation and the fixed cycles for drilling work are executed in the current tool axis.

Milling head

When a milling head is used, the axis configuration of the tool machine remains unchanged. The tool axis and the tool length compensation are determined by the head position.

Deleting

The G17 function is deleted by switching to a different working plane using G18 or G19.



Turning

Format

G17 Y1=... Z1=...

Application

The machine can process workpieces in various working planes when turning. The working plane during turning (G36) is defined with:

- G17 Y1= 1 Z1=2, tool axis Z (vertical) (see figure).

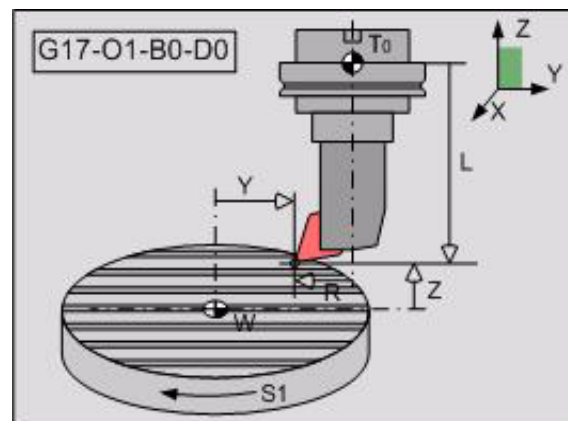
The G17 function defines the axis in which the tool data for length (L) and radius (R) is calculated:

- G17: L in Z direction, R in Y direction

During turning, machining is carried out as individual DIN commands in the YZ or XZ working area. In the various working planes, machining with turning cycles is only carried out in the YZ working surface.

Note

- Y1=1 (first main axis); Z1=2 (second main axis).
- The angle (positive) and circle direction (CW) are defined from the Y axis to the Z axis.
- The G17 plane during turning overwrites the current G17/G18 plane during milling.
- G37 (milling) switches the G17 plane during turning back to the current G17/G18 plane during milling.
- Depending on the tool orientation (O), the tool radius (R) is calculated as a shift in the Y axis.



5.12 G18 Main Plane XZ, Tool Y

The position of the tool axis is determined by the main spindle of the tool machine. G18 is used to specify that the main plane for the milling work is the XZ-plane and that the tool axis is the Y axis.

Address description

No specific addresses.

Application

Modality

G18 is modal with G17 and G19.

Default setting

The last active plane always takes effect after switch-on.

Operations in the plane

The calculations required for the radius compensation, geometry (G64), polar coordinates, milling cycles, pocket cycles etc. are executed in the current plane, i.e. in the XZ-plane in the case of G18.

Operations in the tool axis

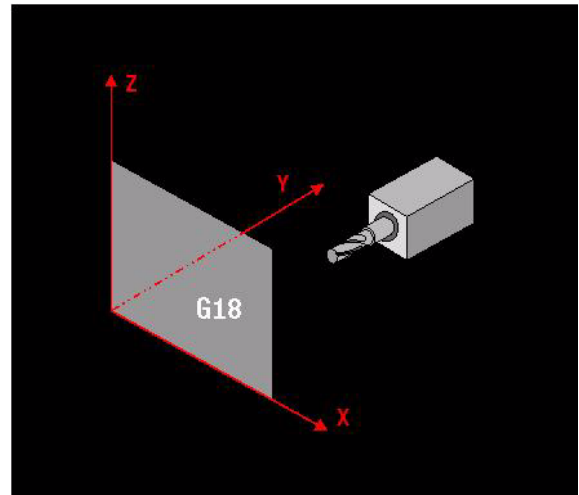
The tool length compensation and the fixed cycles for drilling work are executed in the current tool axis.

Milling head

When a milling head is used, the axis configuration of the tool machine remains unchanged. The tool axis and the tool length compensation are determined by the head position.

Deleting.

The G18 function is deleted by switching to a different working plane using G17 or G19.



Turning

Format

G18 =... Z1=...

Application

The machine can process workpieces in various working planes when turning. The working plane during turning (G36) is defined with:

- G18 Y1=1 Z1=2, tool axis Y (horizontal) (see figure).

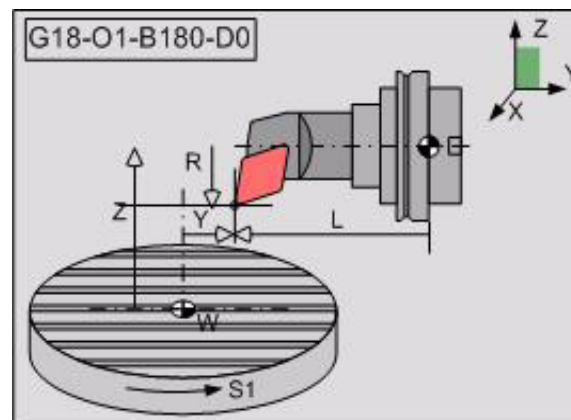
The G18 function defines the axis in which the tool data for length (L) and radius (R) is calculated:

- G18: L in Y direction, R in Z direction

During turning, machining is carried out as individual DIN commands in the YZ or XZ working area. In the various working planes, machining with turning cycles is only carried out in the YZ working surface.

Note:

- Y1=1 (first main axis); Z1=2 (second main axis).
- The angle (positive) and circle direction (CW) are defined from the Y axis to the Z axis.
- The G18 plane during turning overwrites the current G17/G18 plane during milling.
- G37 (milling) switches the G18 plane during turning back to the current G17/G18 plane during milling.
- The tool radius (R) is calculated as a shift in the Z axis, depending on the tool orientation (O).



5.13 G19 Main Plane YZ, Tool X

The position of the tool axis is determined by the main spindle of the tool machine. G19 is used to specify that the main plane for the milling work is the YZ-plane and that the tool axis is the X axis.

Address description

No specific addresses.

Application

Modality

G19 is modal with G17 and G19.

Default setting

The last active plane always takes effect after switch-on.

Operations in the plane

The calculations required for the radius compensation, geometry (G64), polar coordinates, milling cycles, pocket cycles etc. are executed in the current plane, i.e. in the YZ-plane in the case of G19.

Operations in the tool axis

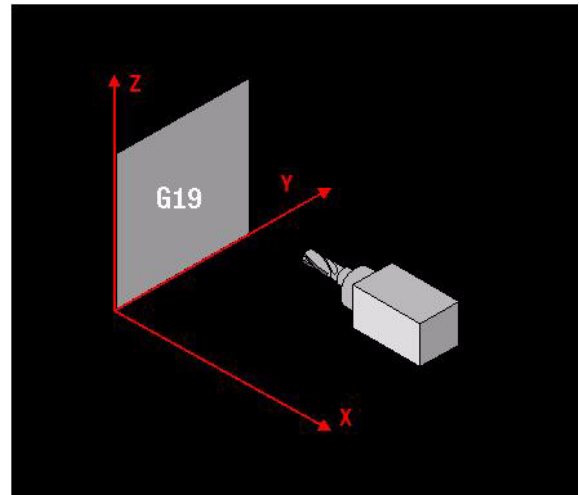
The tool length compensation and the fixed cycles for drilling work are executed in the current tool axis.

Milling head

When a milling head is used, the axis configuration of the tool machine remains unchanged. The tool axis and the tool length compensation are determined by the head position.

Deleting

The G19 function is deleted by switching to a different working plane using G17 or G18.



5.14 G22 Subprogram Call

Calling (with path specification if necessary) and execution of a subprogram (macro) from a main program or subprogram using standard operations.

Address description

- ▶ E parameter definition
- ▶ N= subprogram name
- ▶ N5= folder
- ▶ O1= call status

Format

Calling a macro:

■ G22 N=... {N5=...} {E...=} {O1=E...}

Activating a macro on the condition that E...>0:

■ G22 E... N=... {N5=...} {E...=} {O1=E...}

Application

Subprogram name (N=)

The name of the subprogram can be a number or (N=...).

The desired subprogram can be called with a file name or the file number. For this purpose, the full subprogram name (including <.mm>) must be enclosed in double quotation marks <"> in the N= parameter. For example, G22 N="subprogram.mm". The only file name extension that is allowed is .mm.

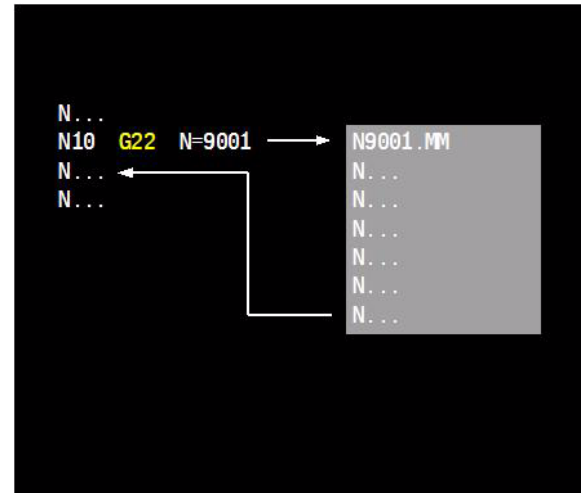
Other directory (N5=)

The macro to be called can be located in another directory. The path to this directory must be enclosed in double quotation marks <"> and should be entered separately in the N5= parameter, or should be before the file name in the N= parameter.

The path must be entered in full and final form, e.g.

N5="v:\nc_prog\Teil1\Makros\" or

N="v:\nc_prog\Teil1\Makros\makro.mm"



Querying the call status (O1=)

The status of the macro call can be queried using O1=E... The result of the call is written in the E parameter.

- E... = 0 macro call successful
- E... = 1 macro call not successful, e.g. macro not found

Nesting macros

If another macro is called from within a macro, then the called macro is referred to as a nested macro. At the end of a nested macro, the execution of the calling macro is continued.

Maximum nesting depth

The maximum nesting depth can be defined in a configuration parameter for each channel, e.g.: NCchannel::channel1::macroCalls. CfgNestingLevels (key:="Channel1", repeatLevels:=4, programCalls:=1, **macroCalls:=8**).

M30 in a macro

The full program execution is stopped after M30 in a macro.

Procedure

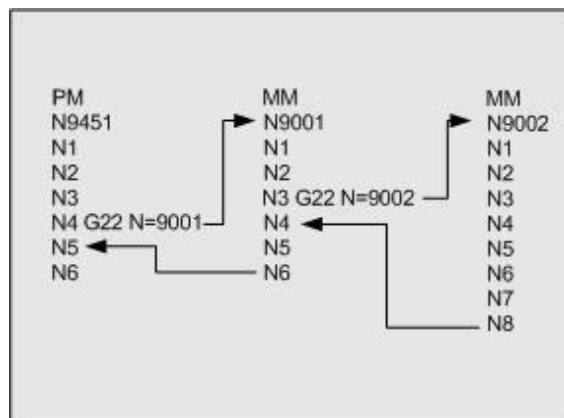
A macro is executed in full if it is called from within a main program or another macro. After the macro has been executed, program execution continues with the next block after G22.

Example

Calling a macro from another macro. (see figure)

The macro 9001 is called in block N4 of main program 9451.

This macro is executed up to N3. Macro N9002 is called in this block. Macro 9002 is processed in full before jumping back to the first block after the G22 block in macro 9001. Execution of this macro is then continued from N4 to the end. The system then jumps back to block N5 of the main program 9451.



5.15 G23 Program Call

Calling (with path specification if necessary) and execution of a main program from within another main program.

Format

G23 N=... {N5=...} {O1=E...}

Address description

- ▶ N program name
- ▶ N5= folder
- ▶ O1= call status

Application

Main program name (N=)

The desired main program can be called with the file name or the file number. For this purpose, the full main program name (including <.pm>) must be enclosed in double quotation marks "<"> in the N parameter. For example G22 N="main program.pm". The only file name extension that is allowed is .pm.

Other directory (N5=)

The main program to be called can be located in another directory. The path to this directory must be enclosed in double quotation marks "<"> and should be entered separately in the N5= parameter, or should be before the file name in the N= parameter.

The path must be entered in full and final form, e.g.

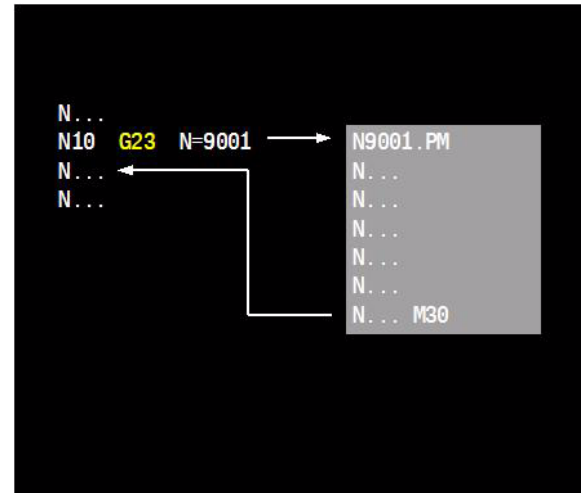
N5="v:\nc_prog\Teil1\Programme\" or

N="v:\nc_prog\Teil1\Programme\programm-2.pm"

Querying the call status (O1=)

The status of the main program call can be queried using O1=E... The result of the call is written in the E parameter.

- E... = 0 program call successful
- E... = 1 program call not successful, e.g. program not found



Nesting main programs

If another main program is called from within a main program, then the called main program is referred to as a nested main program. At the end of a nested main program, the execution of the calling main program is continued

Maximum nesting depth

The maximum nesting depth can be defined in a configuration parameter for each channel, e.g.: NCchannel::channel1::macroCalls. CfgNestingLevels (key:="Channel1", repeatLevels:=4, **programCalls:=2**, macroCalls:=8).

M30 in called main program

M30 must be programmed at the end of the called main program. After this M30, the system jumps back to the calling main program. This M30 does not stop program execution.

Notes

A macro can also contain a G23 function.

Main programs or macros that are called can contain jump instructions.

5.16 G25 Enable Feed/Speed Override

Reactivation of the feed and speed override after being switched off by G26.

Address description

No specific addresses.

Default setting

G25 is automatically effective at the start of a part program.

Application

Modality

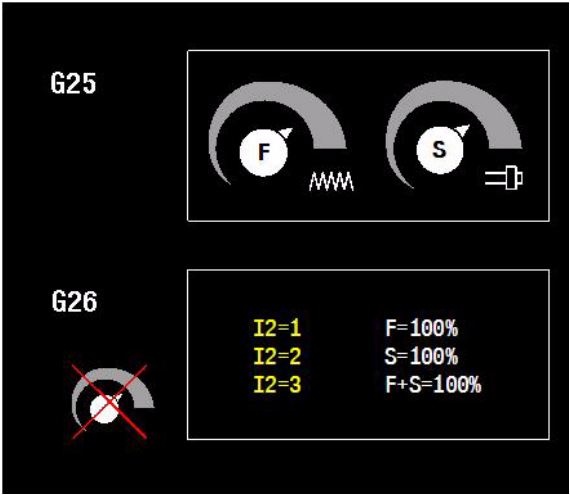
G25 is modal with G26.

Example

Switching on the feed and speed override

G25

G25 Switch on the feed and speed override



5.17 G26 Disable Feed/Speed Override

Deactivation of the feed and speed override. The feed rate and speed are fixed at 100% if the override is disabled.

Address description

► I2= 1=F100%; 2=S100%; 3=F+S100%

Format

G26 I2=

Default setting

I2=1 is activated if the address I2= is not programmed for G26.

Application

Disabling the feed override (F=100%):

■ G26 I2=1

■ G26

Disabling the speed override (S=100%):

■ G26 I2=2.

Disabling the feed and speed override (F and S=100%):

■ G26 I2=3

Modality

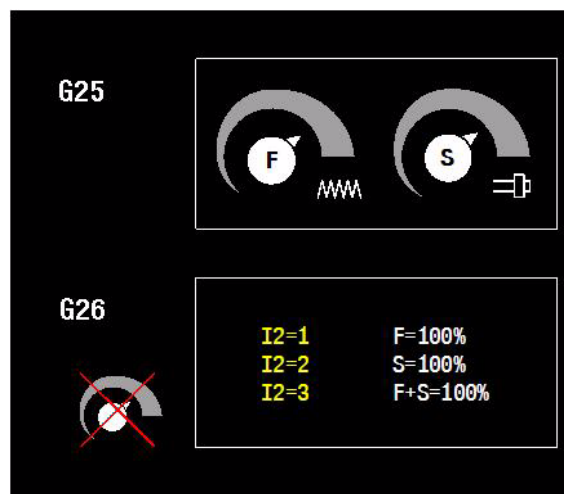
G26 is modal with G25.

Cancelation

G26 is deleted with G25, M30, Cancel program, or CNC reset.

Changes to V5xx

■ See "G26" on page 500.



Example

Disabling the speed override

G26 I2=2

G26 I2=2 Disable the speed override, i.e. fix S at 100%

Disabling the feed and speed override

G26 I2=3

G26 I2=3 Disable the feed and speed override, i.e. fix F and S at 100%

5.18 G27 Reset Positioning Functions

Deletion of the parameters for positioning logic behavior, as specified in G28. Default settings become active.

Address description

No specific addresses.

Format

G27

Application

Modality

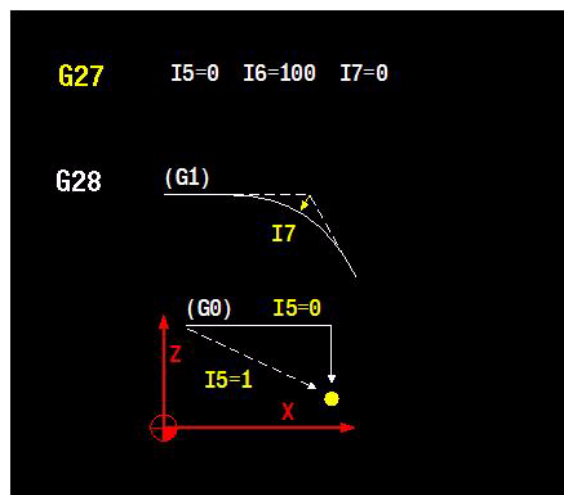
G27 is modal with G28.

Deleting parameters

All parameters used for G28 are reset to their fixed default values with G27, CNC reset, Cancel program, or by M30.

G27 is automatically effective at the start of an NC program.

G27 results in G28 I5=0 I6=100 I7=0



5.19 G28 Positioning Functions

Setting options for the positioning functions. The feed rate and rapid traverse movements, positioning logic, acceleration, jerk and contour tolerance can be defined.

Address description

- I5= position logic: 0=with, 1=without
- I6= reduction acceleration/jerk [%]
- I7= contour tolerance

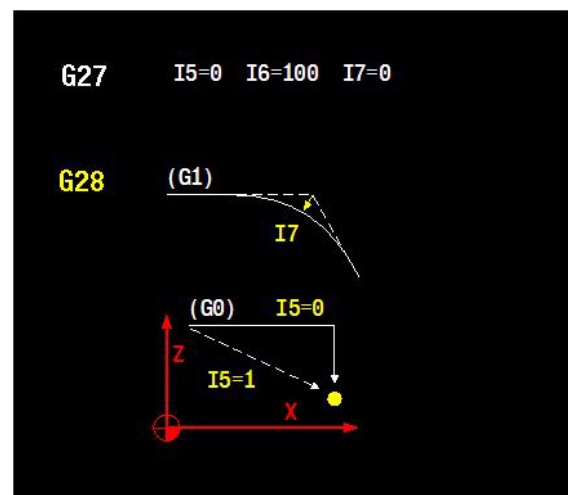
Format

G28 {I5=...} {I6=...} {I7=...}

Default setting

Deleting individual options: G28 {I5=0} {I6=100} {I7=0}

Deleting all options: G27



Application

The feed movement is performed without a precision stop. No stops are taken into account between the movements.

The rapid traverse movement is executed with a precision stop. A stop is taken into account between the movements.

G28

Positioning logic for G0

G28 I5=0 G0 with positioning logic (on-position)
 G28 I5=1 G0 without positioning logic

Reducing acceleration and jerk

G28 I6=100 Factor I6= (between 5 and 100%, normal value 100%)
 G28 I6=... overrules the following machine parameters:
 Path: **maxPathJerk** and **maxPathYank**
 Axis: **maxAcceleration**, **maxDeceleration**, and **maxJerk**
 Address I6= is effective for G0, G1, G2, G3.

Movement with programmable contour tolerance

G28 I7=0 I7= between 0 and 10,000 [mm/inch]). Machine
 G28 I7=... parameter **pathTolerance** applies as the normal value
 for I7=0. Machine parameter **pathToleranceHi** is not
 overruled by address I7=.
 Address I7= is effective for G0, G1, G2, G3.

Changes to V5xx

- See "G28" on page 501.

5.20 G29 Jump Function

Conditional or unconditional jump to another part program or macro section in the same program. During the jump, the program is continued from the block programmed under N=.

Address description

- ▶ **I** search direction
- ▶ **K** jump decrement
- ▶ **E** jump condition: $E > 0$
- ▶ **N=** jump to block number

Format

G29 {E...} N=... {K...} {I...}

Application

The E parameter is used as the jump condition.

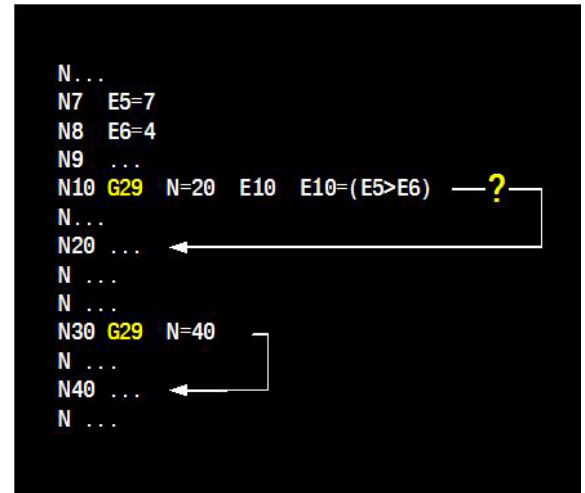
The jump is unconditional if E... has not been programmed.

If E... was programmed, the jump is only executed if the value of E... is > 0 . The value of the E parameter is reduced by the value of the K address if $K \geq 0$. If $K < -0.5$, then an error message is issued.

The E parameter is reduced by 1 after each jump if the K address was not programmed.

Further jump conditions such as $=$, $<>$, $>$, \geq , $<$, \leq can be programmed if a relational expression is used together with the G29 function.

Jumps can be made both forwards and backwards in a program. This can be controlled with the I address. With $I=1$ or $I=0$, only forward searches are performed. If $I=-1$ or no data is specified, the system first jumps to the start of the program before searching forwards for the block number.



Example

Unconditional jump

G29 N=...

G29 Jump instruction
N=... Jump to block number

Conditional jump

N50 G1 ...

...

G29 E1 N=50 E1=E2>E3

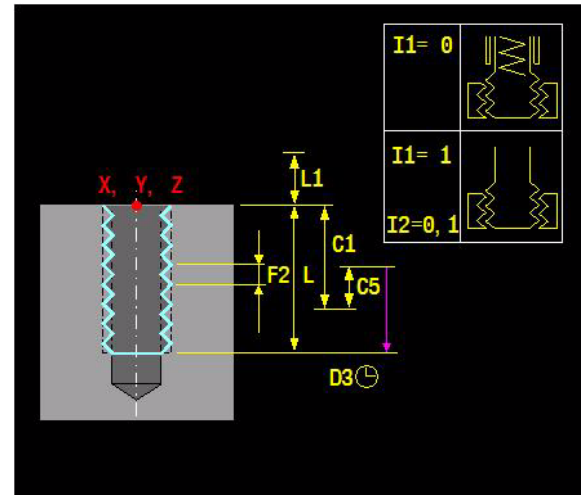
G29 Jump instruction
E1 Jump condition
N=50 Jump to block number N=50
E1=E2>E3 Jump condition: if the value of E2 is greater than the
 value of E3, then parameter E1 is assigned the value
 1 before the jump to N... is made.

5.21 G31 Tapping with Chip Breaking

Basic function for tapping a thread, in several infeeds, up to the programmed depth.

Address description

- ▶ **L depth** Distance between the workpiece surface and the thread end. The algebraic sign determines the working direction.
- ▶ **F2= pitch** Pitch of the thread. No algebraic sign. The feed rate is calculated from the rotational speed and the thread pitch.
- ▶ **D orientation angle spindle** Angle at which the tool is positioned before the thread is cut. This means you can recut the thread if necessary. Only effective for interpolating tapping.
- ▶ **L1= setup clearance** Distance between the tool tip (starting position) and the tool surface.
- ▶ **C1= cutting depth until chip break** Infeed which is followed by a chip break. No algebraic sign.
- ▶ **C5= retract distance for chip break** The tool is retracted by the specified distance during chip breaking. Entering 0 means that it is fully retracted from the hole (to the safety clearance) for chip removal. No algebraic sign.
- ▶ **D3= dwell time at bottom** Dwell time at the bottom of the thread.
- ▶ **I1= guided or interpolated** 0= guided, 1 =interpolating.
- ▶ **I2= thread direction (0=right, 1=left)** Specifies a right-hand or left-hand thread:
 I2=0 right-hand thread (clockwise)
 I2=1 left-hand thread (counter-clockwise).



Application

Retracting after a program interruption

To interrupt the tapping process, press the Stop spindle and feed button and retract the tool from the hole in a controlled manner. The spindle is automatically started (guided or interpolating) when it is retracted so that the thread is not damaged.

Override

The feed rate is determined by the rotational speed. The speed is automatically adjusted if you change the feed rate override during tapping. The spindle override is not active.

End of movement

The spindle comes to a stop at the end of the movement. Before the next operation, restart the spindle with M3 (or M4).

Procedure

■ Procedure for a guided tapping movement (I1=0)

Start with a running or stationary spindle. The tool axis is synchronized with the spindle speed during the build-up of the movement. The tool moves to thread depth at the calculated feed rate. The spindle speed is reduced in line with the tool axis. The spindle comes to a stop at the end of the movement.

■ Procedure for an interpolated tapping movement (I1=1)

Start with a stationary spindle. The spindle starting angle (D) is approached if programmed. The spindle speed increases in line with the tool axis. The tool moves to thread depth at the calculated feed rate. The spindle speed is reduced in line with the tool axis.

- 1 Depending on the definition, the tool executes a spindle orientation.
- 2 The tool travels the programmed safety clearance; the feed rate is synchronized with the speed.
- 3 The tool moves to the programmed infeed depth, the direction of spindle rotation is reversed, and the tool retracts by a specific distance or fully for purposes of chip release, depending on the definition.
- 4 The direction of spindle rotation is then reversed again and the tool advances to the next plunging depth.
- 5 This process (3 to 4) is repeated until the programmed thread depth is reached.
- 6 Waiting time; depends on the definition.
- 7 The tool is then retracted to the safety clearance and the spindle comes to a stop.

Example

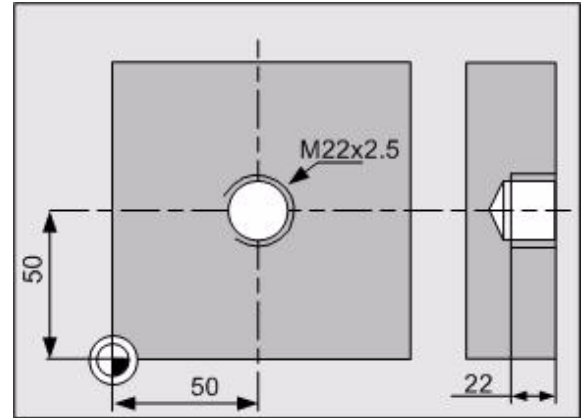
Program example: (see figure)

T202 M6 S56 F140

G0 X50 Y50 Z0

G31 L9 F2=2,5 L1=5 C5=5 D3=1 I2=0

G0 Advance to the tapping position
G31 Perform tapping at the programmed position



5.22 G37 Milling Operation

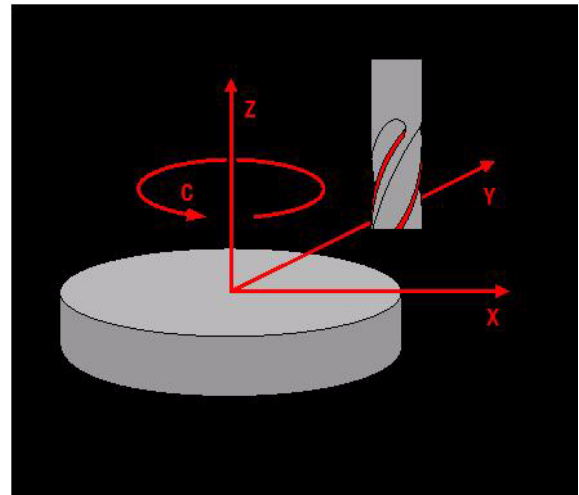
Ending turning. Switching the machine to milling operation.

Address description

No specific addresses.

Application

- The CNC reactivates the C-axis.
- If the turning spindle is still rotating at the start of G37, it is stopped first.
- The position of the rotary axes is shown on the screen monitor with a value between 0 and 359.999 degrees.
- G94 is activated.
- G37 remains active until it is canceled by G36. G37 is not canceled by M30 or Cancel program. G37 is always active after control run-up or CNC reset.



5.23 G39 Tool Offset Change

The programmed contour can be changed by an offset.

Address description

- ▶ L tool length offset
- ▶ R tool radius offset

Format

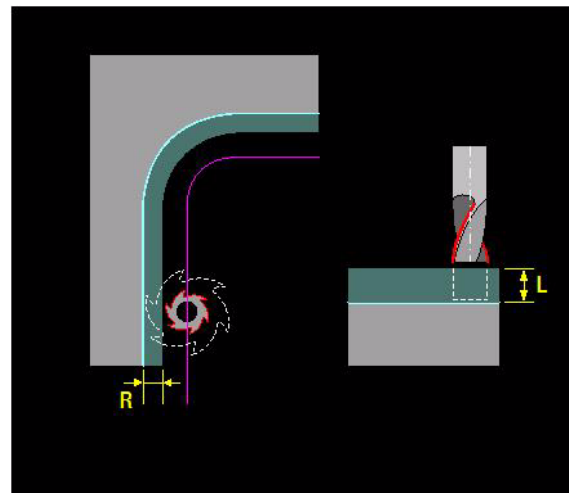
G39 {R...} {L...}: activate offset

G39 L0 and/or R0: deactivate offset

Default setting

G39 L0 R0

The G39 function is deleted by End of program (M30), Cancel program, and CNC reset.



Application

Tool length offset

The tool length offset operates in the direction of the infeed axis. Changes to the tool length offset take effect with the next advance movement.

Tool radius offset

The tool radius offset operates in the working plane and is only effective during active milling radius compensation.

Changes to the tool radius offset when the milling radius compensation is not activated only take effect after milling radius compensation (G41/G42, G43/G44) has been activated.

Offset activation must not be programmed during active radius compensation.

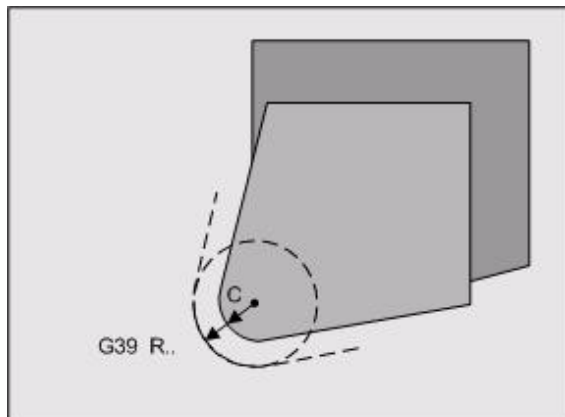
Offset programming is retained after a change of tool (M6, M66) or plane (G17, G18, G19).

The offset R influences the tool corner radius C during turning (G36) and is only effective with active radius compensation.

The offset of the tool corner radius is fully added to the center point of the corner radius (as for orientation 0), and is therefore independent of the active tool orientation.



The radius offset is suppressed when the following functions are activated: G6, G83-G89, G141, G182. The length offset remains in effect. The offset programming should be deactivated before these functions are used.



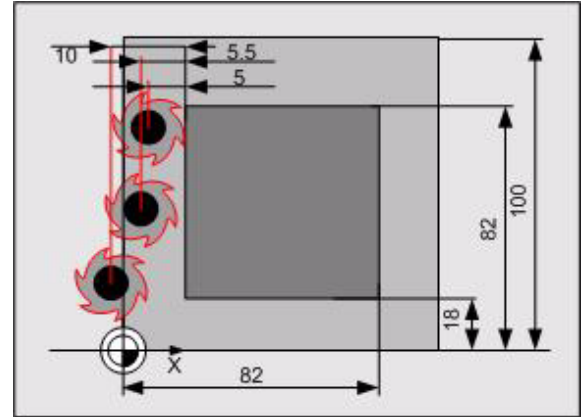
Changes to V5xx

- See "G39_G41_L" on page 501.

Example

Milling a rectangle by roughing twice and finishing once

Program example:



N39001	
G98 X-10 Y-10 Z10 I120 J120 K-60	Specify the graphic window
G99 X0 Y0 Z0 I100 J100 K-40	Specify material
T1 M6	Insert the tool (milling radius 5 mm)
G39 L0 R9	Activate tool radius offset. The offset is 9 mm. (The milling radius for radius compensation is $(5+9 =) 14$ mm).
F500 S1000 M3	Activate feed rate and spindle speed
G0 X0 Y-20 Z5	Advance to starting position
G1 Z-10	Go to depth
N8 G43 X18	Advance to contour with radius compensation
G41 Y82	Rough the rectangle for the first time.
X82	
Y18	
X0	
N13 G40	Switch off radius compensation
G39 R0.5	Change the tool radius offset. The offset is 0.5 mm. (The milling radius for radius compensation is $(5+0.5 =) 5.5$ mm).
G14 N1=8 N2=13	Repeat rectangle (2nd roughing movement).
G39 R0	Change the tool radius offset. The offset is 0 mm. (The milling radius for radius compensation is 5 mm).
G14 N1=8 N2=13	Finish rectangle.
G0 Z10	Retract the tool
M30	End of program

5.24 G40 Cancel Tool Radius Compensation

Deletion of the tool radius compensation. The tool now moves along the programmed path on the workpiece.

Address description

No specific addresses.

Application

Modality

G40 is modal with G41, G42, G43, G44, and G141.

Default setting

G40 automatically takes effect after:

- Control activation
- CNC reset
- Program cancelation
- M30.

Changes to V5xx

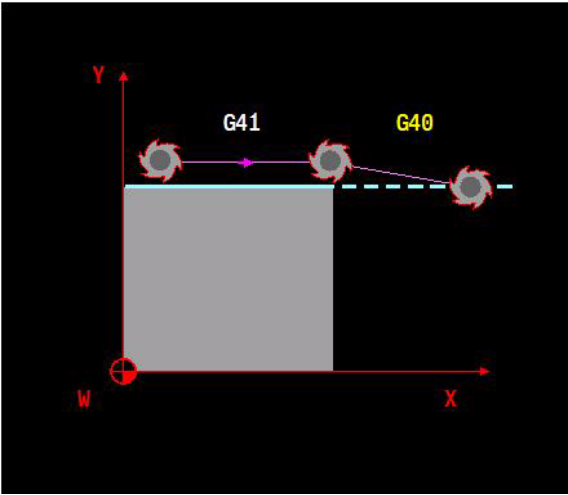
- See "G40_G91" on page 502.
- See "G41-G42_G40" on page 503.

Example

Delete radius compensation

G42
G1 X...
G1 X... Y...
G40

G42	Activate the radius compensation on the right.
G1	
G1	
G40	Delete radius compensation.



5.25 G41 Tool Radius Compensation, Left

Programming of the workpiece dimensions instead of the milling path. The tool path is automatically calculated by the CNC as a path parallel to the programmed workpiece contour. G41 activates LEFT-hand radius compensation on the workpiece, viewed from the direction of workpiece movement.

Address description

No specific addresses.

Application

Modality

G41 is modal with G40, G42, G43, G44, and G141.

Tool radius

The tool radius stored in the tool table is used for the radius compensation. It is assumed that this radius is positive when the program is executed. The following applies if the radius value is negative:

- G41 and negative radius = G42 and positive radius

An error is reported if the tool radius is too great in relation to the contour (e.g. circle radius, internal corner etc.).

Nominal radius

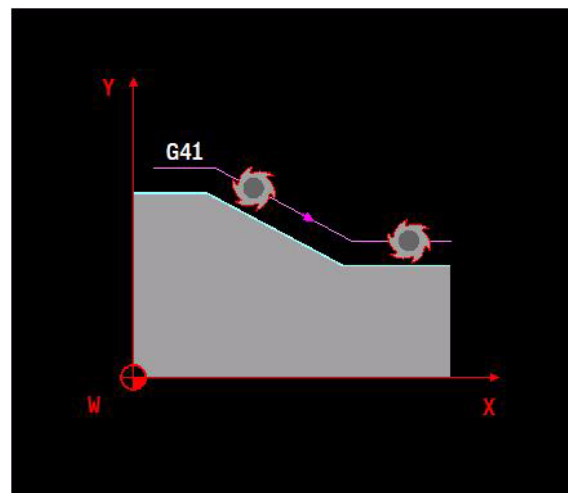
When the nominal radius is used in programming, G39 (negative allowance equal to nominal radius) still allows the actual radius to be used for G41.

Starting radius compensation

There are 3 options for starting radius compensation:

- Use G41 directly
- Use G43 or G44
- Use a tangential approach (G61)

You must ensure that the tool can come into contact with the workpiece at the start of radius compensation. The starting point should therefore be at a secure point.



Contour transitions

Internal contours: When radius compensation is applied, the path is always followed at the same distance from the programmed contour, except for at the points of intersection of the contour elements. These points of intersection are automatically calculated by the CNC.

External contours: The point of intersection of external contour elements is always calculated when the angle between the elements exceeds a specified configuration value. The tool is then advanced to this contour.

External contours with acute angles: If the angle between two external contour elements is smaller than a specified configuration value, then CNC creates a circular movement between the two elements.

Switching from one radius compensation function to another

If, for example, a switch is made from G41 to G42, G43, or G44, then the tool movement finishes at a position that was calculated with G41 and starts at a position that was calculated with G42, G43, or G44. If the two positions do not coincide, a direct feed movement is made from one position to the other.

Rotary axis movement

A simultaneous traverse movement in a rotary axis and in the axes of the main plane is not possible during effective radius compensation.

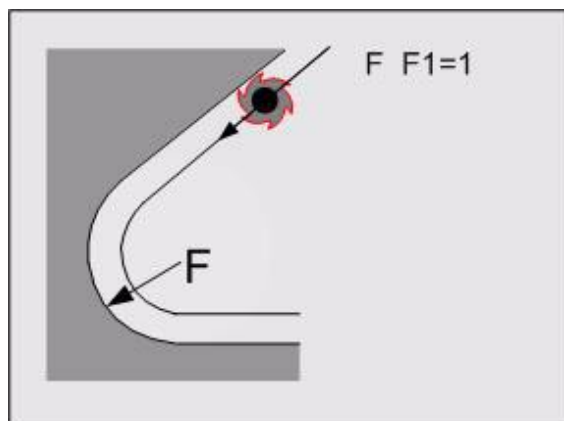
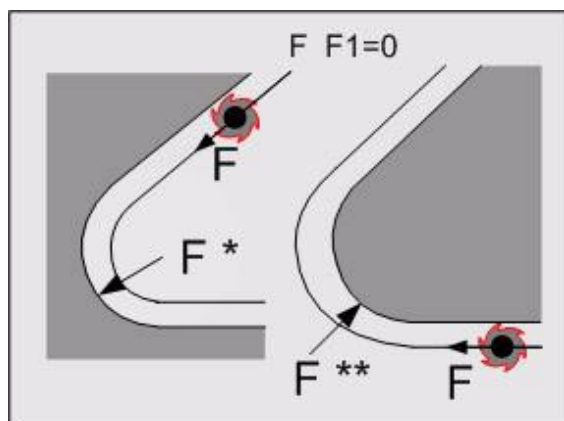
Constant cutting feed during radius compensation for circles

The parameter F1= is used to keep the programmed feed rate on the workpiece contour constant, regardless of the milling radius and the contour shape.

■ **F1=0** No constant cutting feed (default setting, M30, Cancel program softkey, or after CNC reset softkey). The programmed feed rate should represent the speed of the tool tip. (see figure).

* = cutting feed too great; ** = cutting feed too small

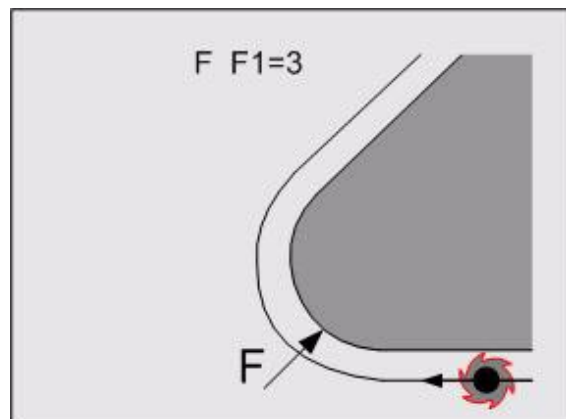
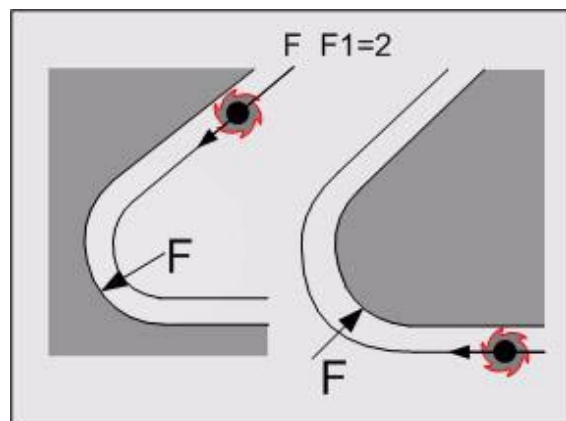
■ **F1=1** Constant cutting feed only on the inside of circular arcs. The programmed feed rate is reduced to ensure that the tool tip travels to the inside of a circular arc at the reduced speed (see figure).



- **F1=2** Constant cutting feed on the inside- and outside of circular arcs. The programmed feed rate is reduced (inner circular arc) or increased (outer circular arc) to ensure that the tool tip travels at the recalculated speed. The maximum feed rate is used if the increased speed is greater than the maximum feed rate defined via a machine parameter (see figure)
- **F1=3** Constant cutting feed only to the outside of circular arcs. The programmed feed rate is increased to ensure that the tool tip travels to the outside of a circular arc at the increased speed (see center figure).

Changes to V5xx

- See "G41-G42" on page 503.
- See "G39_G41_L" on page 501.
- See "G1, G41 und G64" on page 493.
- See "G54_G41" on page 504.
- See "G61-G62_G41-G42" on page 504.
- See "G79_G41" on page 510.



Example

Radius compensation (see figure)

G0 X200 Y-20 Z-5

G43

G1 X150 Y...

G41 X0

G1 Y80

G1 X150

G1 Y50

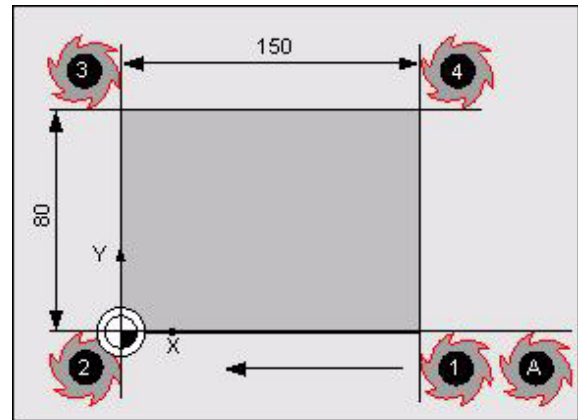
G40

G0 Move the tool in rapid traverse to X200, Y-20

G43 Radius compensation to end point

G41 Activate left-hand radius compensation

G40 Delete radius compensation



5.26 G42 Tool Radius Compensation, Right

Programming of the workpiece dimensions instead of the milling path. The tool path is automatically calculated by the CNC as a path parallel to the programmed workpiece contour. G42 activates RIGHT-hand radius compensation on the workpiece, viewed from the direction of workpiece movement.

Address description

No specific addresses.

Application

Modality

G42 is modal with G40, G41, G43, G44, and G141.

Tool radius

The tool radius stored in the tool table is used for the radius compensation. It is assumed that this radius is positive when the program is executed. The following applies if the radius value is negative:

- G42 and negative radius = G41 and positive radius

An error is reported if the tool radius is too great in relation to the contour (e.g. circle radius, internal corner etc.).

Nominal radius

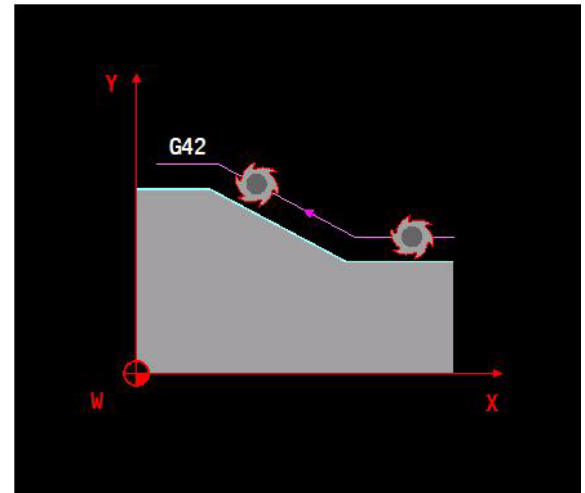
When the nominal radius is used in programming, G39 (negative allowance equal to nominal radius) still allows the actual radius to be used for G42.

Starting radius compensation

There are 3 options for starting radius compensation:

- Use G42 directly
- Use G43 or G44
- Use a tangential approach (G61)

You must ensure that the tool can come into contact with the workpiece at the start of radius compensation. The starting point should therefore be at a secure point.



Contour transitions

Internal contours: When radius compensation is applied, the path is always followed at the same distance from the programmed contour, except for at the points of intersection of the contour elements. These points of intersection are automatically calculated by the CNC.

External contours: The point of intersection of external contour elements is always calculated when the angle between the elements exceeds a specified configuration value. The tool is then advanced to this contour.

External contours with acute angles: If the angle between two external contour elements is smaller than a specified configuration value, then CNC creates a circular movement between the two elements.

Switching from one radius compensation function to another

If, for example, a switch is made from G42 to G41, G43, or G44, then the tool movement finishes at a position that was calculated with G42 and starts at a position that was calculated with G41, G43, or G44. If the two positions do not coincide, a direct feed movement is made from one position to the other.

Rotary axis movement

A simultaneous traverse movement in a rotary axis and in the axes of the main plane is not possible during effective radius compensation.

Constant cutting feed during radius compensation for circles

See description for G41.

Changes to V5xx

- See "G39_G41_L" on page 501.
- See "G41-G42" on page 503.
- See "G61 und G62" on page 504.

5.27 G43 Tool Radius Compensation to End Point

Positioning of the tool with milling radius compensation UP TO a programmed position. The tool radius is subtracted from the programmed distance.

Address description

No specific addresses.

Application

Modality

G43 is modal with G40, G41, G42, G44, and G141.

Using G43

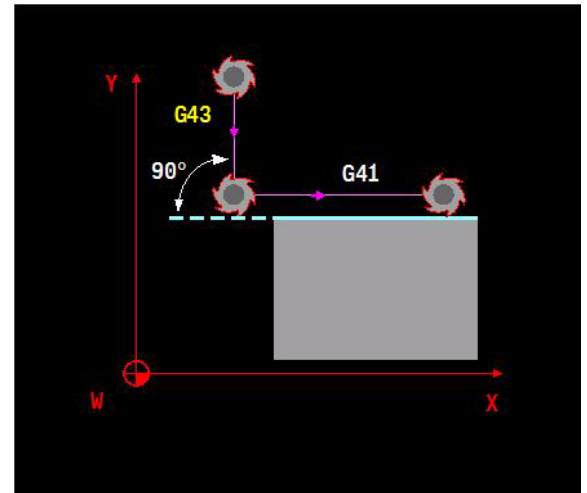
G43 can only be used in connection with a paraxial movement. Applying the G43 function in connection with a circular movement produces an error message. Circular movements should only be programmed in connection with G41 or G42.

Accessing the contour

The G43 function enables vertical access to the contour via the perpendicular of any contour element. This is the recommended contour access method (less chance of collision than with G41 and G42).

Switching from one radius compensation function to another

If, for example, a switch is made from G43 to G41, G42, or G44, then the tool movement finishes at a position that was calculated with G43 and starts at a position that was calculated with G41, G42, or G44. If the two positions do not coincide, a direct feed movement is made from one position to the other.



Example

Switching on radius compensation (see figure below)

G0 X120 Y-15 Z10

G1 Z-10

G43 Y20

G41 X35

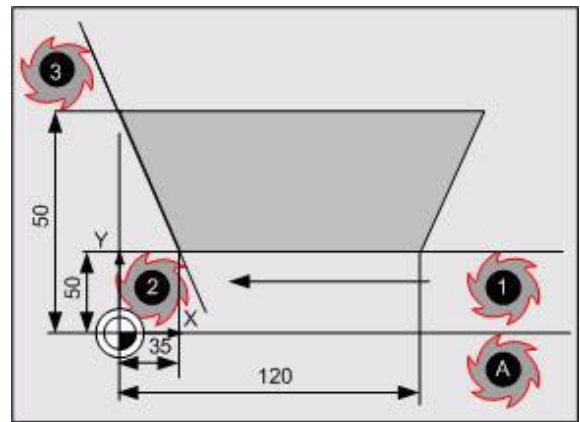
G1 X15 Y50

G0

G1

G43 Radius compensation to end point

G41 Activate left-hand radius compensation



5.28 G44 Tool Radius Compensation Past End Point

Positioning of the tool with milling radius compensation PAST a programmed position. The tool radius is added to the programmed distance.

Address description

No specific addresses.

Application

Modality

G44 is modal with G40, G41, G42, G43, and G141.

Using G44

G44 can only be used in connection with a paraxial movement. Applying the G44 function in connection with a circular movement produces an error message. Circular movements should only be programmed in connection with G41 or G42.

Accessing the contour

The G44 function enables vertical access to the contour via the perpendicular of any contour element. This is the recommended contour access method (less chance of collision than with G41 and G42).

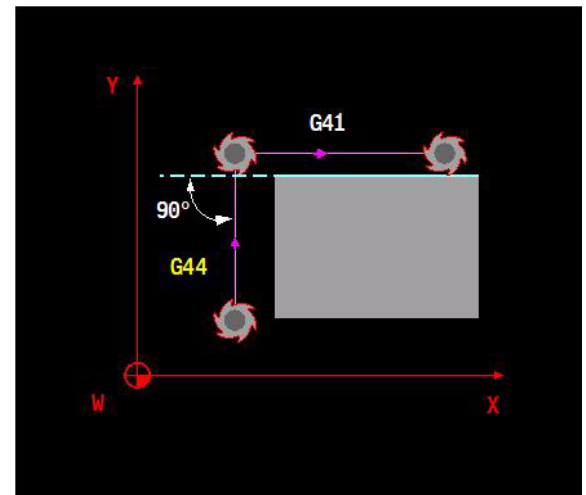
Switching from one radius compensation function to another

If, for example, a switch is made from G44 to G41, G42, or G43, then the tool movement finishes at a position that was calculated with G44 and starts at a position that was calculated with G41, G42, or G43. If the two positions do not coincide, a direct feed movement is made from one position to the other.

Example

Switching on radius compensation

Similar to G43.

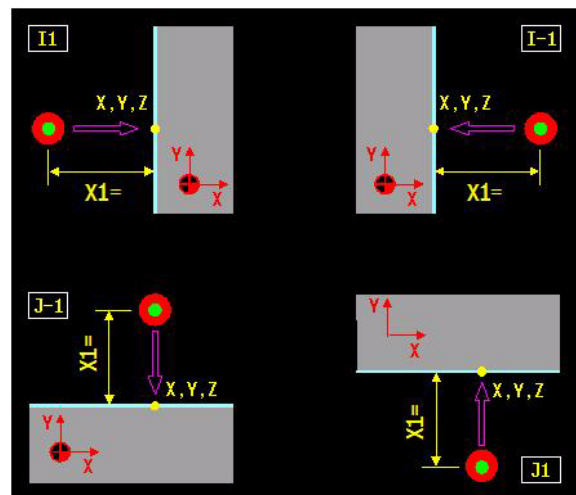


5.29 G45 Measuring a Point

Determination of coordinate values using the touch probe. The clamping status of the workpieces and the workpiece dimensions can be determined. The measurement results can be processed further with G49 or G50. The freely programmed measuring cycle G145-G150 can be used as an alternative to G45.



The G45 function only operates on a paraxial basis. G145 has an expanded functionality and cannot measure parallel to an axis. We therefore recommend that you use the basic measuring movement G145.



Address description

- ▶ X, Y, Z measurement target coordinates
- ▶ B, C measurement target angles
- ▶ I measurement direction for X axis
- ▶ J measurement direction for Y axis
- ▶ K measurement direction for Z axis
- ▶ L measurement direction rotary axis
- ▶ E parameter no. measured coordinate
- ▶ N= point no. for measured coordinate
- ▶ X1= measurement path length
- ▶ ?90= abs. measurement target angle (X,Y,Z..)
- ▶ ?91= incr. measurement target angle (X,Y,Z..)
- ▶ P1= point definition number

Format

G45 [measuring position] {I+/-1} {J+/-1} {K+/-1} {L+/-1} {X1=...} {N=...}
{P1=...}

The plane for the rotary table is determined by the definition of the 4th axis. This must be configured as rotary axis B or C. L refers to the 4th axis B or C. Rotary axis A is not allowed.

Application

Tool table

The touch probe must be entered in the tool table as a touch probe (T_P).

Switching touch probe on/off

The touch probe is switched on/off with the following functions:

- M27 activate touch probe.
- M28 deactivate touch probe.

Measuring position

Position A which is to be measured (see figure) is entered using the measuring point coordinates. The pre-measurement distance $X1=$ defines the measuring range before the measuring point. The specified pre-measuring distance is used if $X1$ is not programmed.

The specified pre-measuring distance (SECU) and the total measuring distance (DIST) are stored in the tool table group "touch probe",

Saving measurement results

The measured coordinates can be stored in the E parameter (E) and/or in the points table (N=).

The difference between the measured and the programmed coordinates is calculated and saved internally for use when operating with G49 or G50.

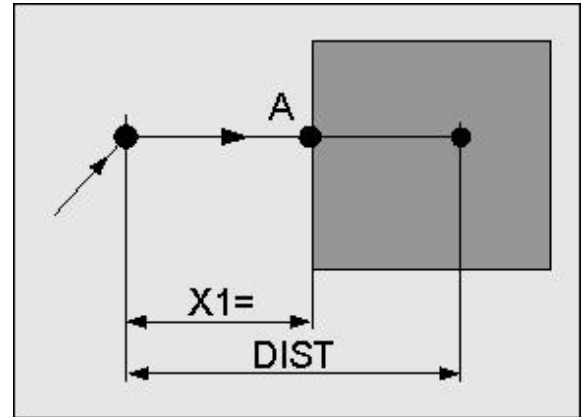
The saved measurement differences are deleted by a new measuring function (G45 or G46), Cancel program, or CNC reset.

Restrictions

- A G45 block can only be used to measure an axis coordinate
- In the tool axis, measurements can only be taken in a negative direction.

Procedure

The touch probe moves to the pre-measurement position in rapid traverse; this position is defined by the programmed position and the pre-measurement distance in the axis to be measured. This movement is executed with positioning logic. Once the touch probe has reached the pre-measurement position, it travels along the specified axis in the programmed direction and at the probe feed rate until it reaches the programmed position. When the touch probe makes contact with the workpiece, the measured coordinate is saved. The touch probe then returns to the pre-measurement position in rapid traverse.



Example

Measuring a point in the X-axis

```
G45 X0 Y20 Z-10 I1 E1 N=1
```

```
G45 X60 Y20 Z-10 I-1 E1 N=1
```

- G45 Measure in a positive direction
 Measure the point, calculate the measuring position,
 save in point table N= or in parameter E1
- G45 Measure in a negative direction

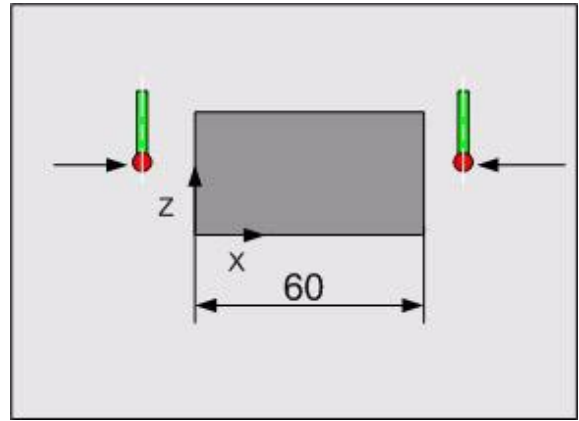
Measuring tool dimensions G45 + M25

For measuring tool dimensions using a touch probe with a cubical measuring tip at a fixed point.

Format: G45 {I+/-1} {J+/-1} {K+/-1} {X1=...} M25

Measuring in the tool axis gives the tool length. Measuring in 2 directions in the same axis gives the tool radius.

The procedure is similar to measuring a point with G45. Instead of programming the measuring point coordinates, the coordinates for the fixed touch probe are queried in the configuration data.



5.30 G46 Measuring a Circle

Measuring of a circle (inside or outside) using 4-point measurement.
The measurement results can be processed further with G49 or G50.

Address description

- ▶ X, Y, Z center point coordinate
- ▶ B, C measurement target angles
- ▶ I measurement direction for X axis
- ▶ J measurement direction for Y axis
- ▶ R circle radius
- ▶ E parameter no. measured radius
- ▶ N= point no. measured center point
- ▶ X1= measurement path length
- ▶ ?90= center point abs. (X,Y,Z..)
- ▶ ?91= center point incr. (X,Y,Z..)
- ▶ P1= point definition number

Format

Measuring the inside circle:

- G46 [circle center point coordinates] R... {I+1 J+1} {I+1 K+1} {J+1 K+1} {F...} {X1=...} {P1=...} N=... E...

Measuring the outside circle:

- G46 [circle center point coordinates] R... {I-1 J-1} {I-1 K-1} {J-1 K-1} {F...} {X1=...} {P1=...} N=... E...

Application

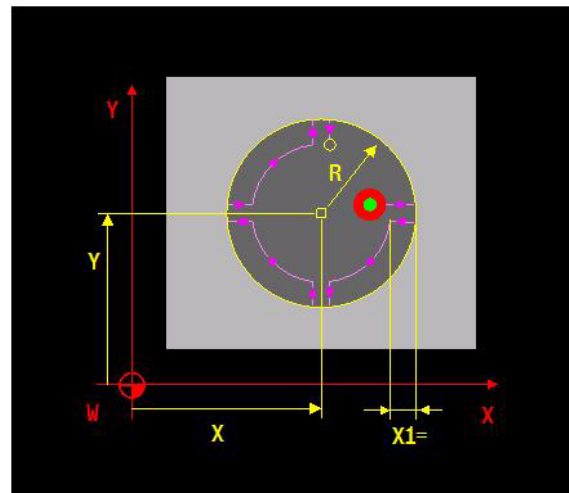
Tool table

The touch probe must be entered in the tool table as a touch probe (T_P).

Switching touch probe on/off

The touch probe is switched on/off with the following functions:

- M27 activate touch probe.
- M28 deactivate touch probe.



Measuring an inside or outside circle

The algebraic sign for the addresses I, J, K defines the type of circle to be measured. A pair of addresses must be specified in each G46 block, depending on the plane.

Plane	Inside circle		Outside circle	
XY (G17)	I+1	J+1	I-1	J-1
XZ (G18)	I+1	K+1	I-1	K-1
YZ (G19)	J+1	K+1	J-1	K-1

Measuring positions

Four positions are measured when a G46 block is executed. The measurements are taken as if four G45 blocks were programmed. The pre-measurement distance X1= defines the measuring range before each programmed position. The specified pre-measuring distance is used if X1 is not programmed.

The specified pre-measuring distance (SECU) and the total measuring distance (DIST) are stored in the tool table group "touch probe",

Procedure

The touch probe moves to the pre-measurement position for the first point to be measured in rapid traverse. This position is defined by the programmed circle center point, the programmed radius, and the pre-measurement distance. This movement is executed with positioning logic. Once the touch probe has reached the pre-measurement position, it travels at the probe feed rate to the first point on the programmed circle. The probe can travel past this point but it must respond within the range of the measuring distance. This automatically saves the measuring position. The touch probe then returns to the initial position with rapid traverse, before travelling clockwise around the circle at the programmed feed rate until it reaches the second pre-measurement position. This process is repeated for the second, third, and fourth positions. Once the fourth position has been measured, the four measured points are used to calculate the circle center point and the radius. The coordinates for the circle center point are stored in the points table, while the radius is stored in the E parameter table.

G46 + M26 Calibrating the touch probe

The touch probe radius is determined by probing the calibration ring. The control calculates the probe radius on the basis of the measured radius of the calibration ring and the programmed radius. The new radius value is stored in the tool table.

The center point coordinates and the radius of the calibration ring are entered in the machine configuration.

Format

Measuring the inside ring gauge:

■ G46 {I+1 J+1} {I+1 K+1} {J+1 K+1} {F...} {X1=...} M26

Measuring the outside ring gage:

■ G46 {I-1 J-1} {I-1 K-1} {J-1 K-1} {F...} {X1=...} M26

Example

Measuring an inside- and outside circle in the XY-plane

G46 X30 Y25 Z20 I+1 J+1 R12.5 F3000 N=59 E24
G46 X30 Y25 Z20 I-1 J-1 R20 F3000 N=58 E23

G46	Inside circle: Measure the circle, store the center point in the points table N=59 and the radii in the parameter table E24.
G46	Outside circle

Calibrating the touch probe

D207 M19	Defined spindle stop
G46 I1 J1 M26	Calibrate the touch probe, store the touch probe radius for T1 in the tool table

5.31 G49 Checking on Tolerances

Comparison between the programmed value and the measuring value determined during G45 or G46 operation to establish whether the difference falls within specified dimensional tolerance limits.

Address description

- ▶ X, Y, Z positive tolerance value in X, Y, Z
- ▶ B, C positive tolerance value in B, C
- ▶ R positive tolerance circle radius
- ▶ N= jump to block number
- ▶ N1= repeater begin block
- ▶ N2= repeater end block
- ▶ X1= Y1= Z1= negative tolerance value in X, Y, Z
- ▶ B1= C1= negative tolerance value in B, C
- ▶ R1= negative tolerance circle radius

Format

If the difference falls within the tolerance limits, then program execution continues with the block after G49.

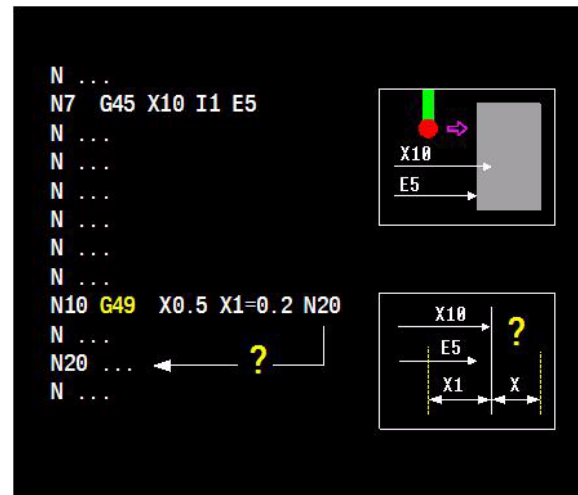
If the difference falls outside the tolerance limits, the following options apply:

Program section repeat:

- G49 {X..., X1=...} {Y..., Y1=...} {Z..., Z1=...} {B..., B1=...} {C..., C1=...} {R..., R1=...} N1=... N2=...

Program jump:

- G49 {X..., X1=...} {Y..., Y1=...} {Z..., Z1=...} {B..., B1=...} {C..., C1=...} {R..., R1=...} N=...



Application

Measuring point

The measuring point must fall between the upper limit (X,Y,Z,A,B,C) and the lower limit (X1=,Y1=,Z1=,A1=,B1=,C1=) of the tolerance range.

Program section repeat

The addresses N1= and N2= are used to repeat a program section if a specific tolerance value has been exceeded.

The block numbers N1= and N2= must both be contained in the same part program or macro. If N2= is not programmed, only the block specified with N1= is repeated once.

Error message

MillPlus issues an error message if the measuring difference has exceeded a specific limit or is not available. MillPlus also issues an error if no program repeat or a jump have been programmed.

Jump

The address N= is used to specify a jump if a specific limit is exceeded. The jump is executed once. The address N= is used to specify the block number in the same main program or macro that the jump is made to.

Changes to V5xx

■ See "G49_E" on page 503

Example

Tolerance comparison

```
G49 R.02 R1=2 N=13
G49 R2 R1=,02 N1=1 N2=6
```

- G49 1. Tolerance comparison:
A jump is made to block N13 if the upper tolerance limit (R0.02) is exceeded (hole too large). The lower tolerance limit must not be reached. (Jump)
- G49 2. Tolerance comparison:
If the lower tolerance limit (R1=0.02) is exceeded (hole too small), the program section between N1 and N6 is repeated. The upper tolerance limit must not be reached. (Program section repeat)

5.32 G50 Processing Measuring Results

Changing of the zero point shifts or tool dimensions depending on the compensation values derived from the recorded difference values.

Address description

- ▶ X, Y, Z 1=zero point shift in X, Y, Z
- ▶ B, C 1=zero point shift in B, C
- ▶ I multiplication factor for X
- ▶ J multiplication factor for Y
- ▶ K multiplication factor for Z
- ▶ L multipl. factor for rotary axis
- ▶ T tool dimensions to be corrected
- ▶ N= offset no. for correction (52-59)
- ▶ X1= multiplication factor for tool radius
- ▶ B1= prog. angle in B after calculation
- ▶ C1= prog. angle in C after calculation
- ▶ L1= 1=correction of tool length
- ▶ R1= 1=correction of tool radius

Format

Compensating zero point shift G52, G54 to G59:

- G50 {X1} {I...} {Y1} {J...} {Z1} {K...} {B1} {C1} {C2} {B1=} {C1=} {L...}
- N=.**(52, 54 to 59)

Compensating zero point shift G54:

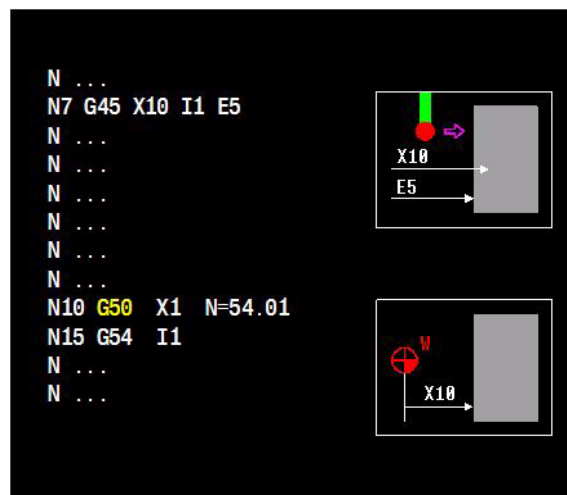
- G50 {X1} {I...} {Y1} {J...} {Z1} {K...} {B1} {C1} {C2} {B1=} {C1=} {L...}
- N=54.**(54.00 to 54.99)

Compensating tool length:

- G50 T... L1=1 {I...} {J...} {K...} {T2=...}

Compensating tool radius:

- G50 T... R1=1 {X1=...} {T2=...}



Application

Compensating shift values

G50 N= allows new shift values derived from the compensation values recorded by G45 or G46 to be stored in the zero point table.

X1, Y1, or Z1 are used to specify which linear axis is to be compensated in the zero point table.

I, J, K, or L is used to multiply the compensation value for the shift by a positive or negative factor. If no factor is specified, the fixed value +1 is used

Compensating tool dimensions

G50 T... allows new tool dimensions derived from the compensation values recorded by G45 or G46 to be stored in the tool table.

X1= is used to multiply the compensation value of the tool radius by a factor.

I, J, or K are used to multiply the compensation value (in the G19, G18, or G17 plane) for the tool length by a factor.

The compensation factor can be positive or negative. If no factor is specified, the fixed value +1 is used

Machine configurations (B1,C1,C2)

B axis B1:

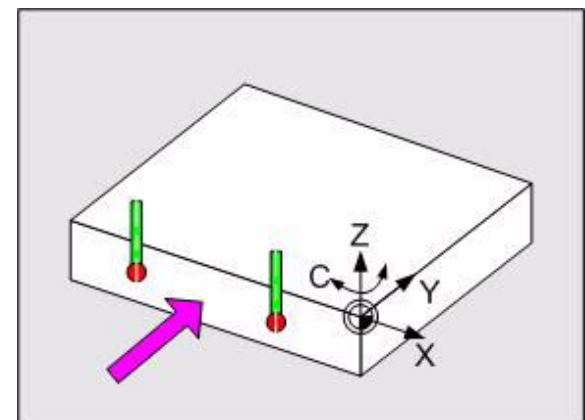
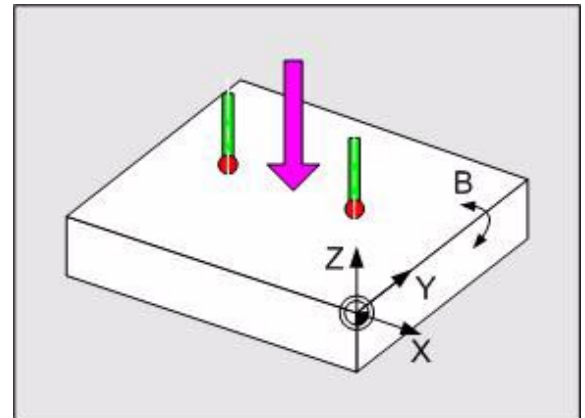
Measuring two points on the X axis is sufficient to align a clamped workpiece on a rotary table (B axis) rotating around the Y axis:

- The angle of rotation is relative to the X axis
- The workpiece rotates around the Y-axis
- The tool axis with the touch probe is the Z-axis or Y axis (see figure).

C axis C1:

Measuring two points on the X axis is sufficient to align a clamped workpiece on a rotary table (C axis) rotating around the Z axis:

- The angle of rotation is relative to the X axis
- The workpiece rotates around the Z-axis
- The tool axis with the touch probe is the Z-axis (see figure).



C axis C2:

This is an enhanced option for C1: (see figure)

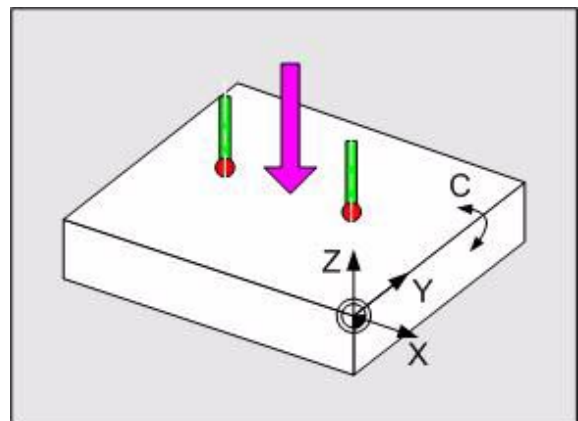
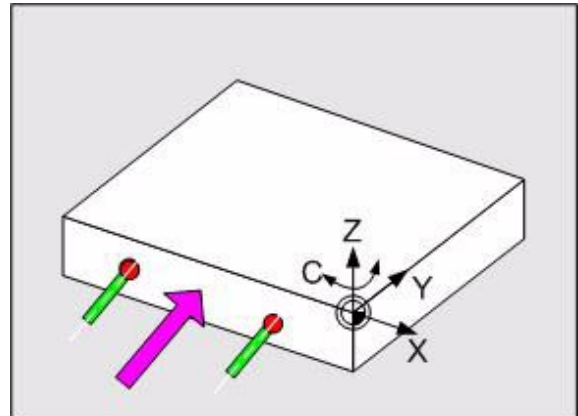
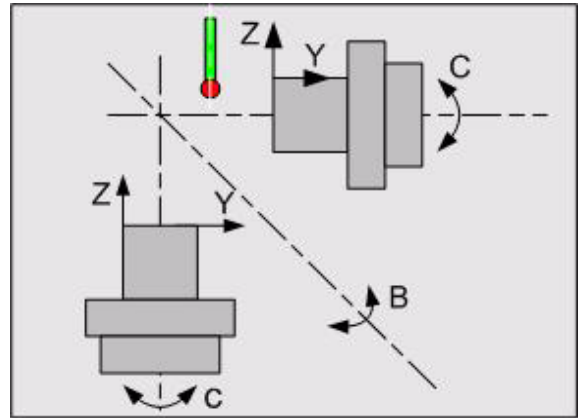
First option: the C axis is rotated by 90 degrees and rotates around the Y axis instead of the Z axis. Measuring two points on the X axis is sufficient to align a clamped workpiece on a rotary table (C axis) rotating around the Y axis:

- The angle of rotation is relative to the X axis
- The workpiece rotates around the X-axis
- The tool axis with the touch probe is in the Z-axis (see figure).

Second option:

Measuring two points on the X axis is sufficient to align a clamped workpiece on a rotary table (C axis) rotating around the Z axis:

- The angle of rotation is relative to the X axis.
- The workpiece rotates around the X-axis
- The tool axis with the touch probe is in the Y-axis. (see figure)



Example

Example 1

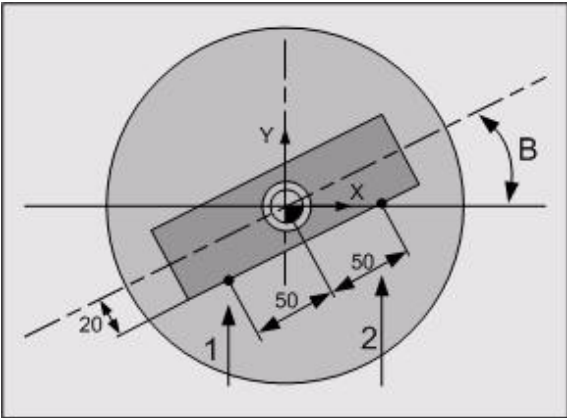
```
G50 X1 I0.8 N=54
G50 T5 L1=1 K0.97 R1=1
```

- G50 Compensate the X-coordinate of the G54-shift with the compensation value multiplied by 0.8 and store in the zero point table.
- G50 Compensate the length of tool 5 with the difference in Z (tool in Z-axis) multiplied by 0.97 and store in the tool table.

Example 2 (see figure)

```
G45 X-50 Z0 Y-20 C0 J1 N=1
G45 X50 Z0 Y-20 J1 N=2
G50 C1 N=54
G54
```

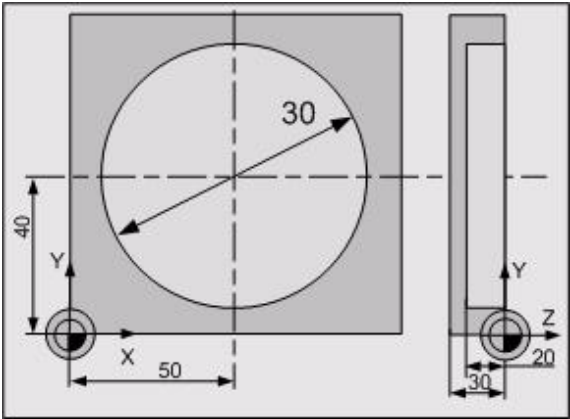
- G45 Measurement at point 1
- G45 Measurement at point 2
- G50 Calculate zero point shift
- G54 Activate zero point shift again



Example 3 (see figure)

T31 M67
M27
G46 X50 Y40 Z-5 R15 I1 J1 F500 E5
G49 R0.02 R1=2 N=21 E5
G49 R2 R1=.02 N=17 E5
G29 E10 E10=1 N=23
N17 G50 T1 R1=1
M28
N21 M0
N22 (HOLE OUTSIDE TOLERANCE RANGE)
N23 M30

T31	Touch probe
M27	Activate touch probe
G46	Measure a full circle
G49	Hole > (15+0.02) jump to N=21 Tolerance comparison
G49	Hole < (15-0.02) jump to N=17 Tolerance comparison
G29	Conditional jump to end of program
N17	Calculate tool radius
M28	Deactivate touch probe
G21	
N22	Error message



5.33 G51 Cancel Pallet Zero Point Shift

Cancellation of pallet zero point shift, activated by G52.

Address description

No specific addresses.

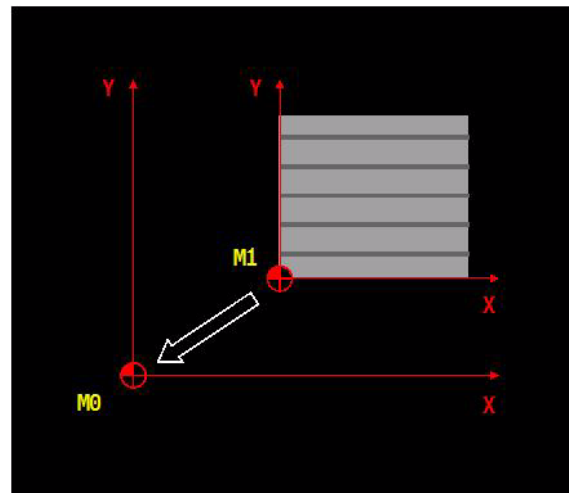
Application

Modality

G51 is modal with G52.

Associated functions

G52, G52 I[no.], G53, G54... G59, G54 I[no.], G92, G93.



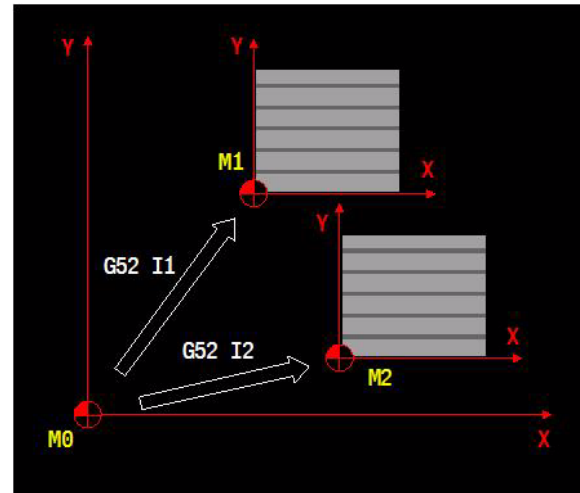
5.34 G52 Activate Pallet Zero Point Shift

Activation of the pallet zero point shift at a position. The coordinate values of several pallet zero points can be entered into the pallet zero point table.



Pallet zero points are used for automation purposes, e.g. pallet control. These zero points are then activated by the PLC program using G52 I, where xx corresponds to the pallet zero point.

In the NC program, the selected zero point can be switched off using G51 and switched back on using G52. The program is thus independent of the pallet number.



Address description

► **I zero point index** Index number of the zero point to be activated.

Format

Activating pallet zero point shift:

- G52 (activate NP value in G52 I0) or (activate an individual pallet zero point)
- G52 I[no.] (activate pallet zero point Ixx and copy to I0).

Default setting

The modal function G52 (Ixx) is deleted by G51 or CNC reset.

G52 remains active after Cancel program, M30, or Switch CNC on/off.

Application

Modality

G52 is modal with G51.

Associated functions

G51, G53, G54... G59, G54 I[no.], G92, G93, G149, G150

Number of zero points

The maximum number of zero points in the table (*.POT) is determined by a configuration value. ($0 \leq \text{value} \leq 99$).



If the configuration value is set to zero, the table (*.POT) is reduced to one block.
All entered values are then deleted.
In this case, no index lxx can be programmed either

Activating a pallet zero point

When changing pallets (M60/M61), the PLC can activate G52 lxx using a machine macro.

Note: G52 lxx can also be activated in the part program. During activation, the active zero point shift is copied into G52 I0.

Machine zero points

If a tool machine has several pallets or tables, then information is required from several zero points. The zero points always refer to the geometric machine zero point (M0). The distances in the axes, measured from the zero point M0, indicate the position of these zero points and are entered in the pallet zero point table.

G54 lxx or G54 to G59 zero point shift

G52 does not affect the functions G54 (lxx) or G54 to G59. If G52 is active, G54 (lxx) takes effect from this shift.

Absolute/incremental zero point shifts G92/G93

A programmed zero point shift (G92 or G93) is deleted by G52 (lxx).

Increasing/decreasing, mirroring, and axis rotation (G73, G92/G93)

G52 (lxx) can be used in a program section to be increased/decreased, mirrored, or rotated. The zero point shift occurs in the coordinate system of the tool machine and is not affected by the programmed coordinate change.

5.35 G53 Cancel G54-G59 Zero Point Shift

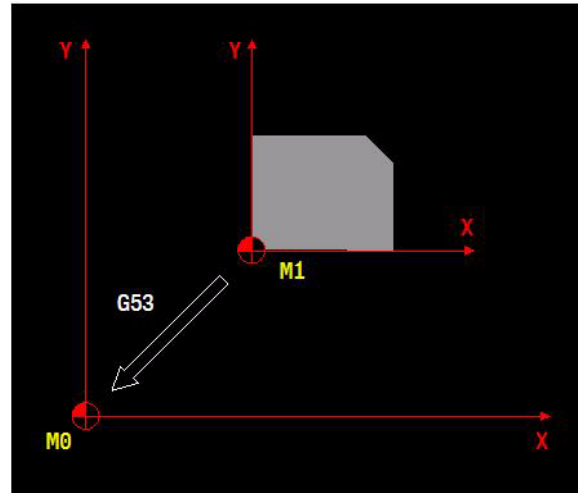
Cancellation of the workpiece zero point shift G54 lxx or G54 to G59.

Application

A zero point shift (G54 lxx or G54 - G59) is canceled by G53.

A pallet zero point (G52) is not canceled by G53.

A program zero point shift (G92 - G93) is canceled by G53.



5.36 G54 - G59 Activate Zero Point Shift

Movement of the workpiece zero point to a new position, whose coordinate values are saved in the zero point table (under the relevant number) or programmed in the same block.

Two different zero point tables are available:

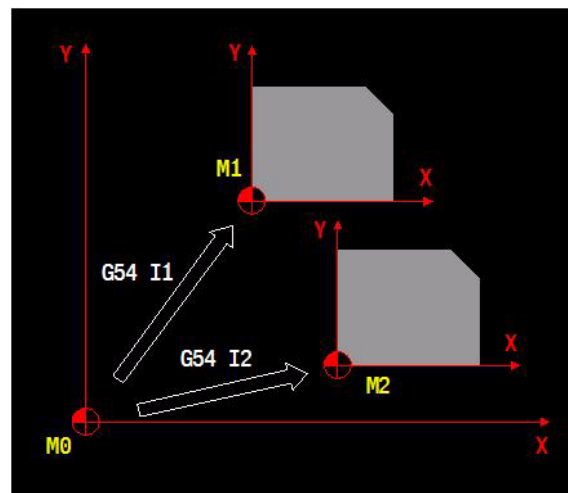
- Zero point table G54 lxx (identification *.ZET) with a maximum of 99 zero point shifts.
- Zero point table G54 - G59 (identification *.ZOT) with a maximum of 8 zero point shifts.

Available functions:

- Programming (offset values) the zero point shift in the program
- Programming an angle of rotation (B4=) in the zero point shift (only for G54 lxx)
- Entering comments in the zero point table (only for G54 lxx)

Address description

- **X, Y, Z zero point coordinates**
- **A, B, C zero point angles**
Addresses that are only available for G54:
- **I zero point index** Index number of the zero point to be activated.
- **B4= angle of rotation absolute** The coordinate system is rotated by the angle B4=.



Format

Zero point table G54 lxx

Define and call zero point shift:

■ G54 l[no.] {X...} {Y...} {Z...} {A...} {B...} {C...} {B4=...}

Call zero point shift:

■ G54 l[no.]

Zero point table G54 - G59

Define and call zero point shift:

■ G54 {X...} {Y...} {Z...} {A...} {B...} {C...}

Call zero point shift:

■ G54

Application

Modality

The functions G53 and G54 lxx or G53 to G59 form a modal group.

Associated functions

G51, G52, G53, G92, G93, G149, G150

Default setting

The functions G54 to G59 are deleted with CNC reset or by programming G53.

The functions G54 lxx and G54 to G59 remain active after Cancel program and M30.

Zero point table

The active zero point table (*.ZET or *.ZOT) is specified in the configuration data (file OEMTABLE.CFG). The zero points can be edited in both tables.



The table is changed (*.ZET <-> *.ZOT) if the configuration value for the zero point table type is changed. The new zero point table is initialized to zero.

There are 2 options for entering the shift values in the zero point table:

- 1 The values of the zero point shifts G54 l[no] or G54 to G59 are entered in the zero point table via the control panel or from a data carrier before the program is executed.

- 2 The values of the zero point shift G54 I[no] X... Y... Z... A... B... C... B4=... or G54 to G59 X... Y... Z... A... B... C... are programmed in an NC program block. The programmed values are transferred to the zero point table and activated when the program is executed.



Attention - danger of collision

If zero point shift values are not programmed in the program block for all axes with G54 [no.] or G54 to G59, then the zero point shift values already in the table are used for the axes that are not programmed.

This means that the zero point shift values that are not programmed are not deleted from the table.

If the active zero point shift is changed in the zero point table in a program run or after M30, then this changed zero point shift is activated immediately.

Additional functionality G54 lxx (*.ZOT)

A comment can be entered in the table for each zero point shift.

Each zero point shift in the table can involve an axis rotation. The shift is carried out first and the coordinate system is then rotated by the angle B4=.

G52 I(xx) pallet zero point shift

G54 (lxx) or G54 to G59 do not affect the function G52 (lxx). If G52 is active, G54 (lxx) takes effect from this shift.

Absolute/incremental zero point shifts G92/G93

A programmed zero point shift(G92 or G93) is deleted by G54 I[no.] or G54 to G59.

Increasing/decreasing, mirroring, and axis rotation (G73, G92/G93

G54 I[no.] can be used in a program section to be increased/decreased, mirrored, or rotated. The zero point shift occurs in the coordinate system of the tool machine and is not affected by the programmed coordinate change.

Changes to V5xx

- See "G54_G41" on page 504

Example

Example G54 to G59 (see figure)

G54

G55

G54 Activate zero point shift G54

G55 Activate zero point shift G55; the coordinates refer to the new zero point.

Example G54 (see figure)

G54 I1

G54 I2

G53

G54 Selection of zero point W1. Its coordinates (X40,Y100,Z300) are taken from the zero point table. All programmed coordinates are measured starting from W1.

G54 Selection of zero point W2. Its coordinates (X200,Y100,Z100) are taken from the zero point table. Zero point W1 is deleted and W2 is activated. All programmed coordinates are then measured starting from W2.

G53 Deactivation of zero point W2. The coordinates (X0,Y0,Z0) are taken from the G53 zero point table. Zero point W2 is deleted and M is activated. All programmed coordinates are then measured starting from M.

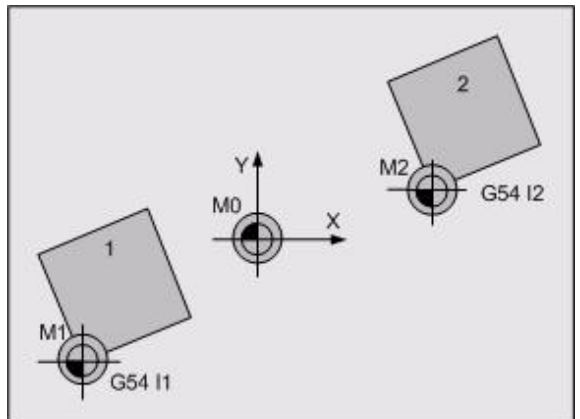
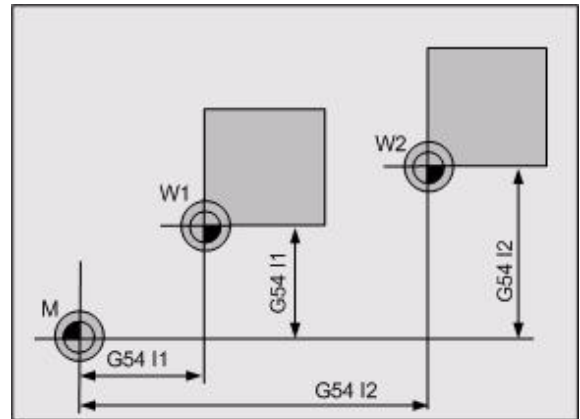
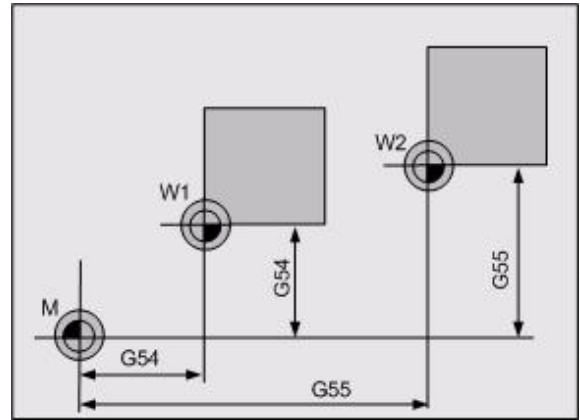
Axis rotation (see figure)

G54 I1 X-42 Y-15 B4=14 (Z0 C0)

G54 I2 X10 Y24 B4=-17

M27 Entry in the zero point table and call:
The zero point shifts are entered in the zero point table.
Machine workpiece 1; all programmed coordinates are measured starting from M1.

G46 Machine workpiece 2; all programmed coordinates are measured starting from M2.



5.37 G61 Tangential Approach

Programming of a tangential approach movement for a contour between a starting point and the starting point of the contour.

Address description

- ▶ **X, Y, Z end point tangential approach** The programmed end point for G61 is the starting point for the contour.
- ▶ **P1 point definition number**
- ▶ **R radius** The programmed approach circle radius is the radius for the tool center point path, i.e. without tool radius compensation.
- ▶ **X1=, Y1=, Z1= auxiliary point in X, Y, Z** The auxiliary point can be programmed in the tool axis. In G17 with the Z1= address, in G18 with Y1=, and in G19 with X1=.
- ▶ **B2= polar angle** The end point can also be programmed on a polar absolute basis. The tool axis cannot be programmed in this case.
- ▶ **L2= polar length** The end point can also be programmed on a polar absolute basis. The tool axis cannot be programmed in this case.
- ▶ **I2= tangential approach definition** Approach movement to the end point (contour starting point).
 - I2=0 with arc and tangentially (default setting).
 - I2=1 with quarter circle and tangentially.
 - I2=2 with semi-circle and tangentially.
 - I2=3 with full circle and tangentially.
 - I2=4 with a line parallel to the contour and tangentially.
 - I2=5 perpendicular to the contour point
- ▶ **?90= end point abs. (X,Y,Z..)**
- ▶ **?91= end point incr. (X,Y,Z..)**

Format

Cartesian:

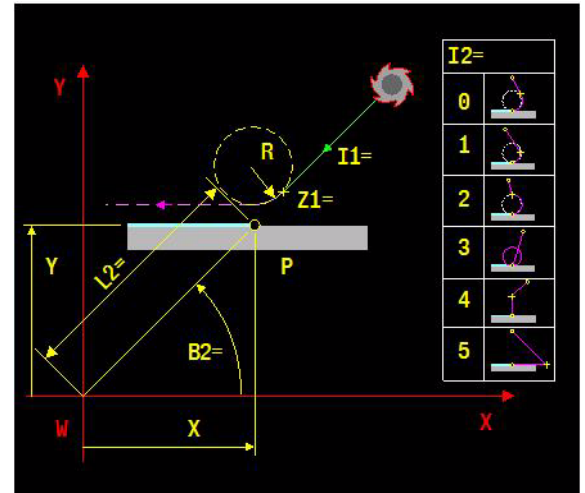
■ G61 {I2=...} X... Y... Z... R... [{X1=...} {Y1=...} {Z1=...}]

Point definition:

■ G61 {I2=...} P1=... R... [{X1=} {Y1=}] {Z1=}

Polar:

■ G61 {I2=...} B2=... L2=... R... [{X1=} {Y1=}] {Z1=}



Application

Approach movement

The approach movement follows the next movement in the working plane. Intermediate blocks without movement in the plane are ignored.

The approach movement consists of two parts: The first part is a feed movement to the (calculated) intermediate point of the approach movement. The second part is a tangential feed movement along the approach contour to the starting point of the contour.

If the distance between the current position and the approach circle is greater than the milling radius ($I2=0$), the approach movement consists of a line and an arc. If the distance between the current position and the approach circle is less than the milling radius, then $I2=0$ is changed to $I2=1$ and the approach movement becomes a quarter circle.

Intermediate point: the control calculates an intermediate point based on the starting point, the type of approach movement, and the end point (starting point of the contour)

Radius compensation

The approach side is determined by the active function G41/G42.

The radius compensation (G41/G42) must be activated immediately before the G61-block.

Restrictions

G61 is not allowed during operation of G64, G182-, and MDI-.

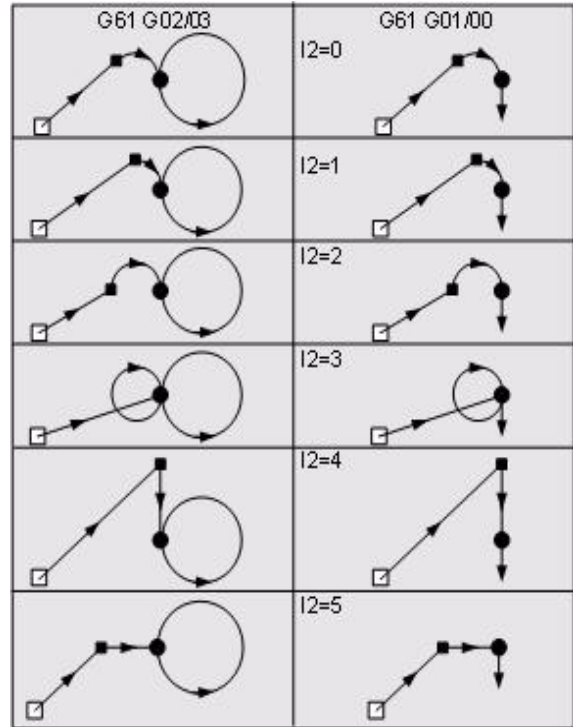
Specific restrictions apply for the blocks immediately after the approach movement (G61). Only following functions G64, G0, G1, G2, G3 with movements in the working plane are allowed.

Rotary axis positions must not be programmed during G61.

G1 does not take effect automatically if no G-function has been programmed after the G61-block. The last movement of the G61 function can be G1, G2, or G3.

Changes to V5xx

- See "G61 und G62" on page 504.

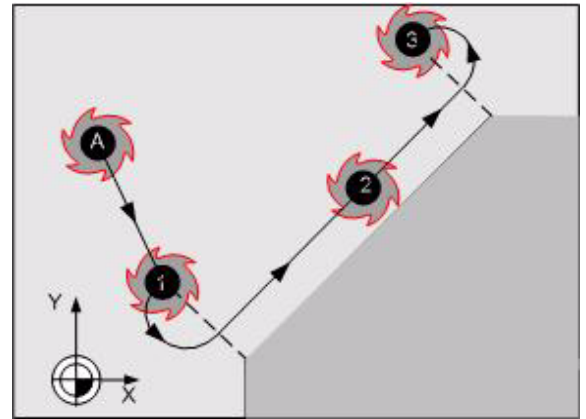


Example

Example (see figure)

G0 X0 Y0 Z30
G41
G61 I2=2 X20 Y20 Z-5 Z1=10 R5 F2=200
G64
G3 I20 J50 R1=0
G1 X60 Y60
G63
G62 I2=2 Z1=10
G40

T31	Advance to starting position. (Position 1: X0 Y0 Z30)
G41	Radius compensation, left.
G61	Tangential approach movement (I2=2) with semi-circle. The first part of the approach movement is a feed movement with positioning logic to the starting point of the semi-circle (position 2: X... Y... Z10). The radius compensation is started during the linear movement to the semi-circle. The arc is executed as a helix. The contour starts at position X20 Y20 Z0 (position 3: X20 Y25 Z-5).
G64	
G3	
G1	
G63	
G62	Tangential exit movement (I2=2) with semi-circle. The semi-circle is executed as a helix. The starting height for the Z axis is -5, the end height is 10. (Position 5: X... Y... Z10).
G40	



5.38 G62 Tangential Exit

Programming of a tangential exit movement at the end point of the contour.

See also the description for G61.

Address description

- ▶ **X, Y, Z, end point tangential exit** The end point for G62 can only be programmed in the case of a tangential exit with an arc (I2=0).
- ▶ **P1 point definition number**
- ▶ **R radius** The programmed exit circle radius is the radius for the tool center point path, i.e. without tool radius compensation.
- ▶ **X1=, Y1=, Z1= auxiliary point in X, Y, Z** The auxiliary point can be programmed in the tool axis. In G17 with the Z1= address, in G18 with Y1=, and in G19 with X1=.
- ▶ **B2= polar angle** The end point can also be programmed on a polar absolute basis. (Only for I2=0).
- ▶ **L2= polar length** The end point can also be programmed on a polar absolute basis. (Only for I2=0).
- ▶ **I2= Tangential exit definition**
Exit movement to the auxiliary point:
 - I2=0 with arc and tangentially.
 - I2=1 with quarter circle and tangentially.
 - I2=2 with semi-circle and tangentially.
 - I2=3 with full circle and tangentially.
 - I2=4 with a line parallel to the contour and tangentially.
 - I2=5 perpendicular.
- ▶ **?90= end point abs. (X,Y,Z..)**
- ▶ **?91= end point incr. (X,Y,Z..)**

Format

Intermediate point equal to end point:

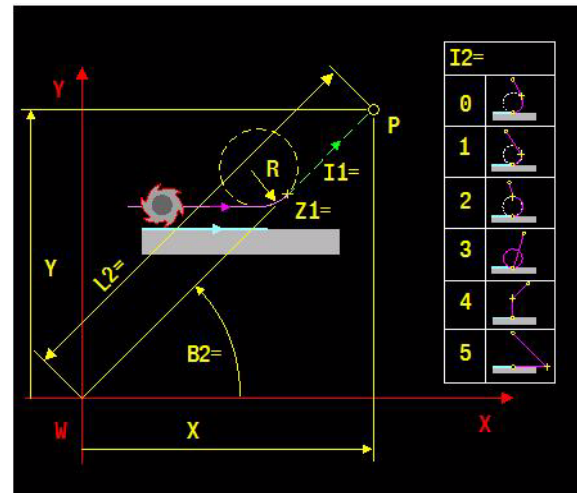
- G62 I2>0 Z1=... R... {F2=}

With arc, Cartesian:

- G62 I2=0 X... Y... Z... Z1=... R...

With arc, polar:

- G62 I2=0 B2=... L2=... Z... R...



Application

Exit movement

The control calculates an intermediate point based on the starting point (the end point of the contour), the type of exit movement, and the end point. The first movement is a tangential or perpendicular exit movement to the calculated intermediate point. Then another positioning is carried out with a feed to the programmed end point. The end point for G62 can only be programmed during a tangential exit with an arc (I2=0). The intermediate point is also the end point for exit movements programmed with I2=1 to I2=5. If the radius compensation with G40 is not canceled, then both the circle- and the linear movement are executed with radius compensation.

Radius compensation

Radius compensation is switched off in the G62-block. The movement to the calculated intermediate position is still carried out with radius compensation.

Restrictions

G62 is not allowed during operation of G64, G182-, and MDI-.

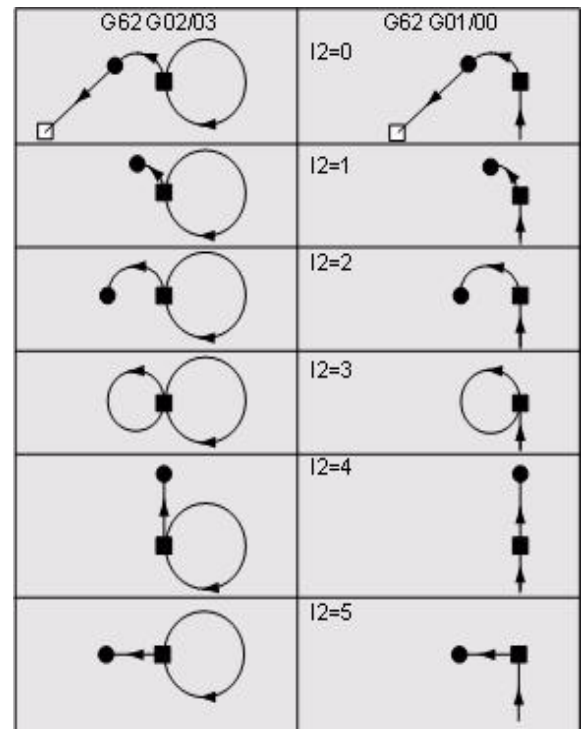
G1 is automatically takes effect if no G-function has been programmed after the G62-block.

Changes to V5xx

■ See "G61 und G62" on page 504.

Example

See example G61.



5.39 G63 Cancel Geometric Calculations

Cancellation of G64 geometric calculations and switch to programming of complete blocks.

Address description

No specific addresses.

Application

Modality

The functions G63 and G64 form a modal group.

Default setting

G63 automatically takes effect after:

- Control activation
- CNC reset
- Program cancellation
- M30.

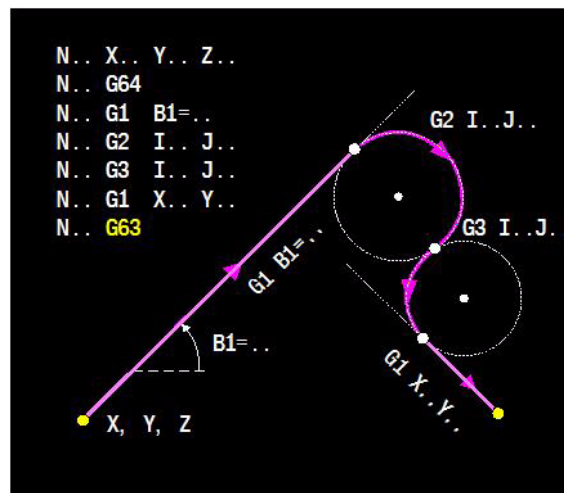
Programming

An absolute position must be programmed in the final block, before the geometry calculations are canceled with G63.

Complete blocks must be programmed after the G63 block.

Changes to V5xx

- See "G63 und G64" on page 506



5.40 G64 Activate Geometric Calculations

Activation of geometric calculations. A contour can be described between G64 and G63. The fact that straight-line and circular movements can be easily programmed allows the required calculations, e.g. for a point of intersection or a tangential point, to be left to the control.

Address description

See addresses in sections "Address description for straight line" on page 196 and "Address description for circle" on page 201.

Basic functions

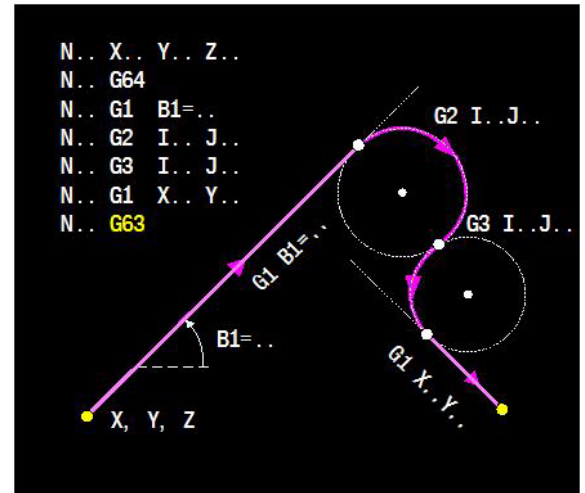
Fundamentals of geometry application

A minimum of two data blocks are always required where a calculation is necessary. Each block is programmed with the standard G functions for straight-line movements (G0 and G1) and circular movements (G2 and G3), as well as with specifications to define the straight lines or circles. The blocks do not have to contain all the data previously specified. Specific special words (indicator addresses) allow the control to calculate the missing data. The first block determines the position of the starting point. The second block provides the data for calculating the end point coordinates in the first block, e.g. as a tangential point or point of intersection of two elements. This end point is also the starting point for the second block.

The following elements can be inserted between these movements:

- A chamfer (between straight-line movements),
- A rounding arc (at the point of intersection of intersecting elements),
- A connecting circle (between elements that do not intersect or are not tangential)

It can happen that the data in the second block is insufficient to calculate the end point in the first block. In this case, the control attempts to calculate the end point for the second and first blocks from the subsequent blocks (maximum 32).



Format

- G64 activate geometric calculations
- G0, G1, G2, or G3 straight-line (G0/G1) and circular movements (G2/G3)
- G63 cancel geometric calculations

Application

Modality

G64 is modal with G63.

G codes that are allowed when G64 is active

G0-G1-G2-G3; G4; G14-G22-G29; G40-G41-G42-G43-G44; G94-G95.

G codes that are not allowed when G64 is active

- All G-codes that are not listed above
- Incremental programming (Cartesian and polar)
- Helical interpolation
- More than one defined point in the block
- M-functions M6, M66, and M67

Plane selection

The plane in which the geometry calculations are carried out is determined with G17 (XY-plane), G18 (XZ plane), or G19 (YZ plane). The definition of the angle B1= refers to the + X axis in the XY or the XZ plane or the – Z axis in the YZ plane.



See coordinate systems in the Programming chapter for an explanation of the possible coordinate systems (Cartesian, polar, absolute, and incremental) and definitions.

Macros

Geometry calculations can be used in the macro. All geometry blocks between G64 and G63 must be in the same macro.

Repeat function

Geometry calculations can be used in the repeat view of the part program (G14 or G29). All geometry blocks between G64 and G63 must be in the same program section.

Scaling, mirroring, and axis rotation

First, activation of scaling, mirroring, or axis rotation; the geometry calculations are then permitted.

Changes to V5xx

- See "G1, G41 und G64" on page 493.
- See "G63 und G64" on page 506.
- See "G2" on page 496.

Straight line

Address description for straight line

- ▶ X, Y, Z end point coordinates
- ▶ I chamfer length
- ▶ X1= Y1= end point of second element
- ▶ B1= angle
- ▶ B2= polar angle
- ▶ I1= parallel shift
- ▶ J1= 1=intersection left, 2=right
- ▶ L2= polar length
- ▶ P1= point definition number
- ▶ R1= R1=0 tangent to line



Both coordinates for the main plane should always be programmed. The tool axis must not be programmed. A combination of angle B1= and just one coordinate is not allowed. The angle B1 must match the direction of movement exactly.

Possible parameters for straight lines between G64 and G63 blocks.



See coordinate systems in the Programming chapter for an explanation of the possible coordinate systems (Cartesian, polar, absolute, and incremental) and definitions.



Most examples consist of three blocks.

- 1) Movement to the initial position (if starting point not determined).
- 2) Incomplete block. The information in the previous and subsequent blocks completes this block.
- 3) Movement to the end position.

Addresses in the figures

A	Starting point
B1	Angle
E	End point
H	Auxiliary point
I1	Chamfer or distance
I2	Intersection point indicator
M	Center point
R	Radius

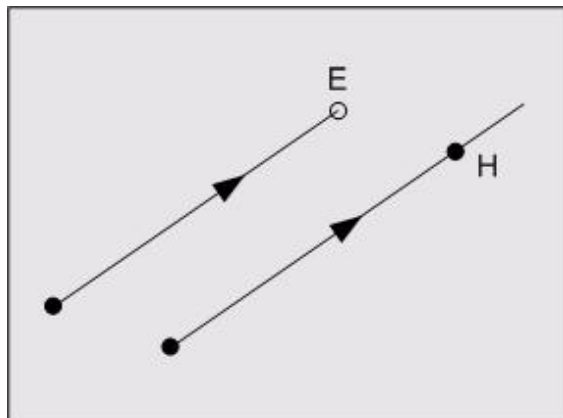
Straight line with end point or auxiliary point (see figure)

G1 X... Y... (G64 STRAIGHT LINE WITH END POINT OR AUXILIARY POINT)

OR

G1 L1=... B1=...

G1 Straight line with end point (E) or auxiliary point (H).
 An auxiliary point lies on the straight line but is not automatically the end point for the straight line.
 The next block can determine the end point.



Straight line with angle and end point or auxiliary point (see figure)

G1 B1=... X... Y... (G64 STRAIGHT LINE WITH ANGLE AND END POINT OR AUXILIARY POINT)

- G1 Straight line with angle (B1=) and end point (E) or auxiliary point (H).
The initial position must not be specified; otherwise the definition will be oversized.
In the case of oversizing, the angle is not taken into account.

Straight line with angle (see figure)

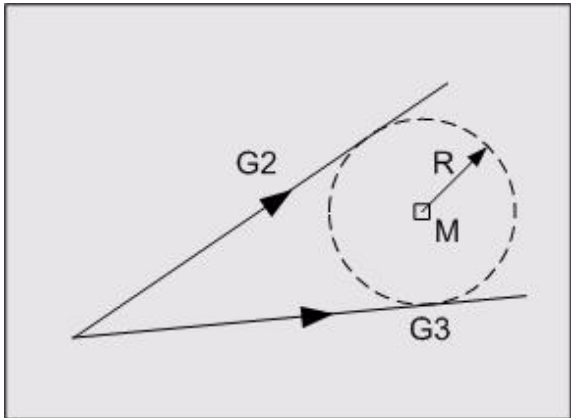
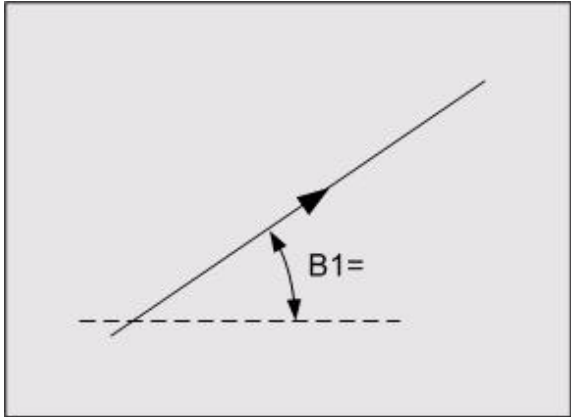
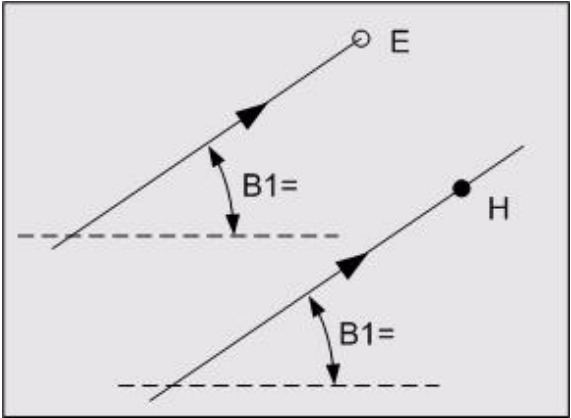
G1 X0 Y0
G1 B1=45 (G64 STRAIGHT LINE WITH ANGLE)
G1 B1=0 X100 Y50

- G1 Movement to the starting point.
G1 Straight line with only one angle.
The angle determines the direction.
The previous element determines the starting position.
G1 Horizontal straight line with one angle and end point.
This defines the end point for the previous straight line.

Straight line tangential to the circle (see figure)

G1 X0 Y0
G1 R1=0 (G64 STRAIGHT LINE TANGENTIAL TO THE CIRCLE)
G2 I50 J50 X60 Y50

- G1 Movement to the starting point.
G1 Straight line is tangential to the circle (indicated by R1=0).
The following element must be a circle.
G2 Circle defined with a center point (M) and an end point. The side where the straight line is tangential to the circle is determined by the direction of rotation of the circle (G2 or G3).
The default setting is that a flowing movement takes place.



Straight line with an angle tangential to the circle (see figure)

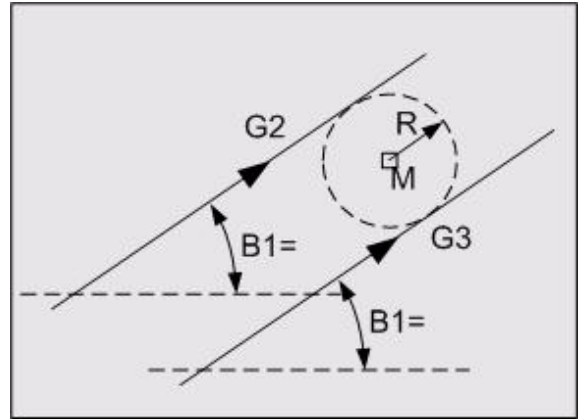
G1 X0 Y0

G1 B1=0

G1 B1=45 R1=0 (G64 STRAIGHT LINE WITH AN ANGLE TANGENTIAL TO THE CIRCLE)

G2 I50 J50 X60 Y0

- G1 Movement to the starting point.
- G1 Horizontal straight line.
- G1 Straight line with an angle (B1=) tangential to the circle (indicated by R1=0).
The following element must be a circle.
- G2 Circle defined with a center point (M) and an end point. The side where the straight line is tangential to the circle is determined by the direction of rotation of the circle (G2 or G3).
The default setting is that a flowing movement takes place.



Straight line parallel to a straight line through auxiliary point and angle (see figure)

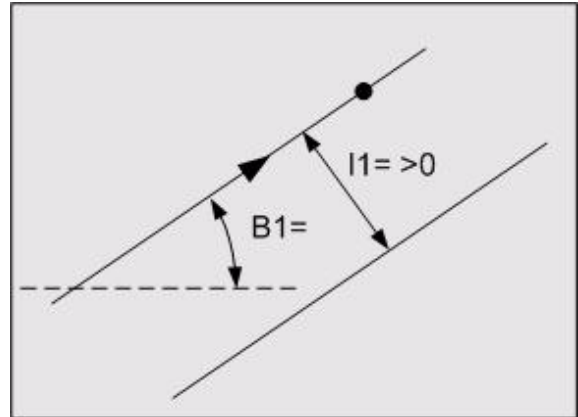
G1 X0 Y0

G1 B1=90

G1 X0 Y0 B1=45 I1=20 (G64 STRAIGHT LINE PARALLEL TO STRAIGHT LINE THROUGH AUXILIARY POINT AND ANGLE)

G1 X0 Y100 B1=-90

- G1 Movement to the starting point.
- G1 Vertical straight line
- G1 Parallel straight line through X0, Y0, 45 degree angle
The distance (I1=) is 20 mm.
Since the distance is positive, the parallel straight line is on the right-hand side.
- G1 Vertical movement to the end point.



Straight line is tangential to the previous circle (see figure)

G1 X50 Y50

G2 I50 J40 R1=0

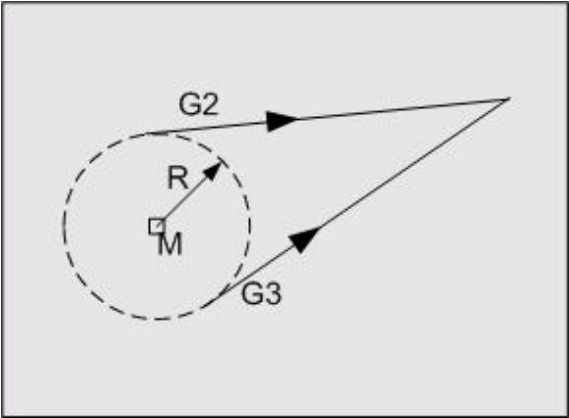
G1 X0 Y0 (G64 STRAIGHT LINE IS TANGENTIAL TO THE PREVIOUS CIRCLE)

- G1

Movement to the starting point on the circle.
- G2

Circle defined with a center point (M). This circle is tangential to the next element (indicated by R1=0).
- G1

Straight line to the end point. The side where the straight line is tangential to the circle is determined by the direction of rotation of the circle (G2 or G3). The default setting is that a flowing movement takes place



Chamfer

The chamfer is arranged symmetrically around the point of intersection.
The chamfer width is programmed with the I expression.

Chamfer between two straight lines (see figure)

G1....

I20 (G64 CHAMFER BETWEEN TWO STRAIGHT LINES)

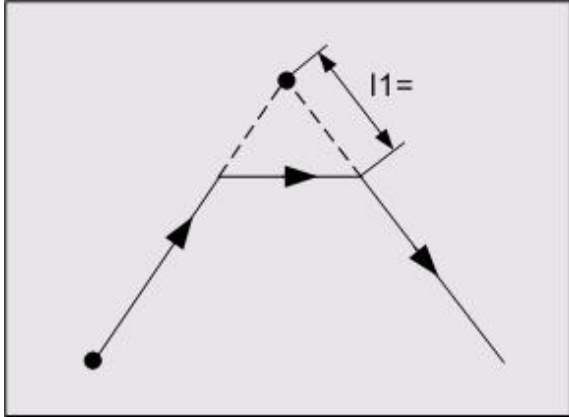
G1....

- G1

Straight line
- I

The chamfer (I20) is arranged symmetrically around the point of intersection.
The chamfer width is programmed with the I expression.
- G1

Straight line



Circles

Address description for circle

- ▶ X, Y, Z end point coordinates
- ▶ I center point in X/pitch in X
- ▶ J center point in Y/pitch in Y
- ▶ K center point in Z/pitch in Z
- ▶ R circle radius
- ▶ B1= angle
- ▶ B2= polar angle
- ▶ B3= polar angle for center
- ▶ B5= angle of arc
- ▶ J1= 1=intersection left, 2=right
- ▶ L2= polar length
- ▶ L3= polar length for center
- ▶ P1= point definition number
- ▶ R1= R1=0 tangent to line

Possible parameters for circles between G64 and G63 blocks.



See coordinate systems in the Programming chapter for an explanation of the possible coordinate systems (Cartesian, polar, absolute, and incremental) and definitions.

Circle with center point and radius (see figure)

G2/G3 I... J... R... (G64 CIRCLE WITH CENTER POINT AND RADIUS)

G2/G3 Circle with center point (M) and radius (R) (Figure 1).

Circle with center point and starting or end point

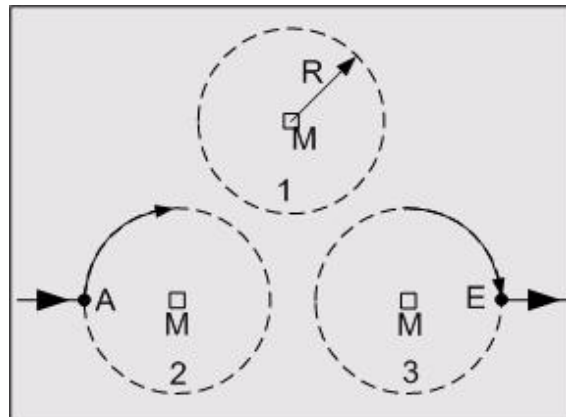
G1 X... Y...

G2/G3 I... J... R... (G64 CIRCLE WITH CENTER POINT AND STARTING POINT)

OR

G2/G3 I... J... X... Y... (G64 CIRCLE WITH CENTER POINT AND END POINT)

- G1 Movement to the starting point on the circle.
- G2/G3 Circle with center point (M) and radius (R) (Figure 2).
- G2/G3 Circle with center point (M) and end point (E) (Figure 3).



Circle with radius and starting or end point (see figure)

G1 X... Y...

G2/G3 R... R1=0 (G64 CIRCLE WITH RADIUS AND STARTING POINT)

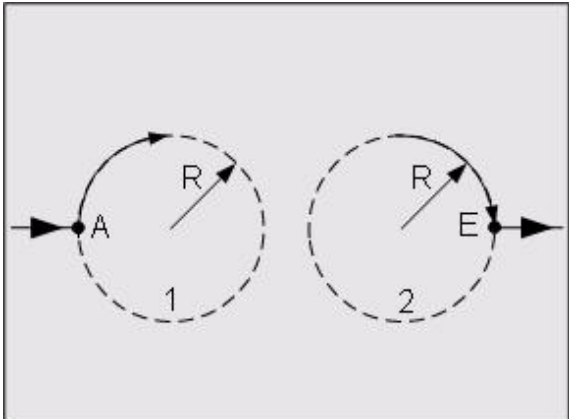
G1 B1=0 X... Y...

- G1

Movement to the starting point on the circle.
- G2/G3

Circle with only a radius (R).
R1=0 indicates that the circle is tangential to the next element.
- G1

Horizontal straight line with angle (B1=) and end point (E).



Circle with only a center point (see figure)

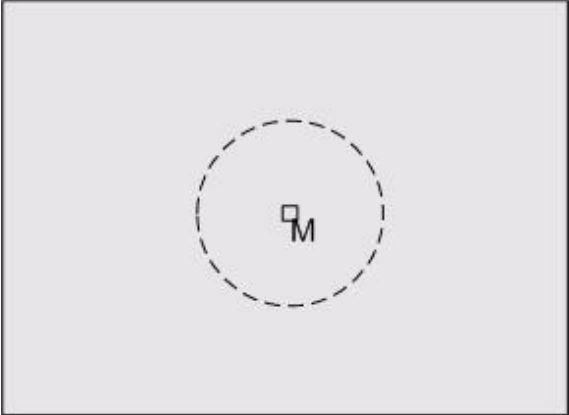
G1 X... Y... R1=0

G2/G2 L3=... B1=45 (G64 CIRCLE ONLY WITH CENTER POINT)

- G1

Straight line, tangential to the next circle.
- G2/G3

Circle with only a center point tangential to the straight line.



Circle with only a radius (see figure)

This circle can intersect a straight line or a circle.

G1 X0 Y0

G1 R1=0 (G64 STRAIGHT LINE TANGENTIAL TO THE CIRCLE)

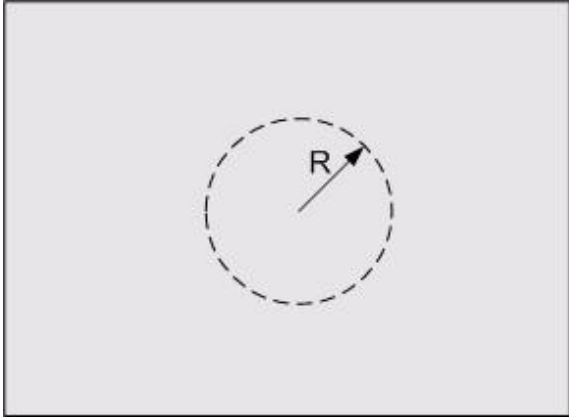
G2 I50 J50 X60 Y50

- G1

Movement to the starting point.
- G1

Straight line is tangential to the circle (indicated by R1=0).
The following element must be a circle.
- G2

Circle defined with a center point (A) and an end point. The side where the straight line is tangential to the circle is determined by the direction of rotation of the circle (G2 or G3).
The default setting is that a flowing movement takes place.



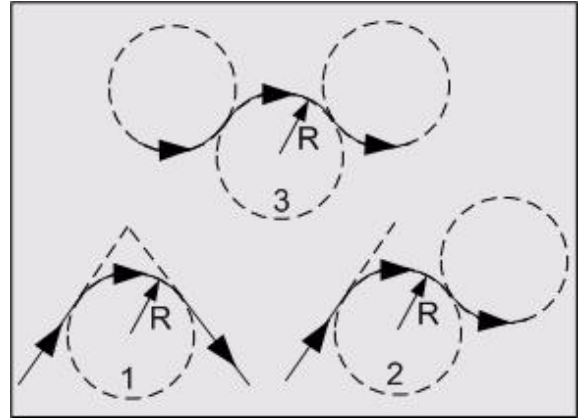
Rounding arcs

The rounding arc is always tangential to the straight lines or circles in the previous and subsequent block and is programmed with G2 or G3 to specify the direction of movement.

The type of connecting circle is determined by the direction of rotation on the three circles.



The connecting circle between two concentric circles is a special case of ONE CIRCLE INSIDE THE OTHER. In this case, the center points of both circles are at the same position. The expression B1=.. specifies the angle formed by the straight line through the circle center point of the concentric circles and of the connecting circle with the main axis. This additional information is incorporated in the block with the connecting circle.



Rounding arc between two elements (see figure)

G1.... OR G2/G3

G3/G2 R20 (G64 ROUNDING ARC BETWEEN TWO ELEMENTS)

G1.... OR G2/G3

G1 Straight line or circle
R Radius of rounding arc
G1 Straight line or circle

Points of intersection

There are two possible points of intersection when a straight line and a circle or two circles intersect. A special address (I2=1 or 2) is used to indicate the intersection coordinates to be calculated. There are two methods for determining which point of intersection belongs to I2=1 and which to I2=2.



The intersection indicator I2= only functions correctly if two elements are involved. I2= must not be used if a rounding arc is also inserted between the intersecting elements.
I2= must be programmed in the block where the point of intersection is selected.

When calculating the point of intersection for a **straight line and a circle or two circles**, the expression I2= indicates which of the two possible points of intersection is meant:

I2=1: the left-hand point of intersection (P1)

I2=2: the right-hand point of intersection (P2), viewed from the circle center point.

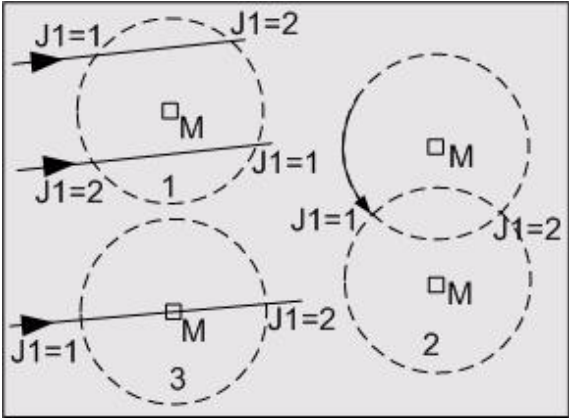
For a **line through the center point**:
I2=1: the short distance to the starting or end point.
I2=2: the greater distance to the starting or end point.

For a **line through the center point, where the starting or end point is at the same position as the center point**. The direction of movement for the programmed straight line, programmed with B1=, determines:
I2=1: the first point of intersection
I2=2: the second point of intersection

Points of intersection between two elements (see figure)

G1 X... Y.... I2=1 (G64 POINTS OF INTERSECTION BETWEEN TWO ELEMENTS)

G1 Straight line with an auxiliary point that intersects a circle.



Non-flowing transitions

These are not allowed. Only continuous movements can be programmed. This means the tool is always moving forwards.

Straight line tangential to circle (R1=0) (see figure)

The expression $R1=0$ is used to specify that a specific geometry element is tangential with the next element (connecting circles are not taken into consideration), thus:

- Straight line is tangential with circle
- Circle is tangential with straight line
- Circle is tangential with circle

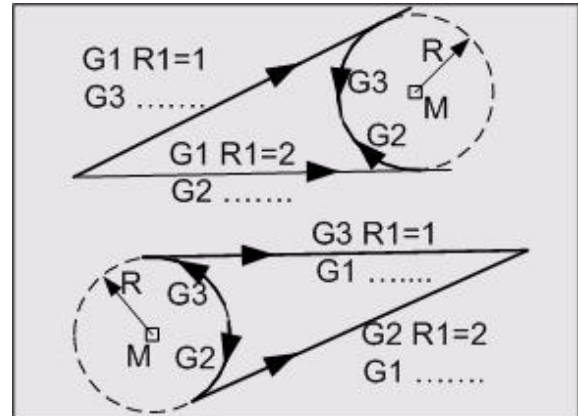
The expression $R1=0$ is written in the block with the first element.

The tangential point is selected so that the tool path is continuous, i.e. the tool is always moving forwards.

Note: With $R1=0$, the CNC automatically determines which tangential straight line maintains the continuous movement so that the tool does not travel backwards.



The tangential indicator $R1=$ only functions correctly if two elements are involved. $R1=$ must not be used if a connecting circle is also inserted between the intersecting elements.

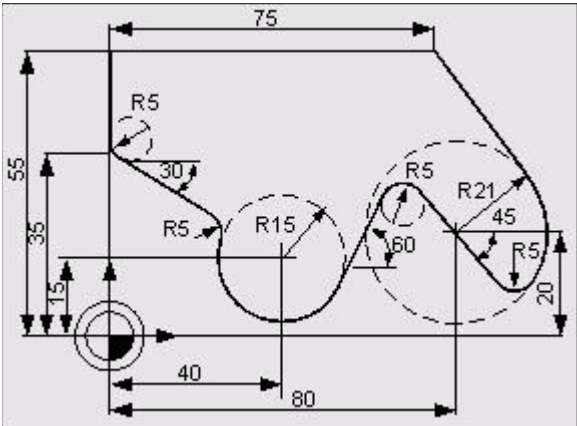


Rounding arc or connecting circle between two straight lines, between a straight line and a circle, or between 2 circles

The control automatically determines which rounding arc or connecting circle is used, depending on the direction of movement on the second element.

Example: G64 geometric calculation

Contour consisting of straight lines and circles.



G98 X-20 Y-10 Z-20 I140 J100	Graphic window definition
G54	Zero point shift
T1 M6	Insert tool
S1000 M3	Spindle speed
G0 X-5 Y60 Z10	Advance to starting point
G1 Z-10 F500	Move tool to depth
G43 X0	Radius compensation to programmed position
G42	Radius compensation, right
G64	Activate geometry
G1 B1=-90	Move tool along the Y-axis
G3 R5	A rounding arc between the two straight lines of the previous and the next block
G1 X0 Y35 I1=0 B1=-30	Move the tool along the straight line. The starting point for the straight line is programmed as an auxiliary point. The angle indicates the direction of movement.
G2 R5	A rounding arc between the last straight line and the next circle
G3 I40 J15 R15 R1=0	Move down the circle to the tangential point of the circle and the next straight line
G1 B1=60 I2=2	A straight line. Point of intersection viewed from the circle center point.
G2 R5	A rounding arc between the last and the next straight line
G1 X80 Y20 I1=0 B1=-45 I1=0	A straight line through the center point of the next G3 circle. The center point is used as an auxiliary point for the straight line.
G2 R5	A rounding arc between the last straight line and the next circle
G3 I80 J20 R21 R1=0	Move down the circle to the tangential point of the circle and the next straight line
G1 X75 Y55	A straight line to the programmed end point of the straight line

G1 X-20 Y55	Move the tool parallel to the X-axis until the tool is free of the workpiece. Note: both axes of the main plane must be programmed.
G40	Cancel tool radius compensation
G63	Cancel geometric calculations
G0 Z100 M30	End of program

5.41 G70 Inch Programming

Loading and calling of part programs that are written in a different unit of measure to that specified in the CNC. (Unit of measure defined in the configuration data)

Address description

No specific addresses.

Application

Modality

The functions G70 and G71 form a modal group.

Default setting

The configured measuring system automatically takes effect when the CNC is initialized.

Programming

G70 allows part programs to be executed in the inch unit of measure although the CNC is set metrically.

If G70 or G71 is not programmed at the start of a part program, then the CNC assumes that the programmed units of measure match the unit of measure set in the CNC.

Units of measure

With G70, the units of measure are as follows:

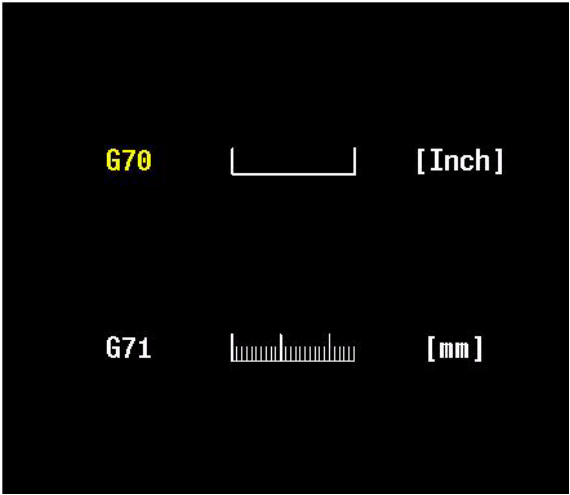
- Linear measurement [inch]
- Feed rate G94 [inch/min]
- Feed rate G95 [inch/rev]
- Cutting speed G96 [feet/min]

Example

N9001 G70

G1 X2 Y1.5 F8

N9001	Unit of measure: CNC: metric Program: inch
G1	Transferring has the result that X50.8 Y38.1 and F203.2 are saved.



5.42 G71 Metric Programming

Loading and calling of part programs that are written in a different unit of measure to that specified in the CNC. (Unit of measure defined in the machine configuration).

Address description

No specific addresses.

Application

Modality

The functions G71 and G70 form a modal group.

Default setting

The configured measuring system automatically takes effect when the CNC is initialized.

Programming

G71 allows part programs to be executed in the metric unit of measure although the CNC is set to inches.

If G70 or G71 is not programmed at the start of a part program, then the CNC assumes that the programmed units of measure match the unit of measure set in the CNC.

Units of measure

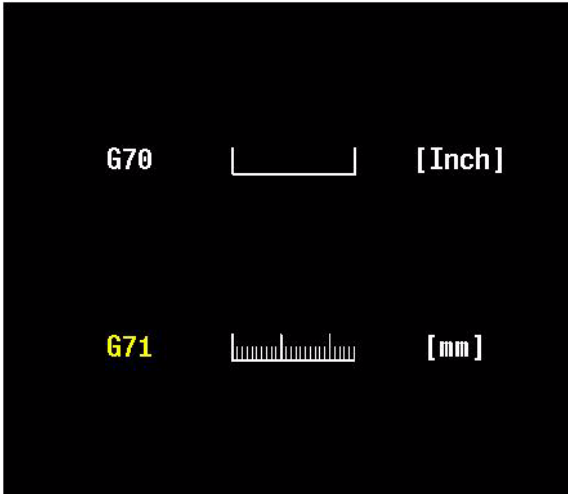
With G71, the units of measure are as follows:

- Linear measurement [mm]
- Feed rate G94 [mm/min]
- Feed rate G95 [mm/rev]
- Cutting speed G96 [m/min]

Example

```
N9001 G71
G1 X50.8 Z38.1 F203.2
```

N9001	Unit of measure: CNC: inch Program: metric
G1	Transferring has the result that X2 Y1.5 and F8 are saved.



5.43 G72 Cancel Mirror Image and Scaling

Cancellation of increase, decrease, or mirroring around an axis.

Address description

No specific addresses.

Application

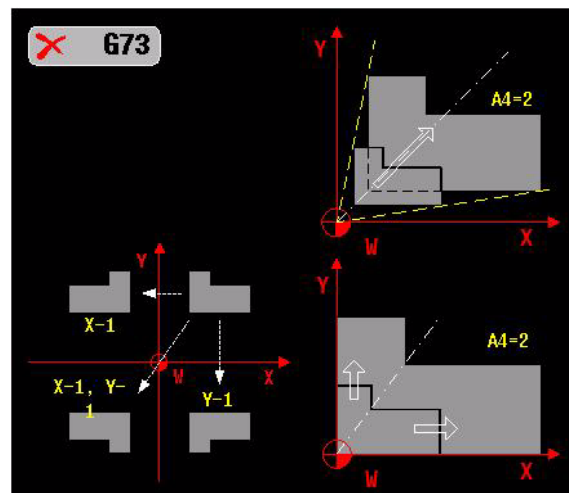
Modality

The functions G72 and G73 form a modal group.

Default setting

G72 automatically takes effect after:

- Control activation
- CNC reset
- Program cancelation
- M30.



5.44 G73 Mirror Image and Scaling

Increasing or decreasing an array of axis coordinates. Mirroring an array of linear main axis coordinates or reversing the algebraic sign for rotary axis coordinates.

Address description

- ▶ **X, Y, Z -1=**set mirror image, **1=**reset
- ▶ **B, C -1=**set mirror image, **1=**reset
- ▶ **A4=** scaling factor

Format

- Activating increase/decrease:
G73 A4= ... (factor or percentage, setting in machine parameter dimension)
- Deleting increase/decrease:
G73 A4=1 (factor)
G73 A4=100 (percentage)
- Mirroring around an axis or changing the algebraic sign per axis:
G73 {X-1} {Y-1} {Z-1} {A-1} {B-1} {C-1}
- Deleting mirroring/changing the algebraic sign per axis:
G73 {X1} {Y1} {Z1} {A1} {B1} {C1}

Application

Modality

The functions G72 and G73 form a modal group.

Increasing and decreasing by the zero point shift W

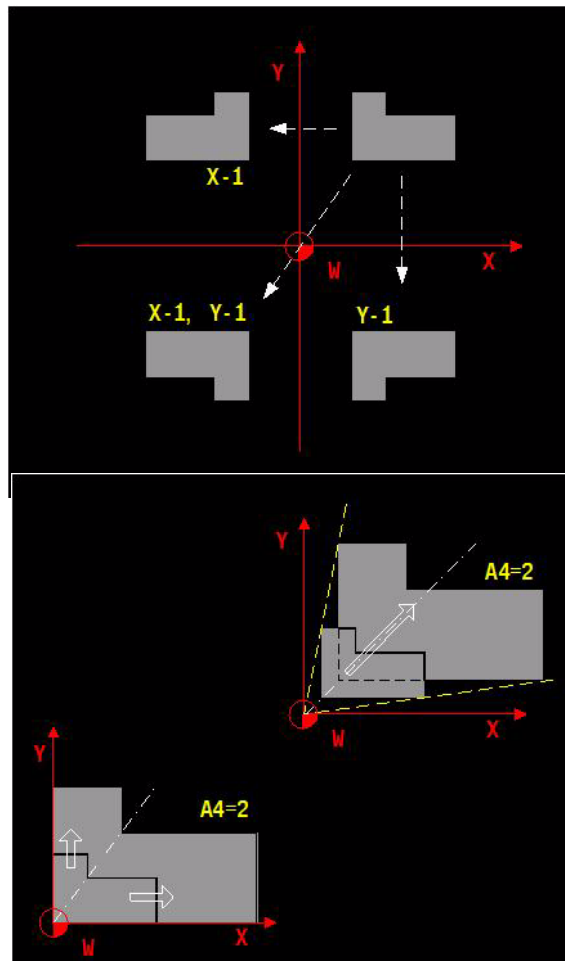
The function G73 uses the current zero point W as the starting point. If necessary, this point should be shifted to the geometric center of the array of coordinates using a G92 or G93 zero point shift, before the increase or decrease. As a result, the coordinates are symmetrical around a fixed point with an unchanging position.

Programmed zero point shifts G92/G93

The programmed zero point shifts G92/G93 are also increased or decreased when increasing or decreasing is active.

Saved zero point shifts G51 to G59

The saved zero point shifts G51 to G59 do not change during increasing or decreasing.



Tool axis

The configuration data specifies whether increasing or decreasing applies only to the axis coordinates in the main plane or to the tool axis as well.

Changes to V5xx

■ See "G73_G92" on page 508.

Example

```
G1 X45 Y45
G73 X-1 Y-1
G1 X45 Y45
G72
```

G1
G73 Mirror coordinates around the X and Y axis
G1
G72 Delete mirroring

5.45 G74 Absolute Position Approach

Rapid traverse movement to a position whose coordinates refer to the machine-based reference point R or to machine positions.

Address description

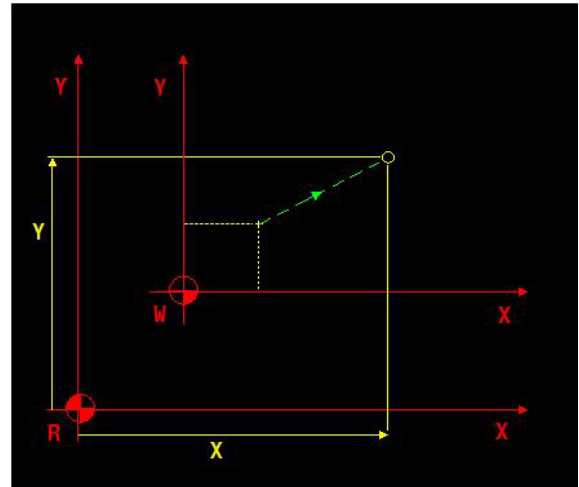
- ▶ **X, Y, Z end point coordinates**
- ▶ **A, B, C end angles**
- ▶ **K block transition: 0=exact, 1=no stop** 0: a precision stop is taken into consideration between the movement of block G74 and the movement in the next block, as is standard for rapid traverse movements (on-position). 1: no stop is taken into consideration between the movement of block G74 and the movement in the next block (smoothing). The next movement begins once the nominal position is almost reached in all axes.
- ▶ **L 0=with tool length, 1=without** 0: tool length compensation is applied (on-position). L1: no tool length compensation.
- ▶ **X1=, Y1=, Z1= absolute position number (1-18)**
- ▶ **A1=, B1=, C1= absolute position number (1-18)**

Format

G74 X... Y... Z... {A...} {B...} {C...} {X1=...} {Y1=...} {Z1=...} {K...} {L...}

Default setting

K0: precision stop between the block transitions, L0: tool length compensation active.



Application

The function G74 is primarily used in cycles for tool changers, pallet stations and similar, specifically when the programmed coordinates are to be independent from the coordinates used to define the tool handling.

The end point coordinates can be defined using two methods:

- X100: position in relation to the reference point.
- X1=2: position in relation to the reference point, defined by the second machine parameter within CfgPlcPositions.

For each axis, 18 machine positions can be specified in the machine parameters (CfgPlcPositions). No traverse movement is performed if the machine parameter used is zero.

With G74, a simultaneous traverse movement is carried out in all programmed axes. The next traverse movement does not begin until the nominal position is reached in all axes.

If an incremental movement is programmed after a G74-movement, then the coordinates refer to the position specified in the G74-block.

The traverse movement immediately before G74 must be programmed with G0 or G1. The traverse movement immediately after G74 is automatically performed with the same G-function.

The G41...G44 radius compensation, the G141 tool compensation 3D, the G64 geometric function, and the G196 graphic contour description must be switched off before the G74-function is activated.

The programmed G74 position is independent of the effective zero point shift, B4= axis rotation, or G72/G73 increasing/decreasing.

Changes to V5xx

- See "G74" on page 509.

Example

Program 1: (see figure)

G0 X95 Y20

G74 X-35 Y-50

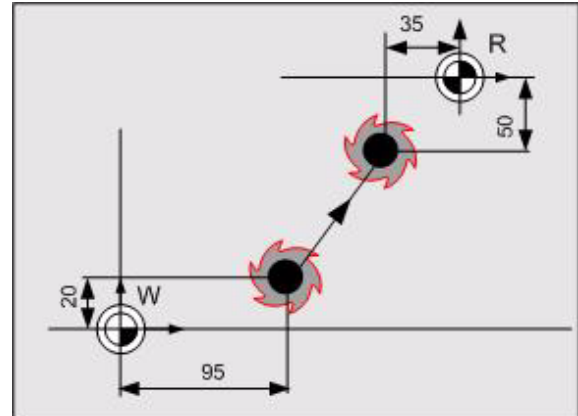
G0 Approach coordinates X95 Y20.
 G74 Movement from X95 Y20 to the absolute position with coordinates X35 Y50 relative to the reference point.

Program 2: (see figure)

G0 X95 Y20

G74 X1=1 Y1=7

G0 Approach coordinates X95 Y20.
 G74 Movement from X95 Y20 to the absolute position that is stored in the machine parameter CfgPlcPositions for the X axis and the Y axis .f



5.46 G77 Bolt Hole Circle

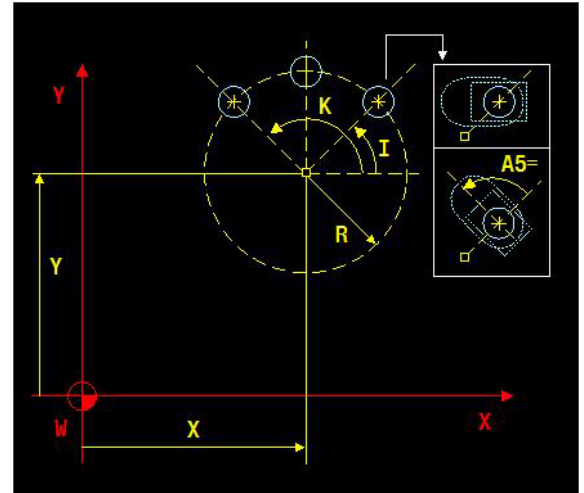
Execution of previously programmed drilling or milling cycles at points located at equal distances on an arc or full circle.

Address description

- ▶ X, Y, Z center point coordinate
- ▶ B, C end point angles
- ▶ I angle to first point
- ▶ J number of points
- ▶ K angle to last point
- ▶ R circular pattern radius
- ▶ B1= angle
- ▶ L1= path length
- ▶ B2= polar angle
- ▶ L2= polar length
- ▶ A5= angle of rotation Pocket angle
- ▶ ?90= center point abs. (X,Y,Z..)
- ▶ ?91= center point incr. (X,Y,Z..)
- ▶ P1= point definition no. for center

Format

- Points on an arc:
G77 [center point] R... J... I... K... {A5=...}
- Points on a full circle:
G77 [center point] R... J... I... {A5=...}
- Points on multiple arcs:
G77 P1=... P2=... P3=... P4=... R... J... I... K... {A5=...}



Application

Associated functions

G79, G81, G83 - G89, G771 - G773, G777, G778.

Switch off radius compensation

Radius compensation must be switched off with G40 before calling a G77 block.

Turned pocket or slot

A predefined pocket or slot can be turned by an angle. The center of rotation is the point that is used in the G77 block to program the position of the pocket or slot.

The angle is programmed with the A5= address and lies between -360 and +360 degrees.

There are three options:

- A5= address is not programmed. In this case, the pocket or slot sides run parallel to the X-axis (G17 and G18) or the -Z-axis (G19).
- A5=0. In this case, the axis for each pocket or slot is radial, i.e. it is positioned in the direction of the radius from the circle center point to the point on the circle.
- A5=<>0. In this case, B1= indicates the angle formed by the pocket or slot with the radius to the pocket center.

Incremental programming following G77

There are two options:

- The incremental movement following G77 is another sample or cycle design. In this case, the next cycle starting point is calculated starting from the circle hole center point.
- The incremental movement following G77 is a separate movement. Here, the end point is calculated starting from the current position.

Changes to V5xx

- See "G77_G91" on page 509.

Example

Example: (see figure)

G78 P2 X... Y... Z...

G81 Y1 Z-10 F100 S1000 M3

G77 P2 R25 I30 K150 J4

G78 P1 X... Y... Z...

G81 Y1 Z-10 F100 S1000 M3

G77 P1 R25 I0 J6

- G78 Second defined point
- G81 Define the cycle
- G77 Repeat the cycle four times on the arc
- G78 First defined point
- G81 Define the cycle
- G77 Repeat the cycle six times on the full circle

Example: turned slots (see figure)

T1 M6

G88 X20 Y10 Z-10 B1 F100 S1000 M3

G77 X78 Y56 Z0 R24 I0 J6 A5=30

- T1 Insert tool 1 (milling tool with a radius of 4.8 mm)
- G88 Define the slot as if the sides ran parallel to the X and Y axes
- G77 The turned slots are milled

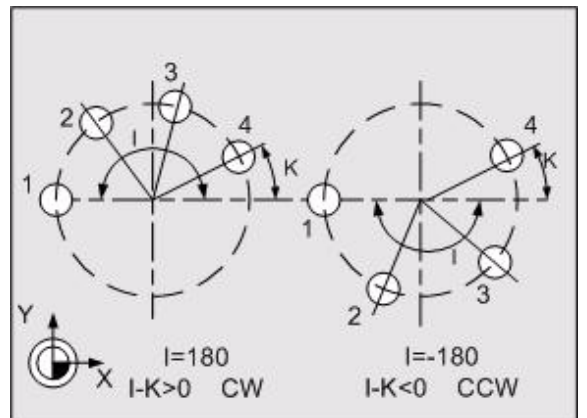
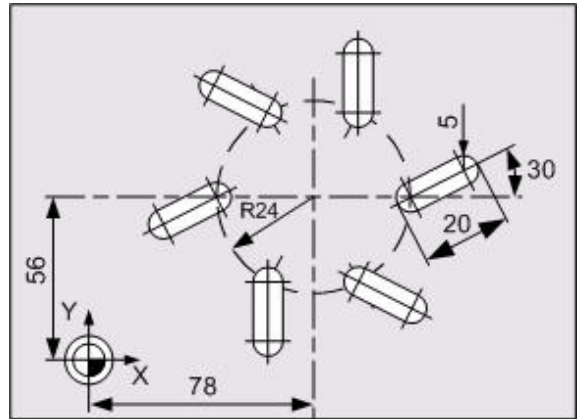
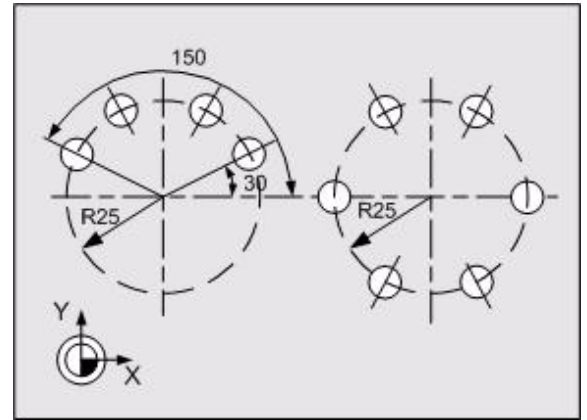
Example: direction of the holes on an arc (see figure)

G81 Y1 Z-10 F100 S1000 M3

G77 X0 Y0 Z0 R25 I180 K30 J4

G77 X0 Y0 Z0 R25 I-180 K30 J4

- G81 Define the cycle
- G77 Repeat the cycle four times on the arc; move from 180 degrees to 30 degrees in a clockwise (CW) direction.
- G77 Repeat the cycle four times on the arc; move from -180 degrees to 30 degrees in a counter-clockwise (CCW) direction.



5.47 G78 Point Definition

Unique definition of the coordinates for a point in a program. You only have to program the point number for a traverse movement to this point.

Address description

- ▶ X, Y, Z point coordinates
- ▶ B, C point angles
- ▶ B2= polar angle
- ▶ L2= polar length
- ▶ P1 point definition number

Format

G78 P1=... [point coordinates]

Application

Point definitions

Only one point can be defined in any G78 block. All point coordinates refer to the active workpiece zero point W.

Only Cartesian coordinates relative to the active zero point W or polar coordinates (B2=, L2=) can be used in the main plane.

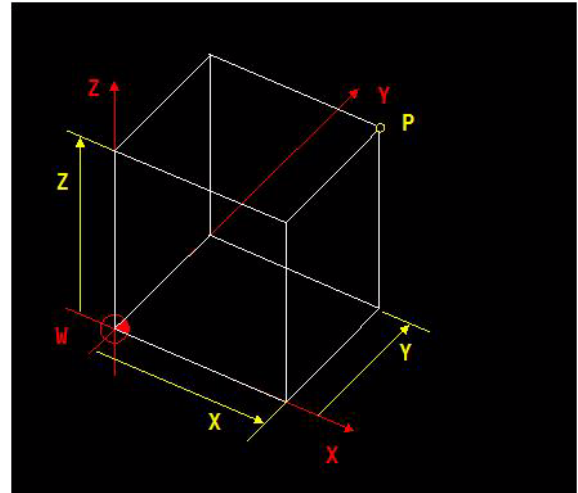
Program blocks with G1 or G79 can contain up to 4 points. Otherwise, a program block can only contain one point.

Example: G1 P1=9 P2=1 P3=3 P4=8

P address with index: The index value (1-4) gives the priority for the execution sequence (1= highest priority, 4=lowest priority). The entry after the equals sign indicates the number of the point in the point table. Another option is to enter the point definition as a parameter, where the index defines the priority.

Number of point definitions

The number of point definitions in the point table for the CNC can be specified with the "POINTS.PTT or SIMPOINTS.PTT" configuration file.



Effectiveness

The coordinates for a defined point remain effective until:

- The point is redefined by another G78 block.
- The point memory is changed or is deleted by the operator.
- New defined points are uploaded.

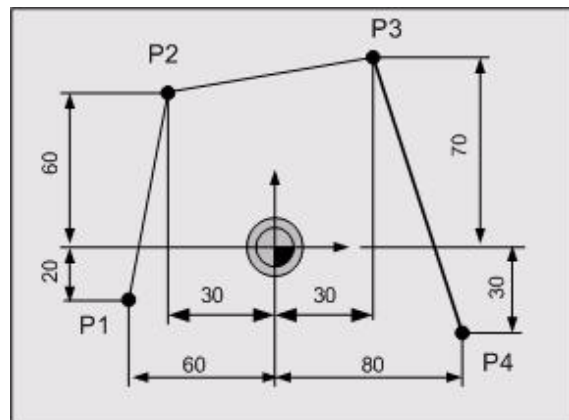
The point memory is not affected by a CNC reset.

Example

Example program (see figure)

G78 X-60 Y-20 P1
G78 X-30 Y60 P2
G78 X30 Y70 P3
G78 X80 Y-30 P4
G0 P1=1
G1 P1=2 P2=3 P3=4

G78	Define point 1
G78	Define point 2
G78	Define point 3
G78	Define point 4
G0	Move the tool in rapid traverse to the position defined by P1.
G1	Move the tool to P2, P3, and then P4 at the programmed feed rate



5.48 G79 Cycle Call

Execution of previously programmed drilling cycles (G81, G83-G86) or milling cycles (G87-G89) at specific positions.

Address description

- ▶ X, Y, Z point coordinates
- ▶ B, C point angles
- ▶ B1= angle
- ▶ L1= path length
- ▶ B2= polar angle
- ▶ L2= polar length
- ▶ A5= angle of rotation Pocket angle
- ▶ ?90= point abs. (X,Y,Z..)
- ▶ ?91= point incr. (X,Y,Z..)
- ▶ P1= .. P4= point definition numbers

Application

Associated functions

G77, G81, G83 - G89, G771 - G773, G777, G778, G781, G783 - G789.

Positions in the main plane

The positions where a predefined cycle is to be executed are programmed in the G79 blocks that follow the cycle definition.

The positions in the main plane are programmed with point or polar coordinates or with a coordinate and an angle.

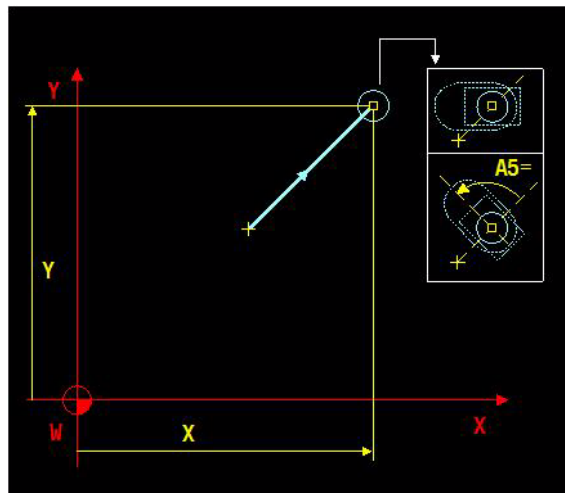
A G79 block can contain up to 4 predefined points. The predefined cycle is executed at each of these points.

Switch off radius compensation

Radius compensation must be switched off with G40 before calling a G79 block.

Turned pocket or slot

A5= represents the angle for turning a pocket or slot. See G77 example "Turned slots".



Incremental programming following G79

There are two options:

- The incremental movement following G79 is another sample or cycle design. In this case, the next cycle starting point is calculated starting from the last cycle starting point.
- The incremental movement following G79 is a separate movement. Here, the end point is calculated starting from the current position.

Changes to V5xx

- See "G79" on page 510.

Example

Three holes are to be drilled (see figure)

G78 P1 X50 Y20 Z0

G78 P2 X50 Y80 Z0

T1 M6

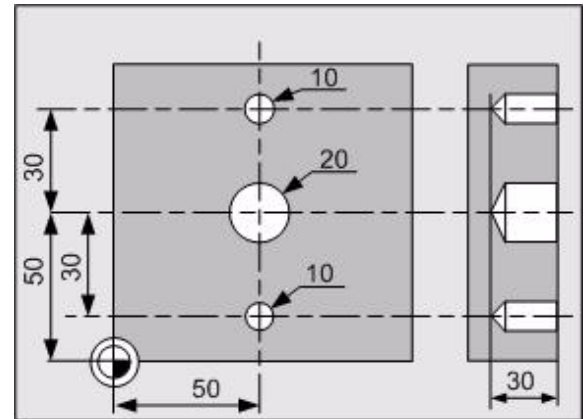
G81 Y1 Z-30 F100 S1000 M3

G79 P1 P2

T2 M6

G79 X50 Y50 Z0 M3

G78	Define point 1
G78	Define point 2
T1	Insert tool
G81	Define bore cycle
G79	Drill holes at point 1 first and then at point 2
T2	Insert a different tool
G79	Drill the hole at the programmed position



5.49 G81 Drilling/Centering

Definition of a drilling cycle in a single program block. See also cycle G781.

Address description

- ▶ Z drilling depth
- ▶ X dwell time [s]
- ▶ Y 1st setup clearance
- ▶ B 2nd setup clearance

Format

G81 Z... {X...} {Y...} {B...}

Application

A G81 drilling cycle is performed with G77 or G79.

Associated functions

G77, G79, G83 - G89, G781.

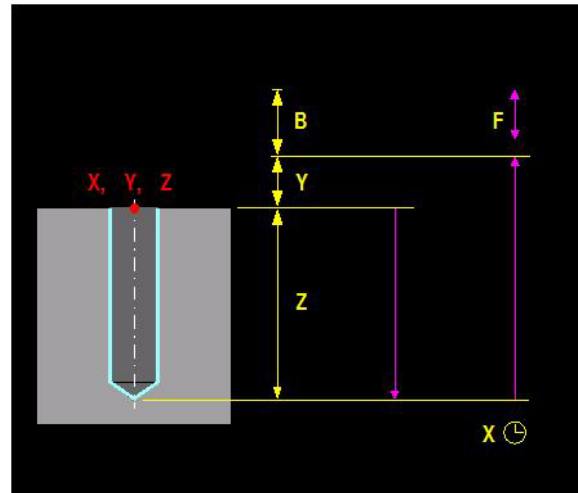
Dwell time

It is possible to implement a dwell time at the bottom of the hole. The unit is 0.1 s.

Deleting the cycle data

The cycle data remains active in the program until it is deleted by:

- Defining a new cycle
- Canceling the program
- M30
- CNC reset



Example

Program example: (see figure)

G78 P1 X50 Y20 Z0

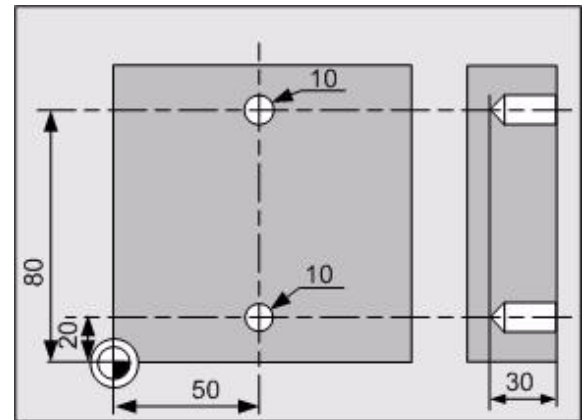
G78 P2 X50 Y80 Z0

G0 Z10 T1 M6

G81 X1.5 Y1 Z-30 F100 S500 M3

G79 P1 P2

G78	Define point 1
G78	Define point 2
G0	Insert tool and start from the tool change position
G81	Define bore cycle
G79	Execute the drilling cycle at point 1 and then at point 2



5.50 G83 Deep-Hole Drilling

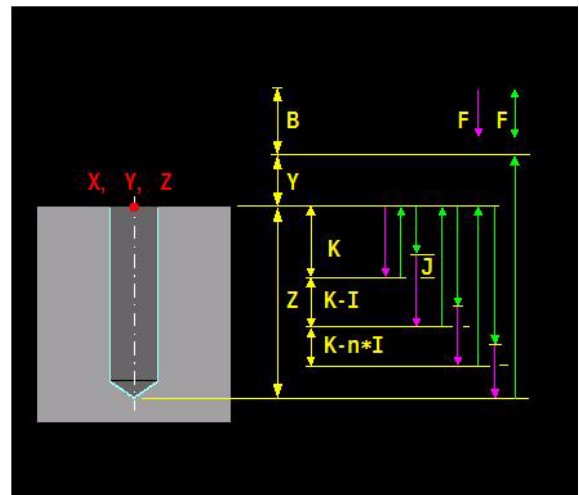
Programming of a deep-hole drilling cycle in a program block. See also cycle G782 and G783.

Address description

- ▶ Z drilling depth
- ▶ X dwell time [s]
- ▶ Y 1st setup clearance
- ▶ B 2nd setup clearance
- ▶ I cutting depth reduction
- ▶ J retract distance for chip break
- ▶ K cutting depth
- ▶ K1= number of steps before retract

Format

G83 Z... {X...} {Y...} {B...} {I...} {J...} {K...} {K1=...}



Application

A G83 deep-hole drilling cycle is performed with G77 or G79.

Associated functions

G77, G79, G81, G84 - G89, G782, G783.

Deep-hole drilling

The Z value is the total depth in relation to the workpiece surface. The algebraic sign determines the tool direction.

The K value is the plunging depth of the first drill step for deep-hole drilling in several steps. The K value has no algebraic sign.

The plunging depth K is reduced by the I value for every subsequent step. The I value is used if the calculated plunging depth becomes less than the I value. Only the last plunging depth can be less than the I value. The plunging depth remains the same until the last cut if I=0.

A retraction by the J value is performed after every infeed. The tool normally remains in the hole during this process, while the chips are broken. If the J value is equal to zero, the tool is retracted to the safety clearance in each case.

The K1= value specifies the number of infeeds before chipping. When the infeed number is reached, the tool is retracted to the safety clearance and not by the J value.

Dwell time

It is possible to implement a dwell time at the bottom of the hole. The unit is 0.1 s.

Deleting the cycle data

The cycle data remains active in the program until it is deleted by:

- Defining a new cycle
- Canceling the program
- M30
- CNC reset

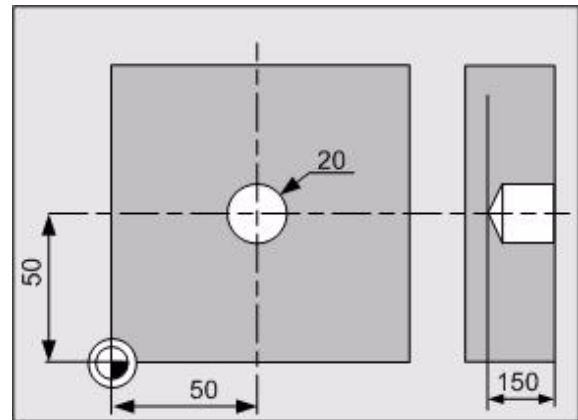
Example

Program example: (see figure)

G83 Y4 Z-150 I2 J6 K20 K1=3

G79 X50 Y50 Z0

G83 Define deep-hole drilling cycle
G79 Execute the deep-hole drilling cycle at the
 programmed position



5.51 G84 Tapping

Definition of a tapping cycle in a single program block. See also cycle G784.

Address description

- ▶ **Z** tapping depth
- ▶ **X** dwell time [s]
- ▶ **Y** 1st setup clearance
- ▶ **B** 2nd setup clearance
- ▶ **I** positioning ramp [revolutions]
- ▶ **J** pitch
- ▶ **I1=** interpolation 0=without, 1=with

Format

- G84 Z... {Y...} {B...} {J...} {X...} or
- G84 **I1=0** Z... {Y...} {B...} {J...} {X...}

The tapping can also be carried out as an interpolation in a closed control loop between the tool axis and the spindle. This interpolation also includes the acceleration power of the spindle. This guarantees that the spindle runs in the desired position and at the required speed. ("Synchronized tapping").

- G84 **I1=1** Z... {Y...} {B...} {J...} {X...}

Application

A G84 tapping cycle is performed with G77 or G79.

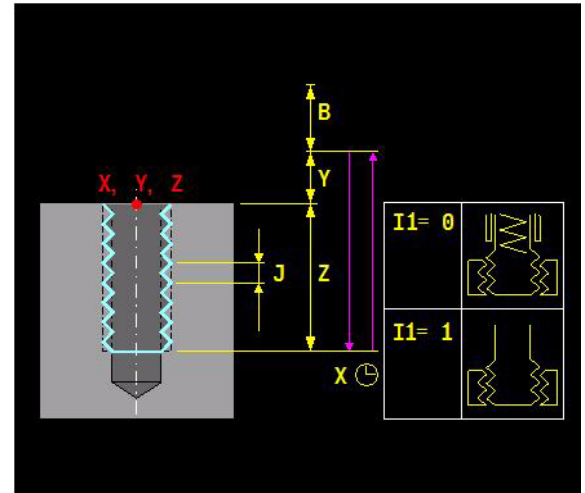
Associated functions

G77, G79, G81, G83, G85 - G89, G784.

Feed rate

$F(\text{feed rate}) = J(\text{pitch}) * S(\text{speed})$.

When a G84-cycle is called using G79, the CNC must be set to G94-operation (feed rate in mm/min) and not to G95-operation (feed rate in mm/rev). G94 must always be programmed before G84.



Interpolation

Tapping can be programmed with or without interpolation.

- I1=0 guided (default setting, open control loop)
- I1=1 interpolating (closed control loop)

Tilting an active G7 working plane can only be machined with interpolation (I1=1). Guided tapping is also possible (I=0) with an active G7, where the head is not tilted (the tool axis is directly on the Z axis).

Recutting the thread: For machines with interpolation (I1=1), you can recut the thread by programming of an oriented spindle stop (M19) with the D parameter "Spindle offset value".

Dwell time

It is possible to implement a dwell time at the bottom of the hole. The unit is 0.1 s.

Deleting the cycle data

The cycle data remains active in the program until it is deleted by:

- Defining a new cycle
- Canceling the program
- M30
- CNC reset

Changes to V5xx

- See "G84" on page 511.

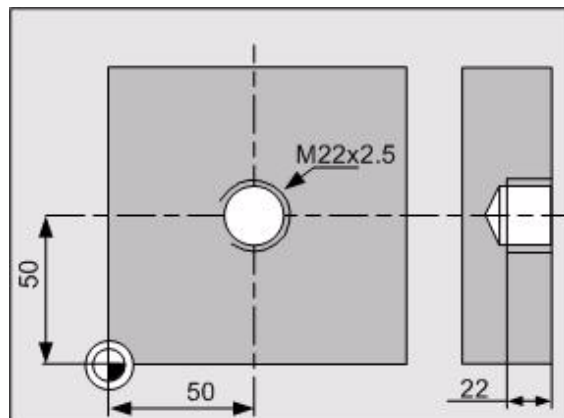
Example

Program example: (see figure)

G84 Y9 Z-22 J2.5 S56 M3 F140

G79 X50 Y50 Z0

G84	Define the tapping cycle
G79	Execute the tapping cycle at the programmed position



5.52 G85 Reaming

Definition of a reaming cycle in a program block. See also cycle G785.

Address description

- ▶ Z reaming depth
- ▶ X dwell time [s]
- ▶ Y 1st setup clearance
- ▶ B 2nd setup clearance
- ▶ F2= feed rate to start point

Format

G85 Z... {X...} {Y...} {B...} {F2=...}

Application

A G85 reaming cycle is performed with G77 or G79.

Associated functions

G77, G79, G81, G83, G84, G86 - G89, G785.

Optimizing the execution time for the reaming cycle

Tool retraction can be accelerated with a programmed retraction feed rate F2=. This shortens the execution time for the reaming cycle. If F2= is not programmed, the retraction is carried out with the programmed feed rate F.

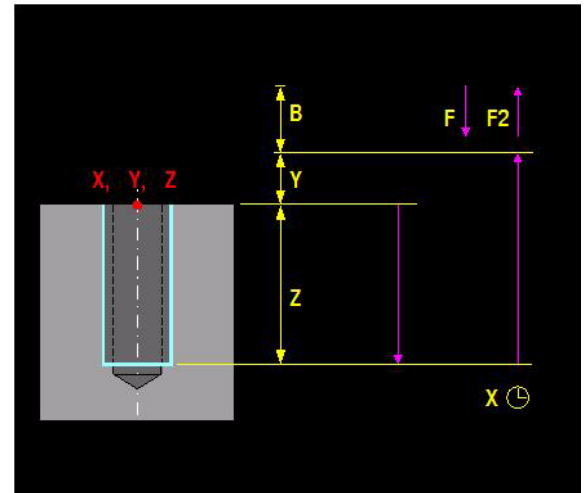
Dwell time

It is possible to implement a dwell time at the bottom of the hole. The unit is 0.1 s.

Deleting the cycle data

The cycle data remains active in the program until it is deleted by:

- Defining a new cycle
- Canceling the program
- M30
- CNC reset



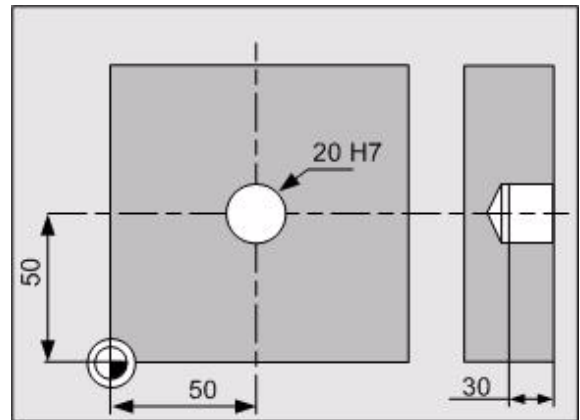
Example

Program example: (see figure)

G85 X2 Y3 Z-30 F50 S100 F2=200 M3

G79 X50 Y50 Z0

G85	Define reaming cycle
G79	Execute the reaming cycle at the programmed position



5.53 G86 Boring

Definition of a reverse boring cycle in a single program block. See also cycle G786.

Address description

- ▶ Z boring depth
- ▶ X dwell time [s]
- ▶ Y 1st setup clearance
- ▶ B 2nd setup clearance

Format

G86 Z... {X...} {Y...} {B...}

Application

A G86 reverse boring cycle is performed with G77 or G79.

Associated functions

G77, G79, G81, G83 - G85, G87 - G89, G786.

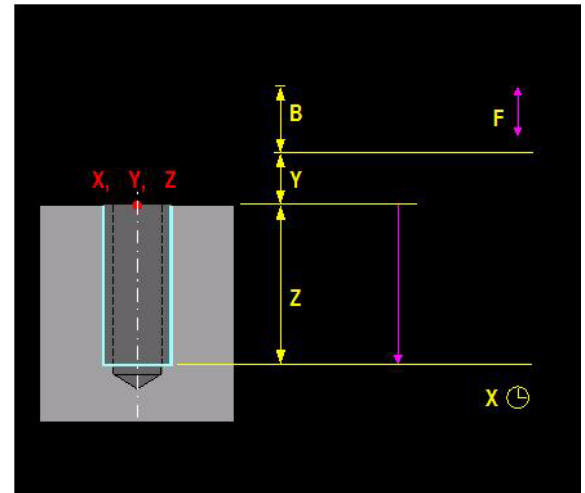
Dwell time

It is possible to implement a dwell time at the bottom of the hole. The unit is 0.1 s.

Deleting the cycle data

The cycle data remains active in the program until it is deleted by:

- Defining a new cycle
- Canceling the program
- M30
- CNC reset



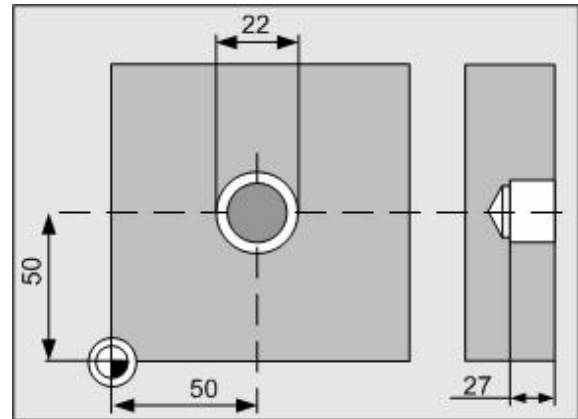
Example

Program example: (see figure)

```
G86 X1 Y9 Z-27 B10 F20 S500 M3
```

```
G79 X50 Y50 Z0
```

G86	Define reverse boring cycle
G79	Execute the reverse boring cycle at the programmed position



5.54 G87 Pocket Milling

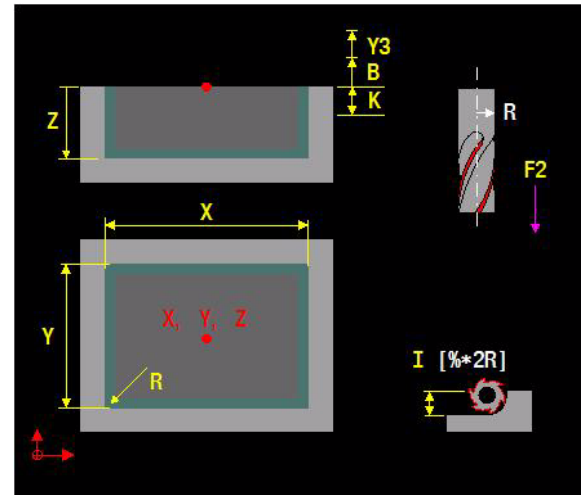
Programming of a rectangular pocket milling cycle in a program block.
See also cycle G787 and G797.

Address description

- ▶ X 1st side length
- ▶ Y 2nd side length
- ▶ Z pocket depth
- ▶ B 1st setup clearance
- ▶ K plunging depth
- ▶ I proportional cutting width
- ▶ R rounding radius
- ▶ J milling 1=climb -1=conventional
- ▶ Y3= 2nd setup clearance
- ▶ F2= feed for plunging

Format

G87 X... Y... Z... {R...} {B...} {I...} {J...} {K...} {Y3=...} {F2=...}



Application

A G87 rectangular pocket milling cycle is performed with G77 or G79.

Associated functions

G77, G79, G81, G83 - G87, G88, G89, G787.

Pocket geometry

The X, Y, Z, and R expressions determine the pocket geometry. The X and Y expressions do not have an algebraic sign.

The other expressions are the machining parameters.

Deleting the cycle data

The cycle data remains active in the program until it is deleted by:

- Defining a new cycle
- Canceling the program
- M30
- CNC reset

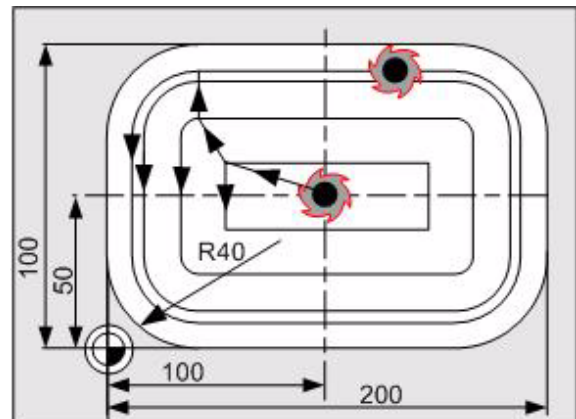
Example

Program example: (see figure)

```
G87 X200 Y100 Z-6 J+1 B1 R40 I75 K1.5 F200 S500 M3
```

```
G79 X120 Y70 Z0
```

G87	Define pocket milling cycle
G79	Execute the pocket milling cycle at the programmed position



5.55 G88 Key-Way Milling

Definition of the geometry for a slot and specific parameters for milling the slot in a program block. See also cycle G788 and G798.

Address description

- ▶ X 1st side length
- ▶ Y 2nd side length
- ▶ Z key-way depth
- ▶ B 1st setup clearance
- ▶ Y3= 2nd setup clearance
- ▶ K plunging depth
- ▶ J milling 1=climb -1=conventional
- ▶ F2= feed for plunging

Format

G88 X... Y... Z... {B...} {J...} {K...} {Y3=...} {F2=...}

Application

A G88 key-way milling cycle is performed with G77 or G79.

The algebraic signs of X and Y determine the direction of the slot from the starting point S.

Associated functions

G77, G79, G81, G83 - G87, G89, G788, G798.

Slot geometry

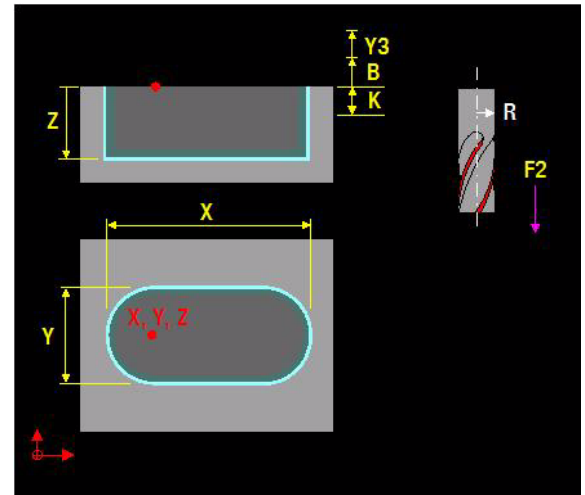
The X, Y, and Z expressions determine the slot geometry.

The other expressions are the machining parameters.

Slot parallel to the X or Y axis

If the slot is parallel to the X axis, then the algebraic sign of the X value determines the direction of the slot from the starting point. The Y value is then programmed without an algebraic sign.

If the slot is parallel to the Y axis, then the algebraic sign of the Y value determines the direction of the slot from the starting point. The X value is then programmed without an algebraic sign.



Deleting the cycle data

The cycle data remains active in the program until it is deleted by:

- Defining a new cycle
- Canceling the program
- M30
- CNC reset

Example

Program example: (see figure)

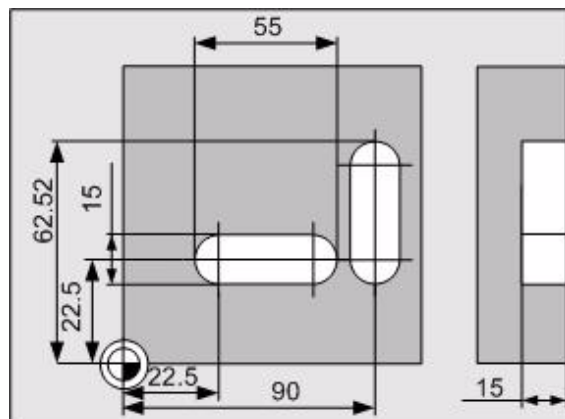
G88 X55 Y15 Z-5 B1 K1 F350 Y3=10 F2=200 M3

G79 X22.5 Y22.5 Z0

G88 X15 Y-55 Z-5 B1 K1 Y3=10 F2=200

G79 X90 Y62.528 Z0

- | | |
|-----|--|
| G88 | Define the cycle for milling a slot parallel to the X axis |
| G79 | Execute the key-way milling cycle at the programmed position |
| G88 | Define the cycle for milling a slot parallel to the Y axis |
| G79 | Execute the key-way milling cycle at the programmed position |



5.56 G89 Circular Pocket Milling

Programming of a circular pocket milling cycle in a program block. See also cycle G789 and G799.

Address description

- ▶ R radius of circular pocket
- ▶ Z pocket depth
- ▶ B 1st setup clearance
- ▶ Y3= 2nd setup clearance
- ▶ K plunging depth
- ▶ I proportional cutting width
- ▶ J milling 1=climb -1=conventional
- ▶ F2= feed for plunging

Format

G89 Z... R... {B...} {I...} {J...} {K...} {Y3=...} {F2=...}

Application

A G89 circular pocket milling cycle is performed with G77 or G79.

Associated functions

G77, G79, G81, G83 - G88, G789, G799.

Circular pocket geometry

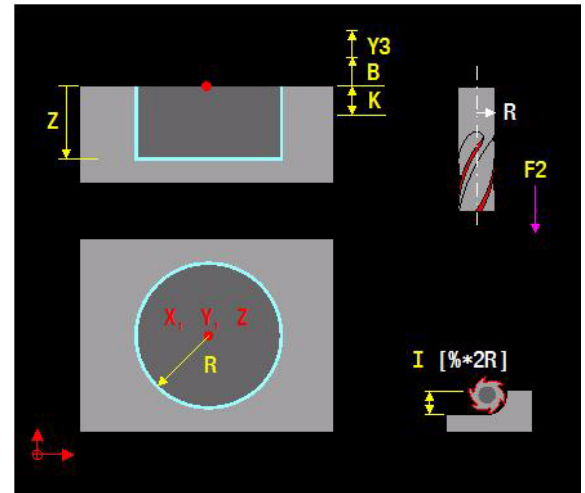
The Z and R expressions determine the circular pocket geometry.

The other expressions are the machining parameters.

Deleting the cycle data

The cycle data remains active in the program until it is deleted by:

- Defining a new cycle
- Canceling the program
- M30
- CNC reset



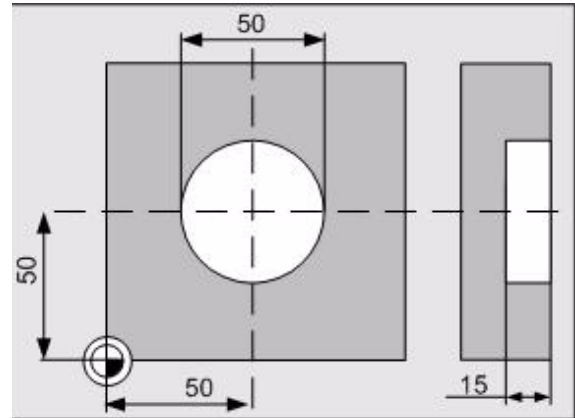
Example

Program example: (see figure)

```
G89 Z-15 B1 R25 I75 K6 F200 S500 M3
```

```
G79 X50 Y50 Z0
```

G89	Define circular pocket cycle
G79	Execute the circular pocket cycle at the programmed position



5.57 G90 Absolute Programming

Absolute coordinates, measured from the program zero point W.

Application

Modality

G90 and G91 are modal together.

Default setting

G90 automatically takes effect after control activation, CNC reset, Cancel program, or M30.

G90 is only canceled by programming G91.

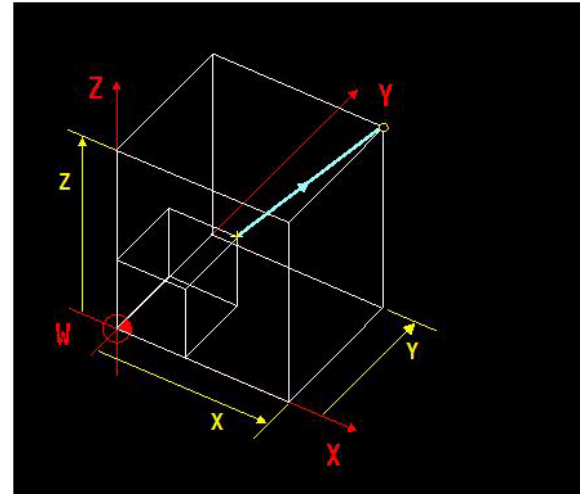
Polar coordinates

The polar coordinates (B1=, L1=), (B2=, L2=), (B3=, L3=) are not affected by G90.

Word-oriented absolute programming

This type of absolute programming is independent of G90 and G91 and is only allowed for the G codes: G0, G1, G2, G3, G9, G45, G46, G61, G62, G77, G79, G145, and G182. The following addresses are used for this:

- X90=, Y90=, Z90= end point, absolute
- U90=, V90=, W90= end point, absolute
- I90=, J90=, K90= circle center point, absolute
- A90=, B90=, C90= end angle, absolute



5.57 G90 Absolute Programming

G90 is the default setting and does not need to be programmed.

G79 X50

Technical drawing of a mechanical part showing front and side views with dimensions.

Front View:

- Overall width: 70 (50 + 20)
- Overall height: 70 (50 + 20)
- Four holes arranged in a 2x2 grid.
- Distance from left edge to first column of holes: 50
- Distance between columns of holes: 20
- Distance from bottom edge to first row of holes: 50
- Distance between rows of holes: 20
- Radius of each hole: 10
- A circular feature (fillet or chamfer) is located at the bottom-left corner.

Side View:

- Width: 30
- Shows the profile of the part, including the thickness of the material.

5.58 G91 Incremental Programming

Incremental coordinates, relative to the last position.

Application

- An absolute position must be programmed before the incremental dimensions of G91.

Modality

G90 and G91 are modal together.

Default setting

G90 automatically takes effect after control activation, CNC reset, Cancel program, or M30.

G91 is also canceled by programming G90.

Polar coordinates

The polar coordinates (B1=, L1=), (B2=, L2=), (B3=, L3=) are not affected by G91.

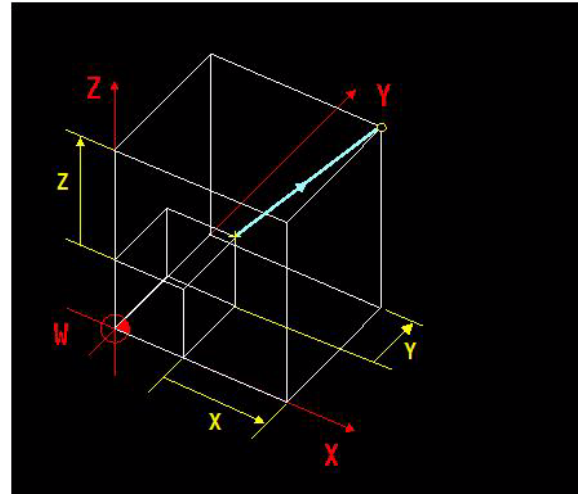
Word-oriented incremental programming

This type of absolute programming is independent of G90 and G91 and is only allowed for the G codes: G0, G1, G2, G3, G9, G45, G46, G61, G62, G77, G79, G145, and G182. The following addresses are used for this:

- X91=, Y91=, Z91= end point, incremental
- U91=, V91=, W91= end point, incremental
- I91=, J91=, K91= circle center point, incremental
- A91=, B91=, C91= end angle, incremental

Changes to V5xx

- See "G0..G3_G91" on page 492.
- See "G40_G91" on page 502.
- See "G73_G92" on page 508.



Example

Program example with G91: (see figure)

G81 Y2 Z-10 F200 M3

G79 X50 Y50 Z0

G91

G79 X20

G79 Y20

G79 X -20

G90

G81	Define bore cycle
G79	Execute the cycle at the absolute position (50.50)
G91	Switch to incremental measurement programming
G79	Execute the cycle at the incremental position X+20
G79	Execute the cycle at the incremental position Y+20
G79	Execute the cycle at the incremental position X+20
G90	Switch to absolute measurement programming

Program example X91=/Y91=:

G81 Y2 Z-10 F200 M3

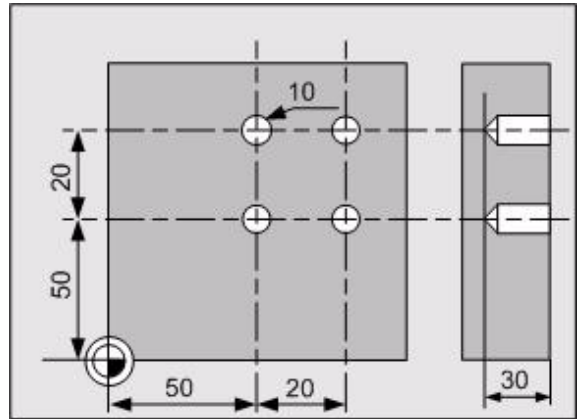
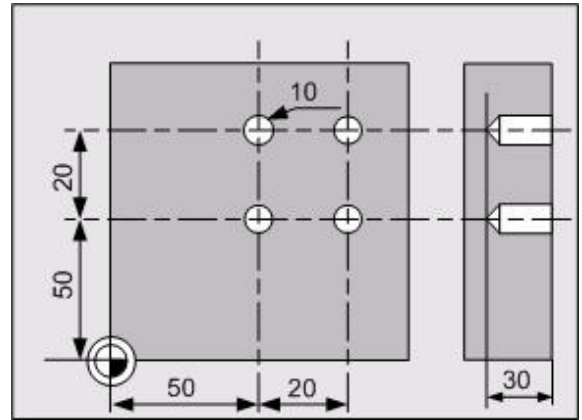
G79 X50 Y50 Z0

G79 X91=20

G79 Y91=20

G79 X91=-20

G81	Define bore cycle
G79	Execute the cycle at the absolute position (50.50)
G79	Execute the cycle at the incremental position X+20
G79	Execute the cycle at the incremental position Y+20
G79	Execute the cycle at the incremental position X+20



5.59 G92 Zero Point Shift Incr./Rotation

Zero point shift using incremental coordinate(s), relative to the last program zero point or a rotation of the coordinate system.

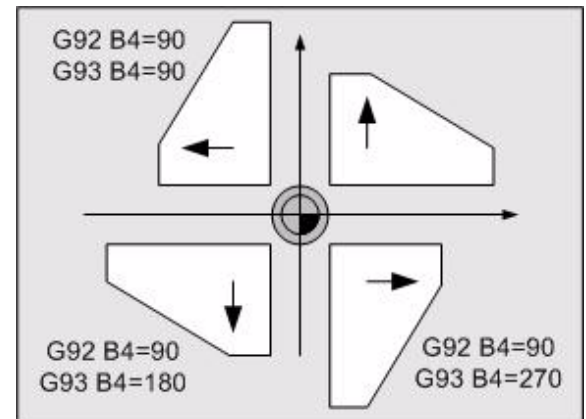
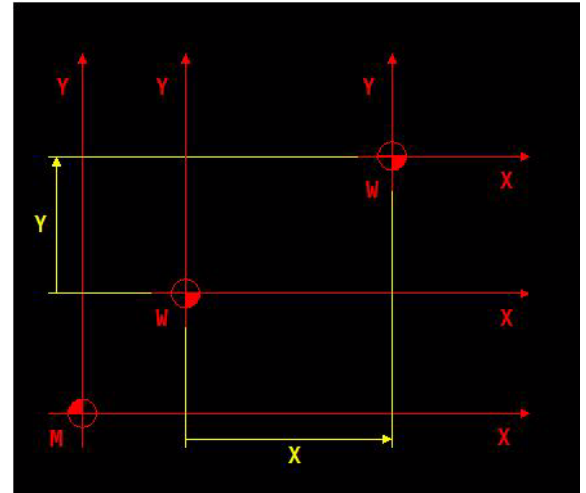
Address description

- ▶ X, Y, Z zero point coordinates
- ▶ B, C zero point angles
- ▶ B1= angle
- ▶ B4= angle of rotation incremental
- ▶ L1= path length

Application

Rotating the coordinate system (see figure)

- FSP: Approaching the tilting position by the shortest path
- FSP now always displays an angle between ± 180 and ± 180 degrees. This is changed so that an angle between the limit switches is displayed. This angle is then the shortest path. The disadvantage is that the position of the rotary axis can increase to very high values, which are to be turned back for a moment. The disadvantage of these very high positions is resolved by means of a separate function, with which the (internal) position is reset to a value between 0 and 360 degrees
- G93 {X}, {Y}, {Z}, {A}, {B}, {C}, {B2=}, {L2=}, {P}, {P1=}, {B4=}, {A3=1}, {B3=1}, {C3=1} where: A3=1, B3=1, C3=1
- The relevant axis position is reset to a value between 0 and 360 degrees. (see figure)



Reset function

A3=,B3=,C3= reset parameters. G93 A3=1 resets the relevant rotary axis position to a value between 0 and 360 degrees.

Example: an A axis with the position 370 degrees is changed to 10 degrees after the programming of G93 A3=1.

Notes

G92/G93 is effective from the machine zero point if no G54-G59 or G54 I... was previously activated.

A zero point shift programmed with G92/G93 is no longer allowed if rotation of the coordinate system (G92/G93 B4=...) is active.

Zero point shift (see figure)**Changes to V5xx**

■ See "G73_G92" on page 508.

Example**Program with G92: (see figure)**

G92 X40 Y50

G81 Y1 Z-12 M3

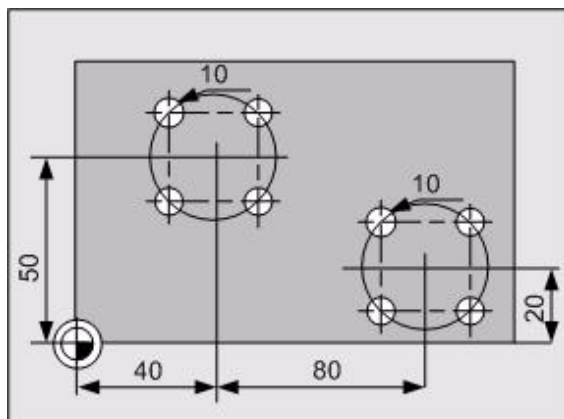
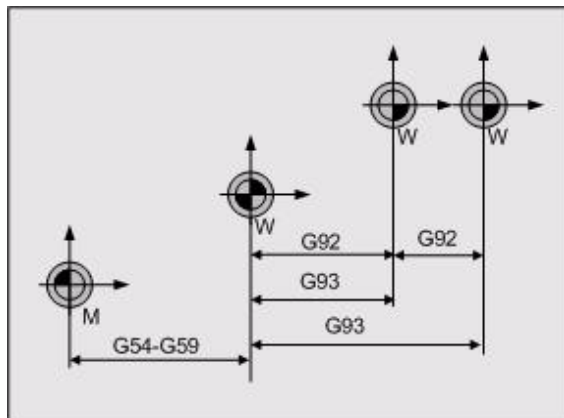
N8 G77 X0 Y0 Z0 I45 J4 R40

G92 X80 Y-30

G14 N1=8

G93 X0 Y0

G92	Incremental zero point shift
G81	Define the cycle
G77	Call the cycle
G92	Incremental zero point shift
G14	Repeat function
G93	Delete incremental zero point shift



5.60 G93 Zero Point Shift Abs./Rotation

Zero point shift using absolute coordinate(s), relative to the zero point (defined with G54-G59 or G54 I...) or a rotation of the coordinate system.

Address description

- ▶ X, Y, Z zero point coordinates
- ▶ B, C zero point angles
- ▶ B2= polar angle
- ▶ L2= polar length
- ▶ B3= 1=reset position 0-360 degrees
- ▶ C3= 1=reset position 0-360 degrees
- ▶ B4= angle of rotation absolute
- ▶ P1= point definition number

Application

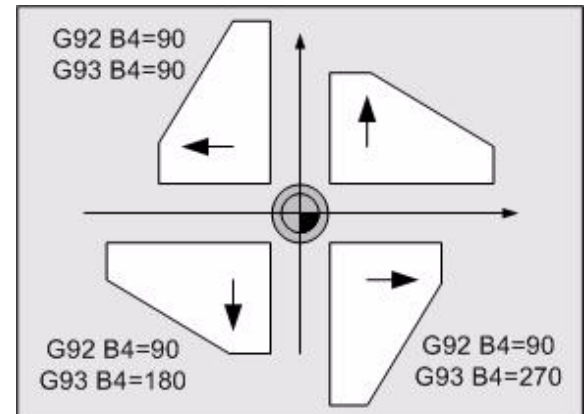
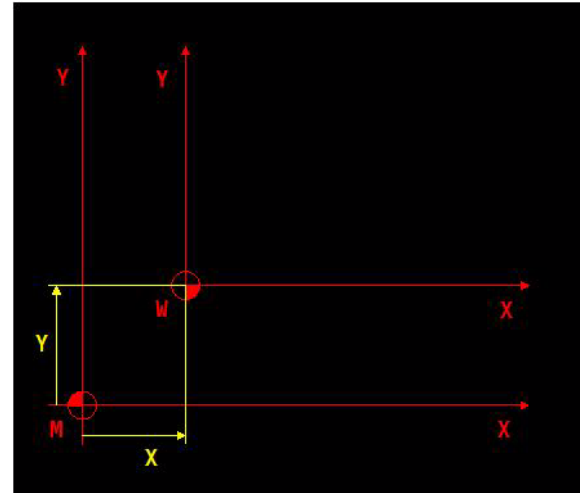
Rotating the coordinate system (see figure)

- FSP: Approaching the tilting position by the shortest path
- FSP now always displays an angle between =180 and +180 degrees. This is changed so that an angle between the limit switches is displayed. This angle is then the shortest path. The disadvantage is that the position of the rotary axis can increase to very high values, which are to be turned back for a moment.
- The disadvantage of these very high positions is resolved by means of a separate function, with which the (internal) position is reset to a value between 0 and 360 degrees.
- G93 {X}, {Y}, {Z}, {A}, {B}, {C}, {B2=}, {L2=}, {P}, {P1=}, {B4=}, {A3=1}, {B3=1}, {C3=1} where: A3=1, B3=1, C3=1
- The relevant axis position is reset to a value between 0 and 360 degrees. (see figure)

Reset function

A3=, B3=, C3= reset parameter. With G93 A3=1, the relevant rotary axis position is reset to a value between 0 and 360 degrees.

Example: an a axis with the position 370 degrees is changed to 10 degrees after the programming of G93 A3=1.



Notes

G92/G93 is effective from the machine zero point if no G54-G59 or G54 I... was previously activated.

A zero point shift programmed with G92/G93 is no longer allowed if rotation of the coordinate system (G92/G93 B4=...) is active.

Zero point shift (see figure)

Example

Program with G93: (see figure)

G93 X40 Y50

G81 Y1 Z-12 M3

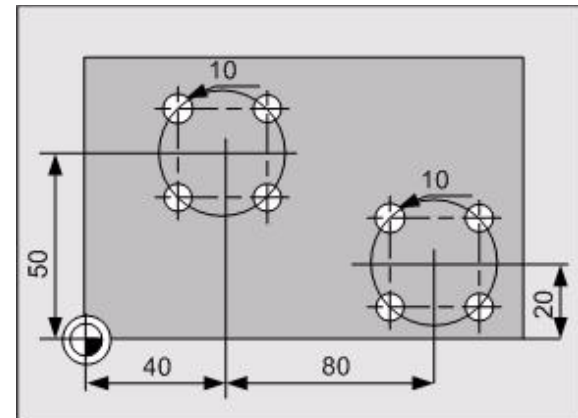
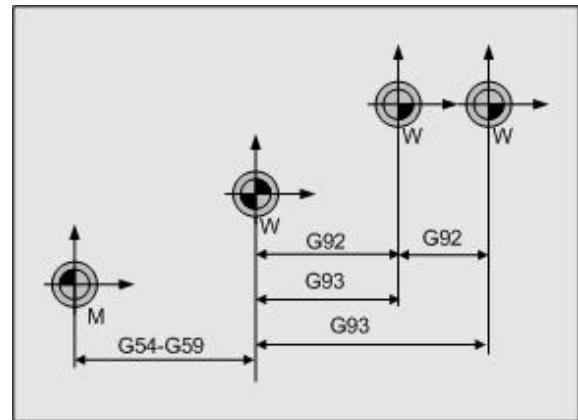
N8 G77 X0 Y0 Z0 I45 J4 R40

G93 X12 Y20

G14 N1=8

G93 X0 Y0

G93	Absolute zero point shift
G81	Define the cycle
G77	Call the cycle
G93	Absolute zero point shift
G14	Repeat function
G93	Delete absolute zero point shift



5.61 G94 Feed in mm/min (inch/min)

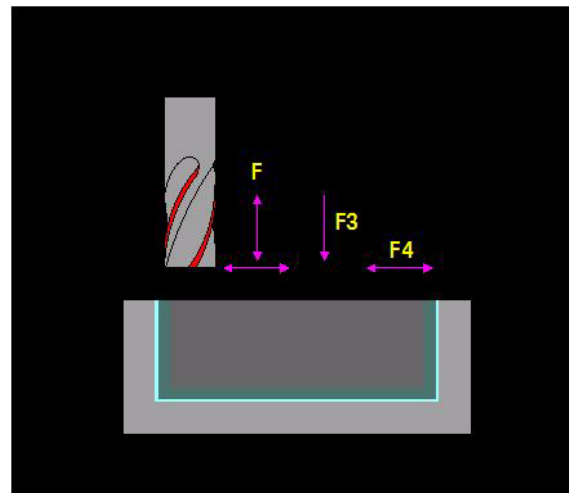
Information to the control that the programmed feed rate (F expression) is to be measured in mm/min or inch/min.

Address description

- ▶ F feed
- ▶ F1= F adaptation:1=red.,2=r/h,3=high
- ▶ F3= in depth feed
- ▶ F4= in plane feed
- ▶ F5= feed rotary axes

Format

- G94: feed rate in mm/min or inch/min
- G94 F5=: feed rate unit for the rotary axes



Application

Milling operation (G37): N... G94 F.. {S..} {M..}

S and M refer to the main spindle.

For rotary axes, the path in the space is calculated from the kinematic model. The feed rate is applied to this path.

Maximum speed

The maximum speed is specified for each gear range (M41-M44).

Feed rate F, F1, F3, F4

See Technology chapter.

Feed rate unit for rotary axes F5=

The unit for the modal feed rate for G1 is specified with G94 F5= for cases where only one rotary axis is programmed.

G94 F5=0 Degree/min (default setting)

G94 F5=1 mm/min or inch/min.

With F5=1 the speed on the current rotary axis radius is calculated. This is the distance from the tool to the center of the rotary axis.

G94 F5=1 is canceled with G94 F5=0, G95, M30, the **Reset CNC** soft key or the **Cancel program** soft key.

Example

Program example

```
G94
G1 X... Y... F200
```

G94	Feed rate in mm/min
G1	Advance to X... Y... with a feed rate of 200 mm/min

5.62 G95 Feed in mm/rev (inch/rev)

Information to the control that the programmed feed rate (F expression) is to be measured in mm/rev or inch/rev.

Address description

- ▶ F feed
- ▶ F1= F adaptation:1=red.,2=r/h,3=high
- ▶ F3= in depth feed
- ▶ F4= in plane feed

Format

G95: feed rate in mm/revolution or inch/revolution

Example

G95

G1 X... Y... F0.5

G95 Feed rate in mm/rev
G1 Advance to X... Y... with a feed rate of 0.5 mm/rev

Application

Milling operation (G37): N... G95 F.. {S..} {M..}

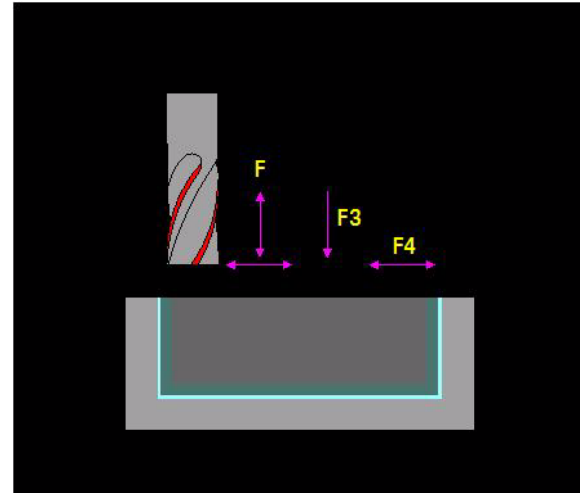
S and M refer to the main spindle.

For rotary axes, the path in the space is calculated from the kinematic model. The feed rate is applied to this path.

The G95 function calculates the feed rate in [mm/min (inch/min)] based on the programmed feed rate in [mm/rev], [inch/rev] and the active spindle speed.

Maximum speed

The maximum speed is specified for each gear range (M41-M44).



5.63 G97 Spindle Speed

Information to the control that the programmed spindle speed (S expression) is in rev/min. This switches G96 off.

Address description

- ▶ S speed (rev/min)
- ▶ M machine function
- ▶ S1= speed (rev/min)
- ▶ M1= machine function

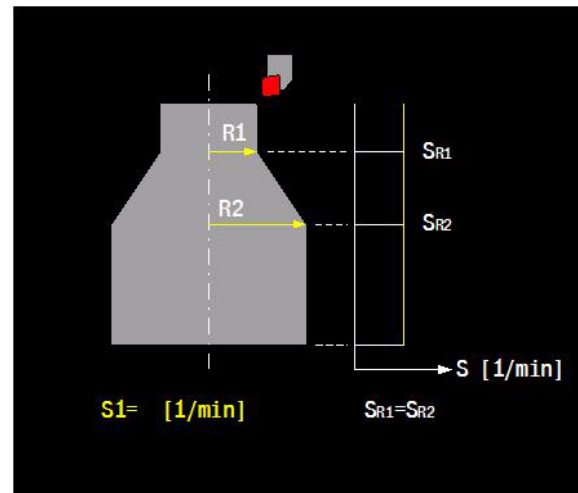
Format

- F.. {S..} {M..}
- S and M refer to the main spindle

Application

Maximum speed (s)

The maximum speed is specified for each gear range (M41-M44).



5.64 G98 Graphic Window Definition

Definition of the position relative to the program zero point W and the dimensions of a 3D-graphic window, which is to be used to display the machining of the workpiece by means of graphical simulation.

Address description

- ▶ X, Y, Z start point coordinates
- ▶ I dimension parallel to X
- ▶ J dimension parallel to Y
- ▶ K dimension parallel to Z

Format

G98 X... Y... Z... I... J... K...

Application

Changes to V5xx

- See "G98" on page 511.

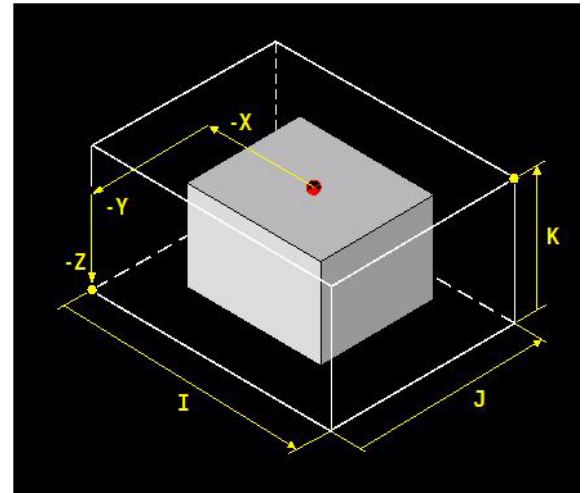
Example

Program example

```
G98 X-20 Y-20 Z-75 I140 J90 K95
```

```
G99 X0 Y0 Z0 I100 J50 K-55
```

- | | |
|-----|--|
| G98 | Starting point and dimensions of the 3D graphic window |
| G99 | Define the workpiece blank as a 3D area |



5.65 G99 Graphic Material Definition

Definition of a three-dimensional workpiece blank and its position relative to the program zero point W. The dimensions are required for the graphical simulation.

Address description

- ▶ X, Y, Z start point coordinates
- ▶ I dimension parallel to X
- ▶ J dimension parallel to Y
- ▶ K dimension parallel to Z

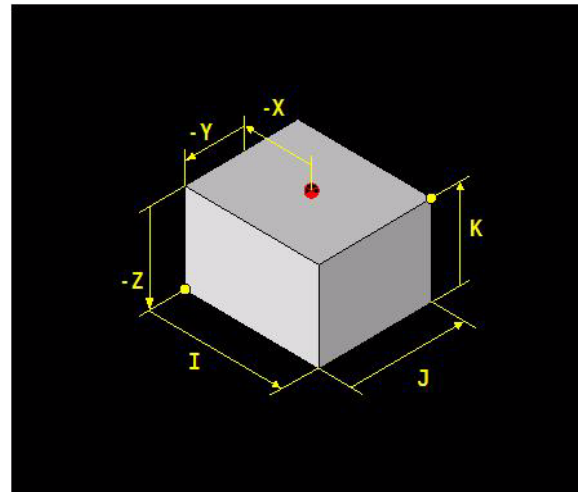
Example

Program example

```
G98 X-20 Y-20 Z-75 I140 J90 K95
```

```
G99 X0 Y0 Z0 I100 J50 K-55
```

- | | |
|-----|--|
| G98 | Starting point and dimensions of the 3D graphic window |
| G99 | Define the workpiece blank as a 3D area |



6

G100-G199 G-Codes

6.1 G125 Lifting Tool on Intervention: OFF

Deactivation of the tool lifting movement.

Address description

No specific addresses.

Application

Modality

G125 is modal with G126

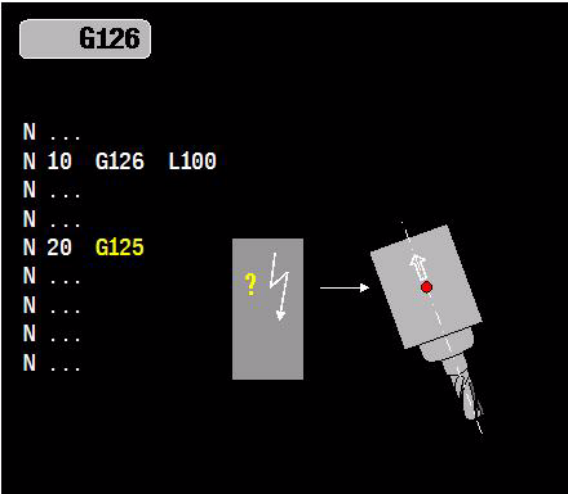
Execution

G125 resets the modal <Lifting permitted status> for the G126 function. No further lifting movement is possible after this.

- G125 is identical to G126 I1=0 I2=0 I3=0
- G125 causes <INPOD>.

Display

The G125/G126 functions are in the processing status display in the modal G series.



6.2 G126 Lifting Tool on Intervention: ON

G126 is a function which lifts the tool from the workpiece under certain conditions (coolant failure, intervention, and faults).

Format

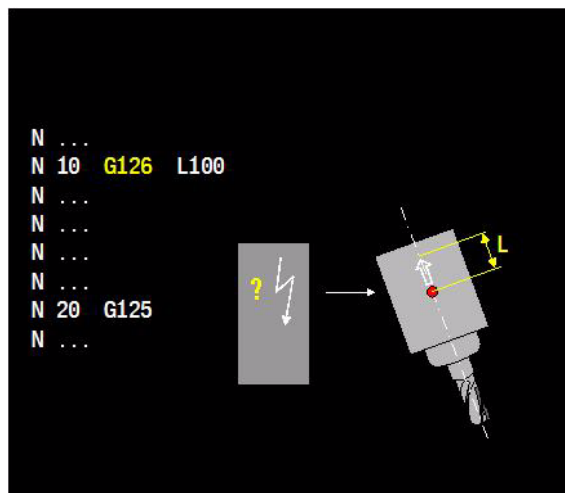
G126 {I1=...} {I2=...} {I3=...} {L...}

Address description

- **I1= lifting by PLC: 0=off,1=on** (e. g. coolant failure): 0= no lifting, 1=lifting.
- **I2= lifting on <INT>: 0=off,1=on** 0= no lifting, 1= lifting.
- **I3= lifting on error: 0=off,1=on** 0= no lifting, 1= lifting.
- **L lifting distance in tool direction** Defines the lifting distance in the direction of the tool or of the tool orientation (G36 turning). Default setting via: machine parameters CfgLiftOff/on and "G126 Lifting distance". The value lies between 0.000000000 and 2.000000000 [mm] or 0.0001 and 9999.9999 [inch].

Default setting

I1=1, I2=0, I3=0, L=**lifting distance**



Application

Modality

G125 is modal with G126.

Execution

G126 causes <INPOD>. A modal <Lifting permitted status> is then set.

The lifting movement is activated if:

- One of the actions described in the parameters I1-I13 (coolant failure, intervention, or error) takes place.
- G126 modal <Lifting permitted status> is activated.
- A feed rate is active. Lifting does not take place if the feed rate override is set to zero.
- For fixed cycles including when rapid traverse is active.
- Specific G functions are active.

Note: Machining stops even if the lifting movement is not activated. If, for example, WOX_RETRACT_TOOL is set during a rapid traverse, processing stops without a lifting movement.

The lifting movement takes place:

- In the programmed direction
- In tool direction (G37 "Milling operation", G126 L parameter, or default setting) or until the programmed lifting height or software limit switch is reached

After the lifting movement, the machining and the spindle stop with an (additional) error message I264 "Machining stopped with lifting movement".

Note: if the lifting movement is activated by an error (G126 I3=1) that also triggers the emergency stop, the drive motors are switched off before the lifting movement is completed.

Motion sequence

Before the lifting movement starts, the MillPlus decelerates until it reaches the correct (smooth) corner speed.

When the G126 function is active, the lifting function is not possible in the following G functions (not yet complete):

Movements	0, 6, 31, 33 Depending on G28 setting of feed movements
Planes	7, 182
Measuring cycles	45, 46, 49, 50, 145, 148, 149, 150
Positioning	74, 174
Fixed cycles	84, 86
Cycles	784, 786, 790, 793
Graphic	98, 99, 195, 196, 197, 198, 199
Pocket cycle	200, 201, 203, 204, 205, 206, 207, 208

Switching off G126

G126 ("Lifting tool on intervention: ON") is deactivated for <M30>,soft key **Cancel program**, G125 active, and soft key **CNC reset**.

Status display

The G125/G126 status is displayed in the modal G group display.

Block access

The functions G125 and G126 are saved during block searches and the last of these functions is carried out immediately after the block is accessed.

Interrupting the lifting movement

The lifting movement itself can be interrupted. However, it is not resumed after the interruption. A new <start> means returning to the contour.

Returning to the contour

After the lifting movement and additional error message, the normal options during interruption are possible. The system returns to contour by means of positioning logic.

Machine parameters

CfgLiftOff	Tool lifting distance
On	Lifting movements active
Off	Lifting movements not active
distance	0.000000000 to 2.000000000 [mm] default: 0.0 [mm]

G320 can be used to query the G126/G125 status and programmed distance (not yet available)

I1=72	Programmed statuses
0	G125
1	PLC (G126 I1=1)
2	INT (G126 I2=1)
3	PLC + INT (G126 I1=1 I2=1)
4	ERR (G126 I3=1)
5	PLC + ERR (G126 I1=1 I3=1)
6	INT + ERR (G126 I2=1 I3=1)
7	All (G126 I1=1 I2=1 I3=1)
I1=73	Programmed distance

Changes to V5xx

- See "G126" on page 513.

Example

Activate lifting function

```
G126 I1=1 I2=1
```

G126 Activate the lifting function by IPLC or interruption.

6.3 G141 3D Tool Correction

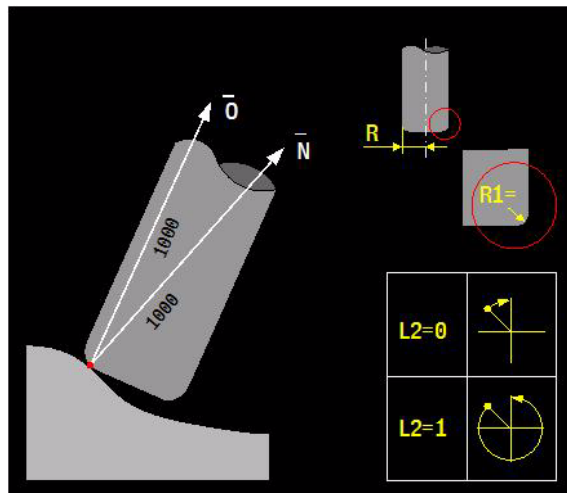
Allows the tool dimensions to be corrected for a 3D tool path, generated from short, straight sections, with 3-axis and 5-axis machining. Rotary axes can be programmed directly with an angle or indirectly with a tool vector. Radius compensation occurs if the normal vector is programmed in the end point. A typical application is the machining of free-form surfaces.

Address description

- ▶ **R nominal tool radius** Defines the tool radius used to calculate the end points of the G0/G1- blocks in the CAD- system.
- ▶ **R1= nominal tool corner radius** Defines the tool corner radius used to calculate the end points of the G0/G1- blocks in the CAD- system.
- ▶ **L2= rotary axes (0=shortest, 1=abs.)**
 - L2=0 rotary axes traverse the shortest route (default setting).
 - L2=1 rotary axes approach their absolute position (with rotary axis programming).
- ▶ **F2= feed limitation** Highly-curved surfaces can cause the rotary axes to move abruptly at maximum feed. F2= limits this feed rate. F2= is programmed in the G141 block and acts for all G0/G1 movements up to the block with G40.

With G0/G1

- ▶ **X, Y, Z linear end point coordinates**
- ▶ **I, J, K axis components of the surface normal vector**
- ▶ **I1=, J1=, K1= (TCPM) axis components of the tool vector**
- ▶ **A, B, C (TCPM) rotary axis coordinates of the tool vector**
- ▶ **F feed rate on the path**



Format

3-axis machining with normal vector (I,J,K) for radius compensation:

- G141 {R...} {R1=...} {L2=...} {F2=...}
- G0/G1 [X..Y.. Z..] [I... J... K...]

5-axis machining with TCPM (Tool Center Point Management). Normal vector (I,J,K) for radius compensation.

- G141 R.. {R1=..} {L2=..} {F2=..}
- G0/G1 [X..Y.. Z..] [I.. J.. K...] {I1=.. J1=.. K1=..}/{A.. B.. C..}

Canceling 3D- tool compensation:

- G40

Default setting

G141 L1=0 R1=0 R=0

Application

5-axis machining of a curved workpiece surface involves guiding the tool to the surface at an optimized angle. Dynamic TCPM is used for this 5-axis machining and guides the rotary axes and linear axes, allowing for current tool length and tool radius. In the G0/G1 block, the rotary axes can be programmed directly with angle (A,B,C) or indirectly with a tool vector (I1=, J1=, K1=). The radius compensation is calculated by MillPlus if the normal vector (I, J, K) is programmed in the G0/G1 block.

- N = normal vector (I, J, K) (see figure)
- O = tool vector (I1=, J1=, K1=)

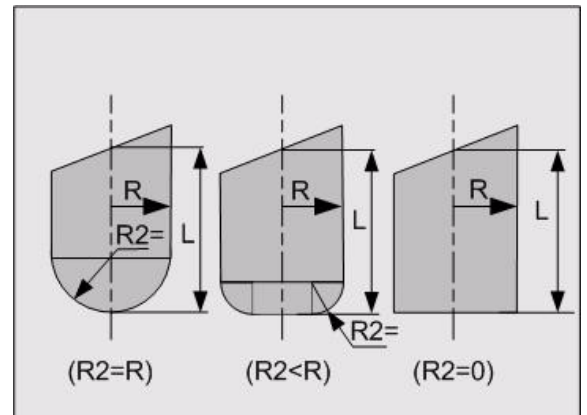
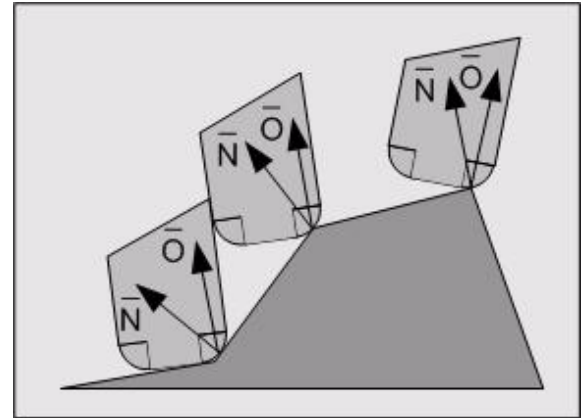
G7 can be active. In this case, the normal- and tool vectors are defined in the G7- plane.

Possible tools

The following dimensions must be loaded in the tool memory for use of the various tool types (see figure):

Radius cutter:	R (tool radius)
	L (tool length)
	R2 (rounding radius) R2=R
Radius end milling tool:	R (tool radius)
	L (tool length)
	R2 (rounding radius) R2<R
End milling tool:	R (tool radius)
	L (tool length)
	R2=0

If no R2 value is entered, R2 automatically becomes 0.





The rounding radius in the G141 block is programmed with the expression $R1=$. The $R2$ expression is used to store the rounding radius in the tool memory.

Radius compensation

The radius compensation is calculated by MillPlus if the normal vector (I, J, K) is programmed in the G0/G1 block. The radius compensation is not activated if no normal vector is programmed.

The tool is positioned so that this vector always passes through the center point of the corner rounding. If the end points are calculated in the CAD/CAM system with the nominal radius and corner radius, this can be defined in the G141 block using R and $R1=$. The true radius R and corner radius $R2$ are then entered into the tool table. The control corrects the difference between the nominal and actual radius.

R radius defines the tool radius used to calculate the end points of the G0/G1- blocks in the CAD-system.

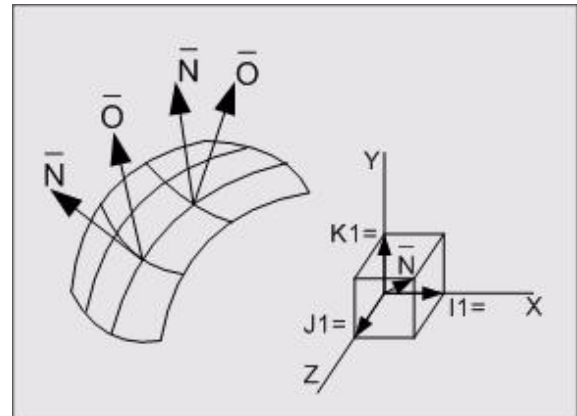
R1= radius defines the tool corner radius used to calculate the end points of the G0/G1- blocks in the CAD-system.

Normal vector (I, J, K)

The normal vector is perpendicular to the workpiece surface. I,J,K are the vector components in directions X,Y,Z. The tool is positioned so that this vector always passes through the center point of the tool corner rounding. See figure.

Tool vector (I1=, J1=, K1=) (TCPM)

This vector points towards the tool axis. I1=,J1=,K1= are the vector components in directions X, Y, Z. The tool vector can be programmed in the G0/G1 block instead of the rotary axes. During the movement, the rotary axes A, B, C and linear axes are interpolated so that a straight line is generated in the machining area. The tool points towards this vector at the end point of the movement.



Vector components

A vector is programmed with at least one component in the G0/G1 block. Unprogrammed components are equal to zero.

Vector accuracy

The vector components are programmed with up to 9 places. The input format for the vectors (I, J, K) and (I1=, J1=, K1=) is limited to seven decimal places. Six decimal places should be programmed to achieve sufficient dimensional accuracy. However, the normal- and tool vectors do not need to have length 1.

Activating G141

In the first block after G141, the milling tool traverses from the current tool position to the corrected position in this block. G141 deletes a radius compensation programmed using G41..G44.

Canceling G141

The function G141 is canceled using G40, M30, the Cancel program soft key, or the CNC reset soft key. The milling tool stops at the last corrected position. The rotary axes are not turned back automatically.

Switch on condition before G141

Before switching on G141, the following functions must be switched off: Geometry G64, Scale change G73 A4=, Axis rotation B4= with G54-G59, G54 I.., G92/G93, cylinder coordinates G182

The following functions are permitted when G141 is switched on:

Basic motions	0, 1
Free working plane	7
Planes	17, 18
Program control	14, 22, 23, 29
Positioning feed rate	25, 26, 27, 28, 94, 95, 96, 97
Tool dimension	39
Radius compensation	40
Zero points	51, 52, 53, 54-59, 54 I.., 92, 93
Geometry	72, 73
Absolute/incremental	90, 91
Graphic	195, 196, 197, 198, 199

The following G-functions are permitted if G141 is **active**:

Basic motions	0, 1
Program control	14, 22, 23, 29
Positioning feed rate	4, 25, 26, 27, 28, 94, 95, 96, 97
Radius compensation	40
Zero points	51, 52, 53, 54-59, 54 I.., 92, 93
Geometry	72, 73
Absolute/incremental	90, 91

An error message is issued if a G-function that is not permitted is programmed.

Programming limitations

G-functions not listed above must not be used. Point definitions (P) must not be used. No tool change must be made after activating G141.

End point coordinates

Absolute or incremental (X, X90, X91) Cartesian dimensions can be used.

G1

When a tool vector I1=, J1=, K1= is being programmed, G0 or G1 must be in the same block.

Mirroring

If the mirroring function (G73 and axis coordinates) is effective before G141 is activated, the mirrored coordinates will be used during the 3D-tool compensation. Mirroring is re-enabled once G141 is activated. Mirroring is canceled using the function G73.

Undercuts

Undercuts or collisions between the tool and material at points not to be machined are not detected by the CNC.

Displaying the rotary axis position

If a rotary axis has been defined as the rollover axis, it is possible for positions greater than 360° to be displayed in the position window. The position is reset between 0° and 360° by programming G141.

Behavior of the rotary axes at the limit switches

If the rotary axes on G141 are programmed directly with A.. B.. C.., an error message is issued if the programmed position lies past the limit switch.

Selecting a solution with vector programming

If the rotary axes are programmed via the tool vector I1=, J1=, K1=, there are often two solutions for the rotary axis positions. Selecting a solution:

- The solution that lies past the limit switch is invalid.
- The solution that goes beyond the limit switch of a linear axis during interpolation is invalid.
- If two solutions are valid, the solution with the shortest route is selected, even when L2=1 (rotary axis absolute).
- If both solutions are invalid, an error message is displayed indicating that the programmed level cannot be reached.

End point coordinates

With end point coordinates, only the programmed axes are moved.

Changes to V5xx

- See "G141" on page 514.

Example

Example 1: G141 and TCPM

Tool vector with (I1=..., J1=..., K1=...)

This programming is **independent** of the machine.

N113 (RECTANGLE MATERIAL WITH UPPER ROUNDINGS (R4) AND A TILTED TOOL (5 DEGREES))

G17

T6 M67 (ROUND SPHERICAL CUTTER 10: IN TOOL TABLE T6 R5 C5)

G54 I10

G0 X0 Y0 Z0 B0 C0 S6000 M3

F50 E1=0

G141 R0 R1=0 L2=0 (ALL DEFAULT SETTINGS, DO NOT HAVE TO BE PROGRAMMED)

(R IS 0 MM IN CAD SYSTEM)

(R1 IS 0 MM IN CAD SYSTEM)

(L2=0 ROTARY AXES TRAVEL THE SHORTEST PATH)

G0 X-1 Y=E1 Z0 I1=-1 K1=0

(GENERATED IN THE CAD SYSTEM)

(ARC FRONT LEFT)

G1 X0 Y=E1 Z-4 I1=-0.996194698 K1=0.087155743

G1 X0.000609219 Z-3.930190374 I1=-0.994521895
K1=0.104528463

G1 X0.002436692 Z-3.860402013 I1=-0.992546152
K1=0.121869343

G1 X0.005481861 Z-3.790656175 I1=-0.990268069
K1=0.139173101

N.... (EACH DEGREE A POINT)

G1 X3.790656175 Z-0.005481861 I1=0.034899497
K1=0.999390827

G1 X3.860402013 Z-0.002436692 I1=0.052335956
K1=0.998629535

G1 X3.930190374 Z-0.000609219 I1=0.069756474
K1=0.99756405

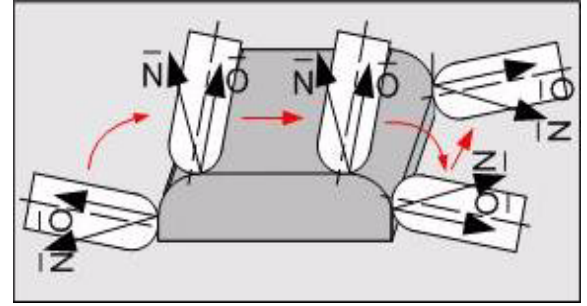
G1 X4 Z0 I1=0.087155743 K1=0.996194698

(ARC FRONT RIGHT)

G1 X36 Z0 I1=0.087155743 K1=0.996194698

G1 X36.06980963 Z-0.000609219 I1=0.104528463
K1=0.994521895

G1 X36.13959799 Z-0.002436692 I1=0.121869343
K1=0.992546152



Example 2: G141 and TCPM

Identical workpiece. Tool vector with (A, B, C)

This programming is **dependent** on the machine.

This program is for a machine with a B axis less than 45° on the table and a C axis above it.

N114 (RECTANGLE MATERIAL WITH UPPER ROUNDINGS (R4) AND A TILTED TOOL (5 DEGREES))
G17
T6 M67 (ROUND SPHERICAL CUTTER 10: IN TOOL TABLE T6 R5 C5)
G54 I10
G0 X0 Y0 Z0 B0 C0 S6000 M3
F50 E1=0
G141 R0 R1=0 L2=0 (ALL DEFAULT SETTINGS, DO NOT HAVE TO BE PROGRAMMED)
(R IS 0 MM IN CAD SYSTEM)
(R1 IS 0 MM IN CAD SYSTEM)
(L2=0 ROTARY AXES TRAVEL THE SHORTEST PATH)

G0 X-1 Y=E1 Z0 B180 C-90
(GENERATED IN THE CAD SYSTEM)
(ARC FRONT LEFT)
G1 X0 Y=E1 Z-4 B145.658 C-113.605
G1 X0.000609219 Z-3.930190374 B142.274 C-115.789
G1 X0.002436692 Z-3.860402013 B139.136 C-117.782
G1 X0.005481861 Z-3.790656175 B136.191 C-119.624
N.. (EACH DEGREE A POINT)
G1 X3.790656175 Z-0.005481861 B2.829 C1
G1 X3.860402013 Z-0.002436692 B4.243 C1.501
G1 X3.930190374 Z-0.000609219 B5.658 C2.001
G1 X4 Z0 B7.073 C2.502
(ARC FRONT RIGHT)
G1 X36 Z0 B7.073 C2.502
G1 X36.06980963 Z-0.000609219 B8.489 C3.004
G1 X36.13959799 Z-0.002436692 B9.906 C3.507

6.4 G145 Linear Measuring Movement

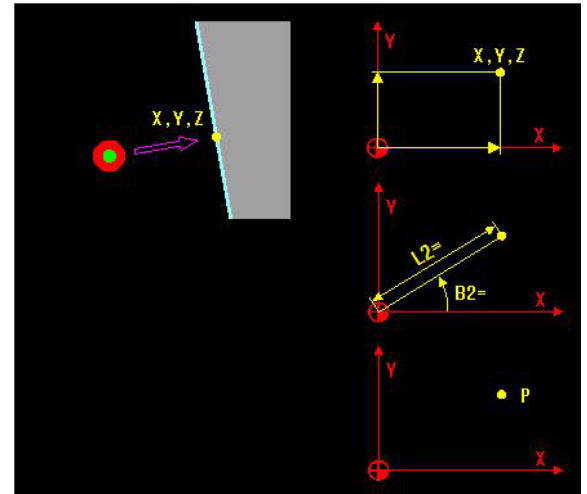
Execution of a freely programmable linear measuring movement to determine axis positions.

Format

G145 [measuring point coordinates] [(axis address) 7=...] {S7=...}
{O1=...} {O2=...} {O3=...} {O4=...} E... {F2=...} {K...}

Address description

- ▶ **X,Y,Z end point coordinates**
- ▶ **B,C end angles**
- ▶ **K 0=tool correction on, 1=off** The following assumptions apply when the measuring positions are corrected with regard to the measure probe dimensions: the measure probe is arranged parallel to the tool axis, the measure probe is completely round, the measure probe is moved perpendicular to the surface to be measured.
 - **K=0 tool correction on** Measuring positions are corrected with regard to tool length and tool radius. Measuring positions in rotation axes are not corrected with regard to tool data.
 - **K=1 tool correction off** Measuring positions are not corrected.
 - **K=2 tool correction on** Measuring positions are only corrected with regard to tool length. Measuring positions in rotation axes are not corrected with regard to tool data.
- ▶ **B1= angle**
- ▶ **B2= polar angle**
- ▶ **X7....S7 E parameter for measured value in X,Y,Z,B,C,S**
- ▶ **?90 absolute end point angle (X,Y,Z..)**
- ▶ **?91 incremental end point angle (X,Y,Z..)**
- ▶ **L1= path length**
- ▶ **L2= polar length**
- ▶ **P1= point definition number**
- ▶ **F2= measuring feed**
- ▶ **I4= blow air** 0=no 1=yes. The air blowing period is determined by the PLC.



- ▶ **O1= E parameter for error "No measuring target"** Defines the E parameter number that states whether the measuring target was found after the measurement. If the O1= address is programmed, the error message "No measuring target found" is no longer issued by the control but is still written in the defined E parameter.
 - E(O1=) = 0 = measuring target was found
 - E(O1=) = 1 = measuring target was not found.
- ▶ **O2= E parameter for error "Probe deflected"** Defines the E parameter number that states whether the touch probe was deflected at the start of the measurement. If the O2= address is programmed, the error message "Probe deflected" is no longer issued by the control but is still written in the defined E parameter.
 - E(O2=) = 0 = touch probe was not deflected
 - E(O2=) = 1 = touch probe was deflected
- ▶ **O3= E parameter for the status measured value** Defines the E parameter number that specifies the status of the measured value.
 - E(O3=) = 0 = measured value is measured position
 - E(O3=) = 1 = measured value is programmed end position
 - E(O3=) = 2 = no value
- ▶ **O4= E parameter for block access operating mode** Defines the E parameter number that specifies whether block access was active.
 - E(O4=) = 0 = block access was not active
 - E(O4=) = 1 = block access was active

Application

Associated functions

G148, G149, G150. The functions G148 to G150 must not be used during operation with G182.

Measurement feed (F2=)

If F2= is not programmed, the measurement feed from the PROBE_FEED columns in the tool table is automatically uploaded.

Interruption

The G145 movement is processed as a G1 movement during an interruption. The measure probe status should not be changed between the starting point of the measurement movement and the point of interruption, otherwise an error message is issued. An error message is also raised if the touch probe is triggered when returning to the contour.

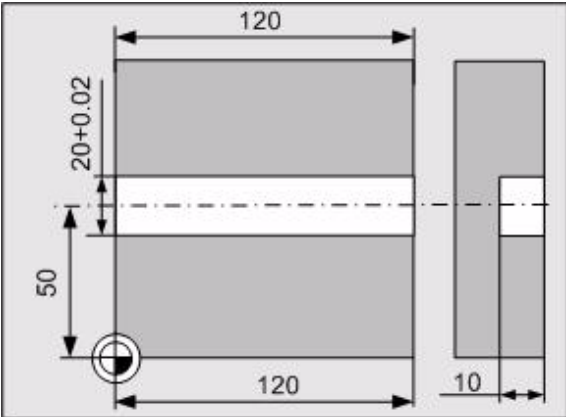
Changes to V5xx

- See "G145" on page 515.

Example

Milling and measuring a slot

A slot is to be milled and its width is to be measured. If the width of the slot is too small, the milling radius must be corrected and the slot must be reworked.



G99 X0 Y0 Z0 I120 J100 K-20	
T1 M6	D=18 mm
S1000 M3	
G0 X-25 Y50 Z-10	Start slot milling
F1400 M8	
G1 X140	
G43	
Y60	
G41	
X-25 Y40	
X140	
G40	
Y50	
G0 Z50	
M5	
M9	End slot milling
G149 T0 E4	
E1=20 E2=19.98 E3=20.02	Nominal, minimum, maximum width
T999 M6	
M27	
G0 X60 Y50 Z0	
G145 Z-4 I4=1 O1=10 F2=1000	Blow air, check the obstruction of the slot
IF E10=0 THEN	Slot obstructed if touch probe has been triggered:
M0	Slot blocked, remove chips, start

END IF	
E11=60 E12=40.05	Test
G0 Y54 Z-4	
G145 Y62 Y7=11 I4=1 O1=99	Measure first side
G0 Y46	
G145 Y38 Y7=12 I4=1 O1=99	Measure second side
G0 Y50	
Z50	
E13=E11-E12	Measured width
E14=(E13-E1)/2	Corrected tool radius
WHILE E13>E3	Width > maximum?
M0	Slot width outside tolerance, cancel program
END WHILE	
G321 T=E4 I1=5 E15	Read DR
IF E15 IS NOTHING THEN	
E15=0	Default setting for DR
END IF	
E16=E15+E14	New DR
G331 T=E4 I1=5 E16	Write new DR
T=E4 M67	Activate new DR
IF E13<E2 THEN	Machine slot again if width < minimum
T1 M6	D=18 mm
S1000 M3	
G0 X140 Y50 Z-10	
F400 M8	
G43	
G1 Y60	
G41	
X-25 Y40	
X140	
G40	
Y50	
G0 Z50	
M5	Machine end again
END IF	
M28	
M30	

6.5 G148 Read Measure Probe Status

Export of measure probe status within the measuring cycle macro.

Format

G148 {I1=...} E...

Address description

► I1= status group (1-2)

I1=1 Active mode.

- E...=0 Measure probe not deflected. Default setting.
- E...=1 Measure probe deflected.
- E...=2 The G145-block was executed during the block search run, test run, or demo operation.
- E...=3 A measure probe error has occurred; no measuring process possible

I1=2 Measure probe error.

- E...=0 No measuring point was determined during the measurement.
- E...=1 A measuring point was determined during the measurement.

► E E parameter for probe status

The priority for the probe status is:

- E...=2 Active mode
- E...=3 Measure probe error
- E...=0 / 1 Measure probe contact

Application

Associated functions

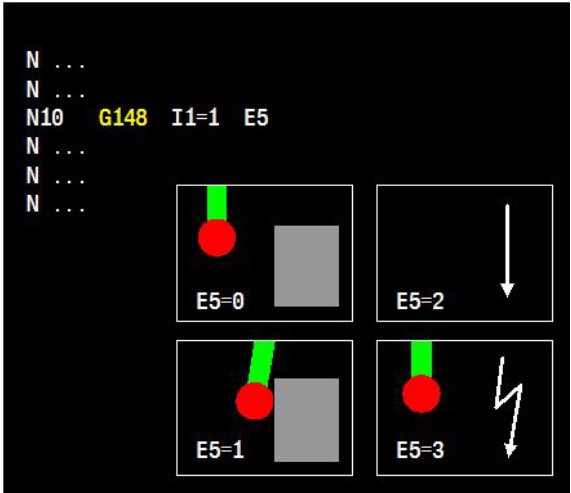
G145, G149, G150

(Modal) G functions that are not allowed when the function is used

■ G64, G141, G182, G195, G197, G198, G199, G280-G286

Changes to V5xx

■ See "G148_I1=3" on page 516.



Example

Read measure probe status

```
G148 I1=1 E27
G29 E91=E27=2 E91 N=300
-----
N300 M0
```

- G148 Save measure probe status in E parameter 27.
- G29 Jump to block N300 if the program is executed in the block search run, test run, or demo operation. This avoids e. g. calculations with parameters that have not been loaded because no measurement has been made.

6.6 G149 Read Tool- or Zero Offset Values

Querying of tool and zero offset data and saving this data in the E parameter.

Querying tool data

Format



We recommend that you use G321 (Read tool data) to read tool data. This G function contains more options.

Querying the active tool number:

■ G149 T0 E...

Querying the tool dimensions and tool life:

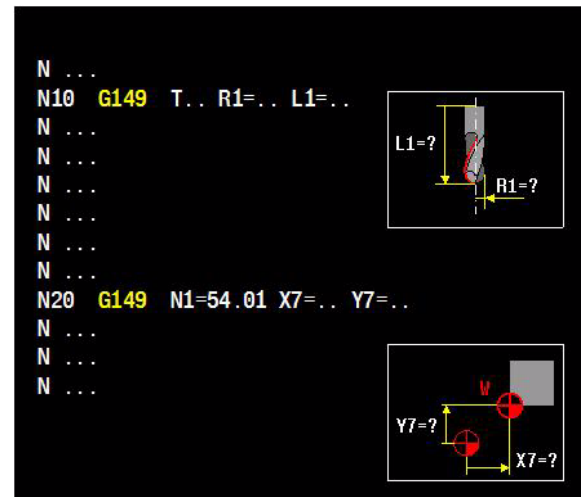
■ G149 T... {T2=...} {L1=...} {R1=...} {M1=...}

Querying the tool status:

■ G149 T... {I1=...} {I2=...}

Address description

- ▶ **T tool number** Number of the tool whose tool data is to be exported. The complete tool number (including the spare tool index) must be specified.
Note: T0 reads the number of the active tool. The tool data from T0 cannot be exported. The relevant E-parameters are not loaded when T0 is used. No corresponding error message is issued.
- ▶ **T2= tool offset index** The tool compensation index 0 to 9 can be specified.
- ▶ **I1= E parameter for tool locked** The tool lock TL is saved in the specified E parameters. Result:
 - 0 tool is not locked.
 - 1 tool is locked.



- ▶ **I2= E parameter for tool status** The tool status TS is saved in the specified E-parameters. Result:
 - 0 tool is released but not measured.
 - 1 tool is released and measured.
 - 2 tool outside tolerance.
 - 3 tool breakage.
 - 4 tool life expired.
- ▶ **L1=, R1= E parameter for tool length and radius** The tool length and tool radius are saved in the specified E parameters. Result:
 - Tool radius = radius (R) + allowance (DR).
 - Tool length = length (L) + allowance (DL).
- ▶ **M1= E parameter for tool life** The current tool life CUR_TIME is saved in the specified E parameters.

Default setting

T2=0

Application

- Associated functions: G145, G148, G150, G321, and G326
- Exporting addresses with no value. If addresses that were not input beforehand are exported in the tool file, a zero value is returned in the specified E parameters.
- G149 is not allowed after the following modal functions: G64, G141, G280-G286

Changes to V5xx

- See "G149" on page 516.

Example

1: Querying the number of the active tool

```
G149 T0 E1
```

E1 E1 contains the number of the active tool

2: Querying the tool dimensions

```
G149 T12 L1=5 R1=6
```

L1=5 E5 contains the sum of the tool length (L) + allowance (DL)

R1=6 E6 contains the sum of the tool radius (R) + allowance (DR)

3: Querying the current tool life

```
G149 T12 M1=3
```

M1=3 E3 contains the current tool time CUR_TIME from T12

4: Querying the tool status

```
G149 T12 I1=12 I2=13
```

I1=12 E12 contains the tool lock TL

I2=13 E13 contains the tool status TS

Querying Zero Offset Values

Format



The best way to read zero offset values is to use G320 (read actual G data) (I1=21 to I1=47). This G function contains more options.

Querying the active pallet offset number:

■ G149 N1=0 E...

Querying the active zero offset number:

■ G149 N1=1 E...

Querying saved zero offset values:

■ G149 N1=54...59 (axis address)7=... {(axis address)7=...}

■ G149 N1=54.[NR] (axis address)7=... {(axis address)7=...} {B47=...}

Querying programmable zero offset values:

■ G149 N1=92 {93} [(axis address)7=...] {(axis address)7=...}

Address description

► N1= zero point shift

- **N1=0 zero offset shift group G52** If G52 is active, then the E-parameter is given the value 52. If G52 is not active, then the E-parameter is given the value 51.
- **N1=1 zero offset shift group G54** The E-parameter is given the value of the effective offset from the range G54..G59 or G54.[no.]. The E-parameter is given the value 53 if no offset is effective.
- **N1=54-59 or 54. [no.] zero offset shift** The number of the saved zero offset G54 - G59, or G54 I[no.] whose data is to be exported.
- **N1=92-93 zero offset shift** The number of the programmable zero offset G92 or G93 whose data is to be exported.

- **X7=,Y7=,Z7=,A7=,B7=,C7=, B47= E par. for offset/position values** X7= E-parameter for offset/position in X. B47= E-par. for rotation in B4= The zero offset values for the zero offset specified in N1= are saved in the specified Eparameters.

Application

- Associated functions: G145, G148, G150, G321, and G326
- Exporting addresses with no value. If addresses that were not input beforehand are exported in the zero point shift memory, a zero value is returned in the specified E parameters.
- G149 is not allowed after the following modal functions: G64, G141, G280-G286

Example

1: Querying the active zero offset number

```
G149 N1=0 E2
G149 N1=1 E3
```

N1=0 E2 E2 contains the active zero point shift (51 or 52)
N1=1 E3 E3 contains the active saved zero offset (53...59) or 54 [no.]

2: Querying the zero offset G54 or G54 I1

```
G149 N1=54 X7=1 Z7=2
G149 N1=54.01 X7=1 Z7=2
```

X7=1 E1 contains the offset in X
Z7=2 E2 contains the offset in Z

3: Querying offset G54 with angle of rotation (only G54 I[no])

```
G149 N1=54.01 X7=1 B47=2
```

X7=1 E1 contains the offset in X
B47=2 E2 contains the angle of rotation for the coordinate system

Querying Current Position Values

Format



We recommend that you use G326 (read actual position) to read current position values. This G function contains more options.

Querying the current position values for the axes.

G149 (axis address)7=... {(axis address)7=...}

Address description

- **X7=,Y7=,Z7= current position values** the axis position values can be exported to E parameters. X7=20 means: the current axis position value is saved in E20.

Application

- G149 is not allowed after the following modal functions: G64, G141, G280-G286

6.7 G150 Change Tool- or Zero Offset Values

Changing of data in the tool and zero offset table.

Changing of tool data

Format



We recommend that you use G331 (write tool data) to change tool data. This G function contains many more options.

Changing tool data:

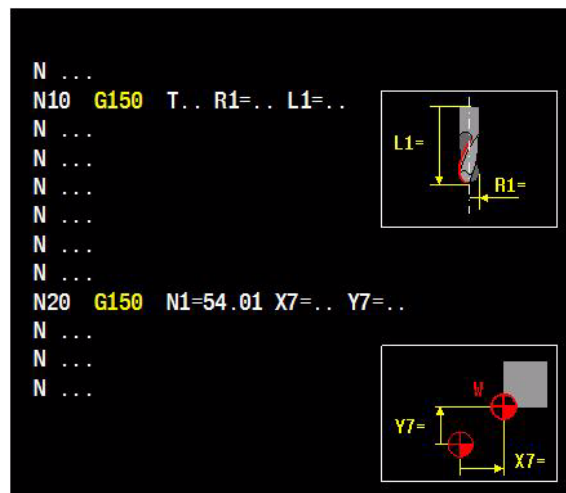
■ G150 T... {T2=...} {L1=...} {R1=...} {M1=...}

Changing the tool status:

■ G150 T... {I1=...} {I2=...}

Address description

- ▶ **T tool number** Number of the tool whose data is to be changed. The complete tool number (including the spare tool index) must be specified.
Note: the tool data of T0 cannot be changed.
- ▶ **T2= tool offset index** The tool offset index 0 to 9 can be specified.
- ▶ **I1= Tool locked** The specified value is written in the tool lock TL.
Values:
 - 0 tool is not locked.
 - 1 tool is locked.
- ▶ **I2= tool status value in T** The specified value is written in the tool status TS. Values:
 - 0 tool is released but not measured.
 - 1 tool is released and measured.
 - 2 tool outside tolerance.
 - 3 tool breakage.
 - 4 tool life expired.
- ▶ **L1=, R1= tool length and radius values in T** The specified values are written in the tool length L and in the tool radius R. The allowance (DL or DR) is set to zero.
- ▶ **M1= current tool life** The specified value is written in the current tool life CUR_TIME.



Default setting

T2=0

Application

- Associated functions: G145, G148, G149
- G150 is not allowed after the following modal functions: G64, G141, G280-G286

Changes to V5xx

- See "G150_T_M1" on page 517.

Example

Example 1: changing tool data

```
G150 T1 L1=E1 R1=4
```

L1=E1 Write value E1 in the tool length L

R1=4 Write value 4 in the tool length R

Example 2: changing the current tool life CUR_TIME

```
G150 T1 M1=10
```

M1=10 Write value 10 minutes in the current tool life
CUR_TIME

Changing Zero Offset Values

Format



We recommend that you use G54 I[no.] for changing values in the zero offset table. This G function contains more options.

Changing zero offset values

- G150 N1=52 or 54...59 (axis address)7=... {(axis address)7=...}
- G150 N1=54.[NR] (axis address)7=... {(axis address)7=...} [B47=...]

Address description

- ▶ **N1= zero point shift**
 - **N1=52 zero offset shift** The zero offset G52 to be changed
 - **N1=54-59 or 54.[no] zero offset shift** The zero offset G54 - G59 or G54.[no.] to be changed.
- ▶ **X7=,Y7=,Z7=,A7=,B7=,C7=, B47= E parameter for offset values**
X7= offset in X. B47= angle of rotation in B4=. The specified values are written in the zero offset specified with M1=.

Application

- Associated functions: G145, G148, G149
- G150 is not allowed after the following modal G functions G64, G141, G280-G286

Example

Changing zero offset values

```
G150 N1=57 X7=E1 Z7=E6
```

```
G150 N1=54.01 X7=E1 Z7=E6
```

X7=E1 Z7=E6 X and Z are written in the zero offset table.

Changing a zero shift with the rotation angle of the coordinate system

```
G150 N1=54.01 X7=E1 B47=E2
```

X7=E1 Write value E1 in the offset X
B47=E2 Write value E2 in the angle of rotation B4=

6.8 G151 Cancel G152

Cancel G152.

Format

G151

Address description

No addresses.

Application

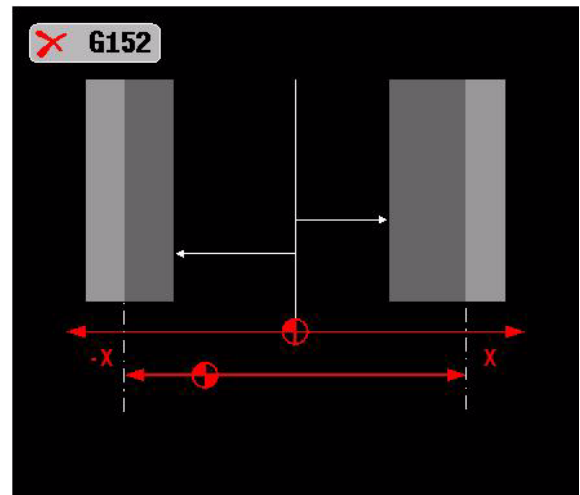
G152 is deactivated with this function.

Associated functions

G152.

Changes to V5xx

■ See "G151 und G152" on page 517.



6.9 G152 Limiting the Traverse Ranges

Limitation of the range of traverses. The programmed positions relate to the reference point.

Format

G152 X1=... Y1=... Z1=... {B1=...} {B2=...} X2=... Y2=... Z2=... {C1=...} {C2=...}

Address description

- ▶ X1= range in positive direction
- ▶ Y1= range in positive direction
- ▶ Z1= range in positive direction
- ▶ B1= range in positive direction
- ▶ C1= range in positive direction
- ▶ X2= range in negative direction
- ▶ Y2= range in negative direction
- ▶ Z2= range in negative direction
- ▶ B2= range in negative direction
- ▶ C2= range in negative direction

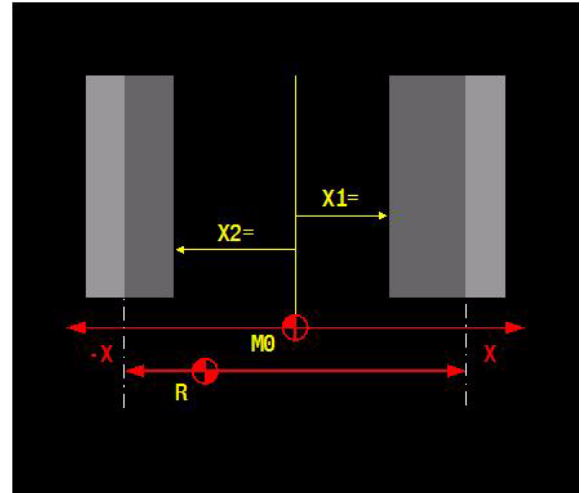
Application

This function enables the traverse range to be limited in the NC program. With G141, for example, it is possible to prevent the C axis (table) from rotating further around a vector solution than is permitted. It is also possible to program a limit plane.

The programmed positions must fall within the range of the SW limit switch Axes/CfgProgAxis/ParameterSets/Pn0/CfgPositionLimits/swLimitSwitchPos, Axes/CfgProgAxis/ParameterSets/Pn0/CfgPositionLimits/swLimitSwitchNeg; otherwise an error message is issued.

Associated functions

G151



Deactivation

G152 can be deactivated via:

- G151
- Program end M30
- Cancel program
- CNC reset
- Switch on controller

Changes to V5xx

- See "G151 und G152" on page 517.

Example

Limiting the range of traverse of the C axis.

```
G152 C1=30.000 C2=-30.000
```

G152 The C axis is only allowed in this range, otherwise an error message is issued.

6.10 G153 Correct Workpiece Zero Point: OFF

G153 deactivates the implementation of the workpiece zero point. The active offset in the linear axes is canceled.

Format

G153

Address description

No specific addresses.

Application

Modality

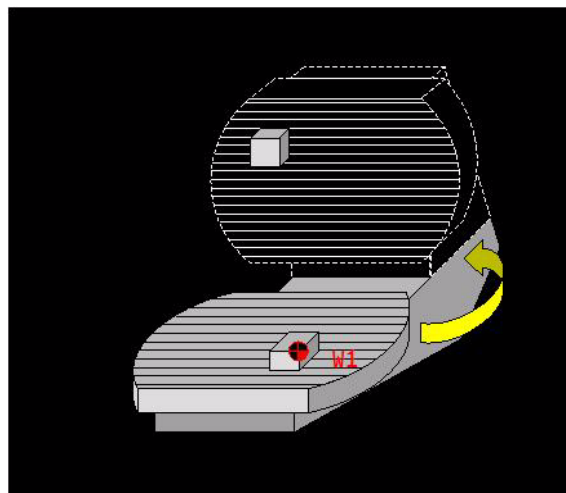
G153 is modal with G154.

Execution

G153 resets the modal status of the G154 function. The tool zero point is then no longer implemented. G153 does not perform any actions until the movement in the preceding block is finished (<INPOD>).

Display

The G153/G154 functions are in the processing status display in the modal G series.

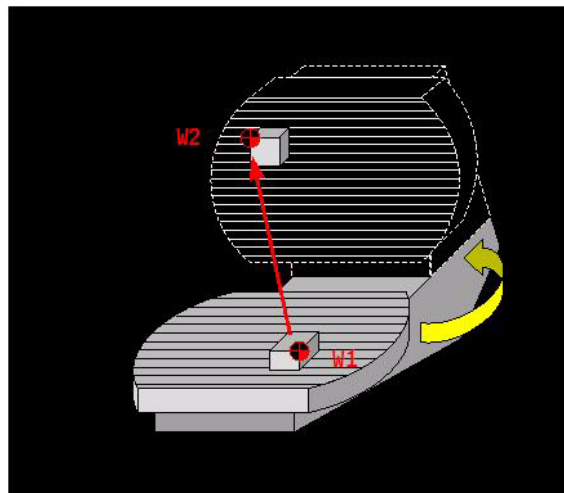


6.11 G154 Correct Workpiece Zero Point: ON

When the rotary axis rotates, the zero point of the workpiece rotates with the workpiece. The difference to G7 is that the axis directions do not rotate as well.

G154 activates the implementation of the workpiece zero point using kinematic calculations. This can only be activated for rotary axes in the table. When active, the status of the programmed rotary axis is offset at the end of a positioning in the position of the linear axes. The linear axes are not included.

Note: the offset in the linear axes due to G108 is not dependent on G154/G153 and remains active. G108 has the same function but is only effective for the tool head.



Format

G154 {A1=} {B1=} {C1=}

Address description

- ▶ **A1 ZPS A-axis (0=not, 1=settle)** Defines whether the position of the A axis in the table is offset in the linear axes:
 - A1=0 not offset (default setting)
 - A1=1 is offset. This address is only allowed if there is an A axis in the table.
- ▶ **B1 ZPS B-axis (0=not, 1=settle)**
- ▶ **C1 ZPS C-axis (0=not, 1=settle)**

Default setting

If no addresses are programmed, all axes in the table are activated.

Application

Modality

G154 is modal with G153.

Execution

When G154 is active, the linear axis display is updated at the end of every positioning movement of the rotary axes defined in G154. G154 does not perform any actions until the movement in the preceding block is finished (<INPOD>).

Switching off G154

G153 switches the G154 function off. G154 remains active after CANCEL PROGRAM, M30, CNC RESET, or switching on the control. The programmed rotary axis is saved in the memory.

Interruption

When a rotary axis movement is canceled, the linear axis display is not updated. During an interruption, the linear axis display is only updated to show the rotary axis status after <Emergency stop>, CANCEL PROGRAM, or <Manual> has been pressed.

Manual operation

The G154 function remains active after M30 and is active during manual operation. The linear axis display is updated when the rotary axis movement has stopped.

Zero point shift

A zero point shift (G54, G92, G93) or IPLC shift in the relevant rotary axis is offset. This means that the new zero point for the rotary axis is used as the zero status for the kinematic calculations.

Status display

The G153/G154 status is displayed in the modal G group display.

Example

Activating implementation of the workpiece zero point

```
G154 B1=1
```

G154 The workpiece zero point is corrected after the table rotation.

6.12 G174 Tool Retract Movement

Movement to disengage the tool axis during 5-axis milling.

Format

G174 {L...} {X1=... or Y1=... or Z1=...}

Address description

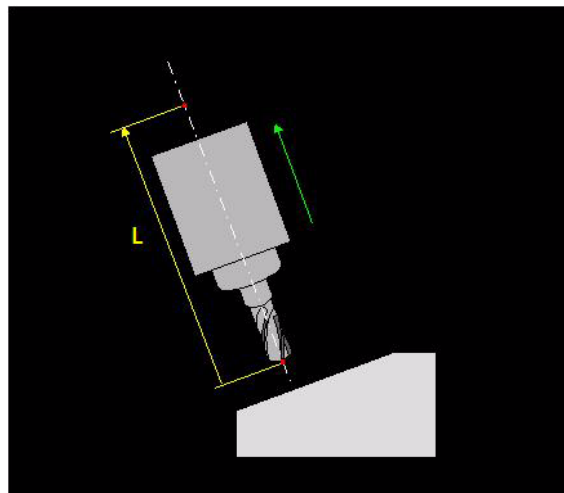
- ▶ **L retract distance** the retraction distance ($L > 0$) defines the distance that is travelled in the tool direction. An error message (Z31) is issued if L is greater than the distance to the software limit switch. If L is not specified, travel continues up to the software limit switch.
- ▶ **X1=, Y1=, Z1= 1=retraction only in this axis** X1= or Y1= or Z1= are used in the program to determine which machine axis travels. In the case of G7, the machine axis can be different to the programmed axis. A combination of X1=, Y1=, and Z1= is not possible (P414). **Retraction is not performed perpendicularly.** X1=1 means that only the X axis travels.
 - No X1=, Y1=, Z1=: With this function, the retraction can always be carried out in the direction of the milling head. The tool is retracted until the first software limit switch is reached. The tool axis orients itself perpendicular to the new plane. The retract movement is performed in this perpendicular direction.

Application

Execution (G0)

G174 is executed in G0. The tool travels at this feed rate if F6= is programmed.

After G174, G0 or G1 from the previous block is reactivated modally.



Example

Tool retraction movement

```
G174 L100
G174 L100 X1=1
```

G174 The tool retracts by 100 mm.
G174 The tool travels 100 mm in the X axis.

6.13 G179 ContourCycle Call

The G179 function executes the last defined machining cycle a single time. The starting point of the cycle is the position that was programmed last before the G179 block.

Address description

No specific addresses.

Application

Associated functions

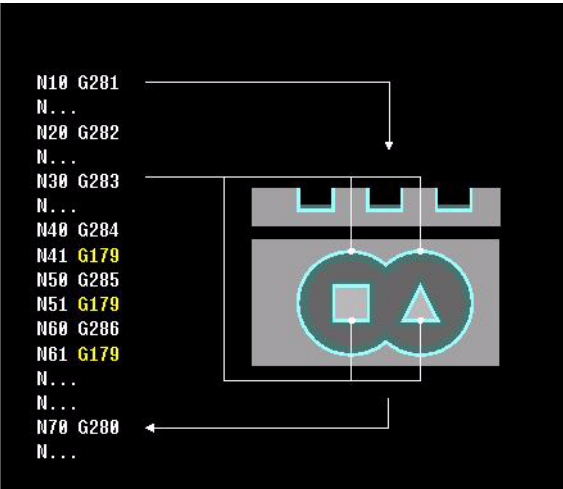
G284, G285, and G286.

Example

Drilling

```
G284 T10=1 C1=5 F2=100
G179
```

- G284 Define contour pilot drilling
- G179 Perform contour pilot drilling



6.14 G180 Cancel Cylinder Interpolation

Canceling the cylindrical coordinate system or defining the main plane and tool axis (basic coordinate system).

Format

G180 basic coordinate system

G180 [principal axis 1] [principal axis 2] [tool axis].

Address description

- ▶ X1= allocate axis to coord.system
- ▶ Y1= allocate axis to coord.system
- ▶ Z1= allocate axis to coord.system
- ▶ A1= allocate axis to coord.system
- ▶ B1= allocate axis to coord.system
- ▶ C1= allocate axis to coord.system

Application

General basic principles

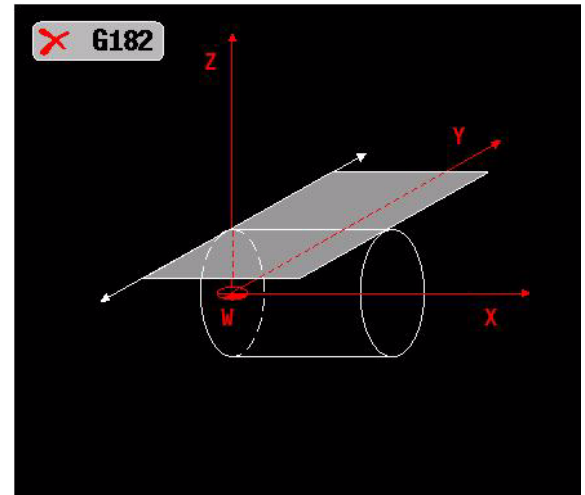
The normal setting is G180 X1 Y1 Z1

The following configurations are possible:

- Principal axis 1 X
- Principal axis 2 Y
- Tool axis Z or W

Three different pieces of information determine the correct method:

- 1 The tool axis is determined using G17/G18/G19 (G17 Z).
- 2 G180 determines which axes have to be implemented. (G17 W in Z).
- 3 The machine parameters for the tool axis definition must be correct. (Axis/CfgProgAxis/W/progKind = ParallelLinCoord tool axis W belongs to Z).



Application

- The functions G41...G44, G64, axis rotation (G92/G93 B4=), and G141 must be deleted before G180 is activated. We recommend that you delete the radius compensation with G40.
- The tool length compensation is active in the defined tool axis. The radius compensation is active in the main plane.
- The machine parameters must be set correctly. Axis/CfgProgAxis/A/index must = 3 if the W axis is the fourth axis (same as for Z axis). Axis/CfgProgAxis/A/progKind = ParallelLinCoord (W axis is a linear axis).
- Only Cartesian coordinates can be used.
- If G180 is programmed and the radius compensation is still active, then it is deleted by G180.

Example

Tool retraction movement

```
G180 X1 Y1 Z1
G81 Y2 B10 Z-22
G79 X0 Y0 Z0
```

G180	Activate main plane XY and tool axis Z.
G81	The tool travels 100 mm in the X axis.
G79	Drilling, with the feed movement taking place in the Z axis.

6.15 G182 Activate Cylinder Interpolation

Selection of the cylindrical coordinate system. This system enables simple programming of contours and positions on the curved cylinder surface.

Format

G182 [cylinder axis] [rotary axis] {tool axis} R...

Address description

- ▶ X cylinder plane:2/tool axis:3
- ▶ Y cylinder plane:2/tool axis:3
- ▶ Z cylinder plane:2/tool axis:3
- ▶ B cylinder plane:1
- ▶ C cylinder plane:1
- ▶ R cylinder radius

Application

Modality

G182 is modal with G180.

Movements within G182:

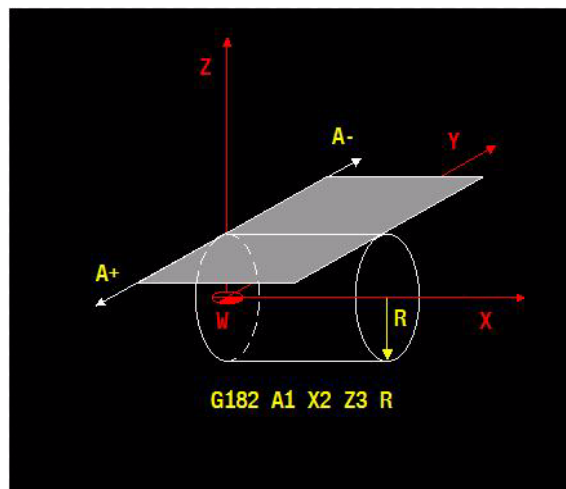
Rapid traverse for active G182: G0 [cylinder axis] [rotary axis] {tool axis}

Linear feed movement: G1 [cylinder axis] [rotary axis] {tool axis} {F...}

Circular feed movement: G2/G3 [cylinder axis] [rotary axis] R.

Return to basic coordinate system

With G180 or M30, soft key **Cancel program**, **CNC reset**.



Possible configurations

The expressions X,Y,Z,A,B,C must not be programmed without a value. The configuration for the cylinder interpolation is programmed in the G182-block:

Standard configuration

Rotary axis	A1	B1	C1
Cylinder axis	X1	Y1	Z1
Tool axis	Y1/Z1	X1/Z1	X1/Y1
Cylinder radius	R	R	R

Advanced configuration

Rotary axis marked with 1	A1	B1	C1
Cylinder axis marked with 2	X2/Y2/Z2	Y2/X2/Z2	Z2/X2/Y2
Tool axis marked with 3	Y3/Z3/X3	X3/Z3/Y3	X3/Y3/Z3
Cylinder radius	R	R	R

Machine parameters

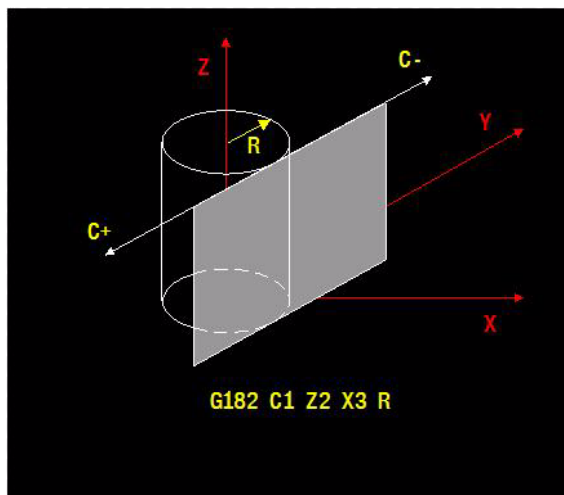
The machine parameters for the axis definition must be correct.

CfgProgAxis/index = 1, CfgProgAxis/axName = X (axis)
 CfgProgAxis/index = 2, CfgProgAxis/axName = Y (axis)
 CfgProgAxis/index = 3, CfgProgAxis/axName = Z (axis)
 CfgProgAxis/index = 4 belongs to axis 1 (4-3),
 CfgProgAxis/axName = A (axis rotating)

- The following functions must not be active if G182 is activated: G41-G44, G64, G92/G93 B4=, G141
- The following cannot be programmed when G182 is active: G25/G26, G27/G28, G51-G59 or G54 I..., G61/G62 G70/G71, G73, G92/93, switch working plane.
- The tool radius selected should only be slightly less than the recess width. (Undercuts)
- Restriction: 5 mm < cylinder radius < 500 mm
- Only Cartesian coordinates can be used.

Changes to V5xx

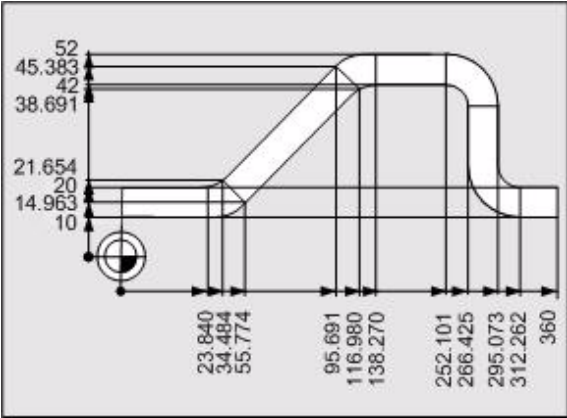
- See "G182" on page 517.



Example

The recess on the curved surface of a cylinder (diameter 40 mm) is to be milled with a double-edged endmilling tool (diameter 9.5 mm) The machining depth is 4 mm. The horizontal machining of the workpiece is performed in the rotary axis C, the cylinder axis Z, and the tool axis Y.

```
N12340
G18 S1000 T1 M66
G54
G182 Y1 C1 Z1 R20
G0 Y22 C0 Z15 M3
G1 Y16 F200
G43 Z10
G41
G1 C23.84
G3 Z14.963 C55.774 R15
G1 Z38.691 C116.98
G2 Z42 C138.27 R10
G1 C252.101
G2 Z37 C266.425 R5
G1 Z26
G3 Z10 C312.262 R16
G1 C365
G40
G41 Z20
G1 C312.262
G2 Z26 C295.073 R6
G1 Z37
G3 Z52 C252.101 R15
G1 C138.27
G3 Z45.383 C95.691 R20
G1 Z21.654 C34.484
G2 Z20 C23.84 R5
G1 C0
G40
G180
G0 Y100
M30
```



6.16 G195 Graphic Window Definition

Definition of the dimensions of a 3D-graphic window and its position relative to the zero point W.

Format

G195 X... Y... Z... I... J... K... {N1=...} {N2=...}

Address description

- ▶ X, Y, Z start point coordinates
- ▶ I dimension parallel to X
- ▶ J dimension parallel to Y
- ▶ K dimension parallel to Z
- ▶ N1= repeater begin block
- ▶ N2= repeater end block

Application

Default setting

If no addresses are programmed, all axes in the table are activated. If no dimensions are defined for the 3D windows, the distances of the software limit switch are used

Application

In programs with several level definitions, only the operations in the last programmed machining levels are displayed graphically.

Addresses N1= "Graphic begin block" and N2= "Graphic end block" are used to record the graphic window for a particular program part. All movements in the blocks from address N1= up to and excluding the block number in Address N2= are displayed in the graphic window.

Changes to V5xx

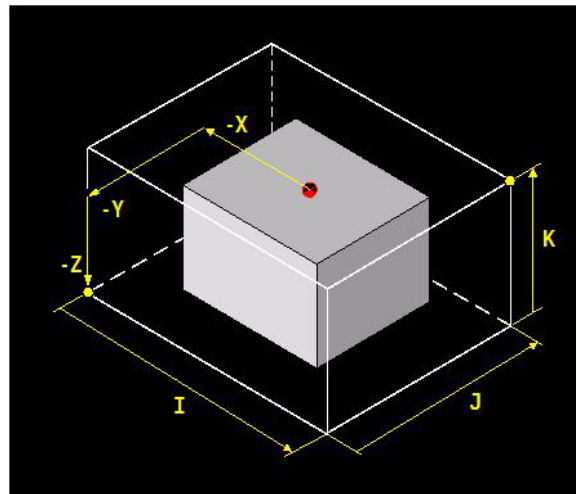
- See "G98_B" on page 512.

Example

```
G195 X-30 Y-30 Z-70 I170 J150 K100
```

```
G199 .....
```

G195 Graphic window definition
G199 Start graphic contour description



6.17 G196 End Graphic Model Description

Conclusion of the graphic contour description.

Format

G196

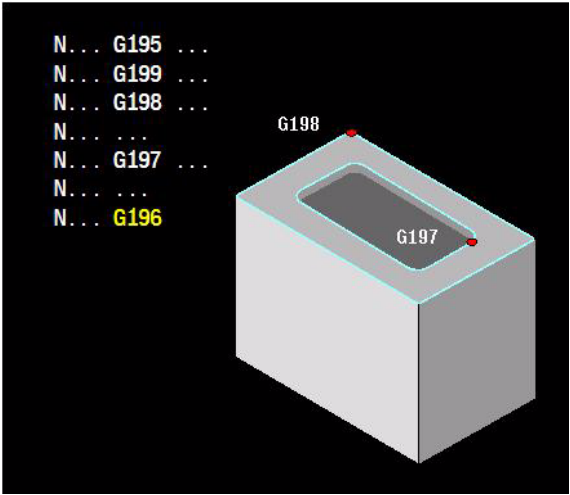
Address description

No specific addresses.

Example

```
G195 X... Y... Z... I... J... K...
G199 X... Y... Z...B... C...
G198 X... Y... Z... D...
-----
G197 X... Y... D...
-----
G196
```

G195	Graphic window definition
G199	Start graphic contour description
G198	Begin outside contour description
G197	Begin inside contour description
G196	End graphic model description



7

G200-G299 G-Codes

7.1 G240 Contour Pre-Calculation: OFF

G240 is used to deactivate contour pre-calculation G242.

Address description

No specific addresses.

Application

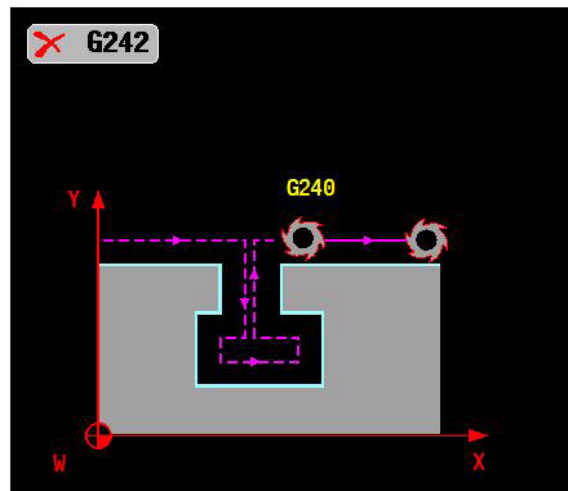
G242 is deactivated by G240, M30, or **Cancel program**.

G240 automatically takes effect after:

- Control activation
- Cancel program
- M30.

Modality

G240 is modal with G242.



7.2 G242 Contour Pre-Calculation: On

Checks the radius-compensated contour for undercuts and overcuts in advance.

Address description

► **I2= Look ahead check:** 0=off, >0=number

■ **I2=0** No check.

■ **I2=...** If nn > 0, the check is active. The maximum value is 99.

Default setting

I2=5

Application

G242 is deactivated by G240, G40, M30, or **Cancel program**.

Modality

G242 is modal with G240.

Block access

The G242 function checks are normally executed during a block entry.

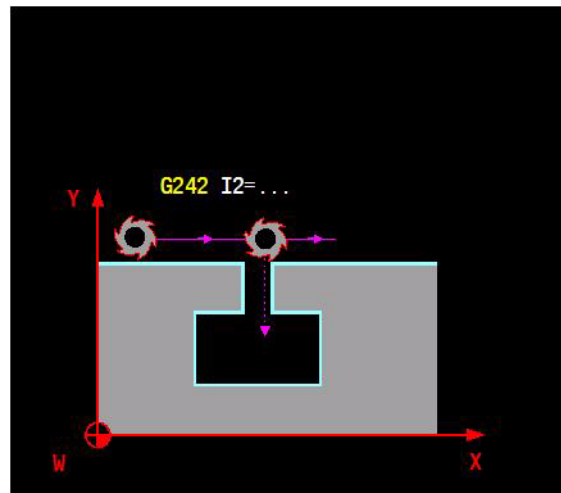
Associated functions

G242 works with G41 and/or G42.

Procedure

The tool path is calculated in advance from the current block. Areas of the contour that might be damaged by the tool are not machined. The number of pre-calculated blocks (maximum 99) is set at I2=. The larger the number of blocks to be pre-calculated, the longer the block processing time will be.

Function G242 works only if radius compensation is active.



7.3 G251-G269 Contour Programming

Workpiece drawings that are not dimensioned in accordance with NC often contain coordinate data. For example:

- known coordinates can lie on the contour element or in its proximity
- Coordinate data can be referenced to another contour element
- Directional data and data regarding the course of the contour

You can enter such dimensional data directly by using the contour programming functions G251 to G269. MillPlus calculates the contour from the known coordinate data and supports the programming dialog with the interactive programming graphic.



The contour programming feature can only be used for programming contour elements that lie in the working plane. The working plane is defined in the first block of the part program.



If both conventional blocks and contour blocks are entered in a program, each contour section must be fully defined before you can return to conventional programming.

MillPlus needs a fixed point from which it can calculate the contour elements. Directly before the contour section, program a position that contains both coordinates of the working plane.



Contour programming can be supported by the ICP contour programming dialog.



G90 absolute programming must be active at the start of contour programming.

Points cannot be programmed or used (G78).

Programming of MSTH functions is not permitted.

If functions G64, G182, G141, or G91 are active, functions G251 - G269 are not permitted.

Overview of Contour Programming

G functions

G9 Define pole position
G251 Free linear movement
G252 Free circular movement, CW
G253 Free circular movement, CCW
G261 Free linear movement, tangential
G262 Free circular movement, CW, tangential
G263 Free circular movement, CCW, tangential
G265 Free chamfer
G266 Free rounding
G269 Free contour selection

Auxiliary points

For both free-programmed straight lines and free-programmed circular arcs, you can enter the coordinates of auxiliary points that are located on the contour or in its proximity.

Auxiliary points on a contour

The auxiliary points are located on a straight line, the extension of a straight line, or on a circular arc.

Example

```

N1234 FIGURE
G98 X-10 Y-10 Z-2 I50 J50 K20
N1 G17
N2 G0 X0 Y0 Z0 F5000
N3 G251 X1=20 Y1=20 B1=45
N4 G251 B1=0 Y15
N5 G251 Y0 B1=-45 X50
N6 M30

```

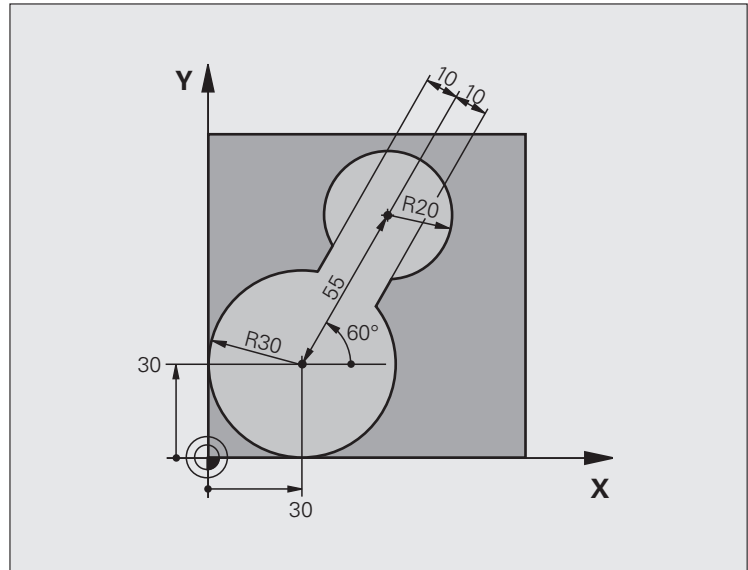
Auxiliary points near a contour

The auxiliary points are located near the straight line, near the extension of a straight line, or near the circular arc.

Example

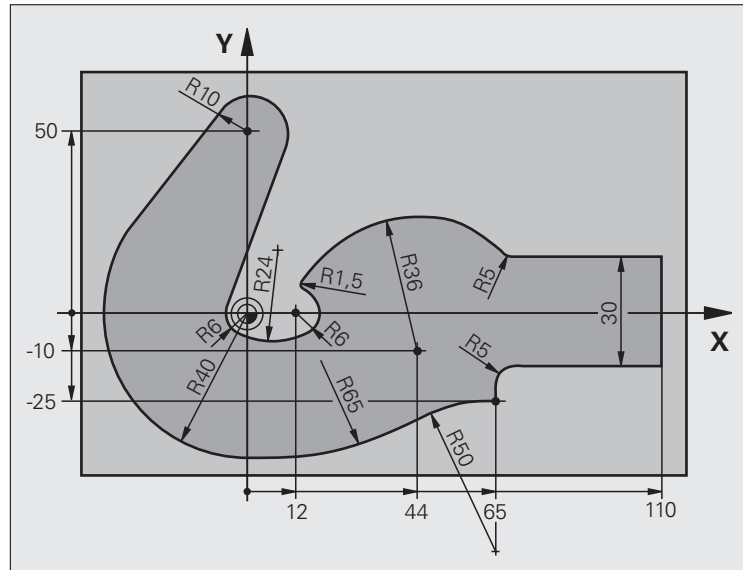
See example below "Contour programming with auxiliary points near the contour"

Example: Contour programming with auxiliary points near the contour



N111 G98 X0 Y0 Z-15 I100 J100 K50	Definition of workpiece blank
N1 G17 F5000	
N6 G0 X30 Y30	
N8 G0 Z-5	Move to working depth
N9 G1 X0 Y30	Approach the contour on a circular arc with tangential connection
N10 G9 X30 Y30	Define pole position
N11 G252 R30 I30 J30	Approach the contour on a circular arc with tangential connection
N12 G251 B1=60 X4=30 Y4=30 L4=10	Straight line with auxiliary point near the contour
N13 G269 I1=3	Selecting a solution for the course of the contour
N14 G252 R20 B3=60 L3=55	Connecting circle
N15 G269 I1=2	Selecting a solution for the course of the contour
N16 G251 B1=-120 X4=30 Y4=30 L4=10	Straight line with auxiliary point near the contour
N17 G269 I1=3	Selecting a solution for the course of the contour
N18 G252 X0 R30 I30 J30	Connecting circle
N19 G269 I1=2	Selecting a solution for the course of the contour
N20 G0 X30 Y30	
N21 G0 Z20	Retract the tool
N22 M30	

Example: Contour programming of hooks



N111 G98 X-60 Y-60 Z-15 I150 J150 K50	Definition of workpiece blank
N1 G17 F5000	
N2 G0 X-70 Y0	
N3 G0 Z-5	
N4 T1 M67	
N5 G41	
N6	Approach the contour at a tangent
N8 G1 X-40 Y0	
N9 G252 R40 I0 J0	Circular CW
N10 G261	Connecting straight line
N11 G262 R10 I0 J50	Circular CW with tangential connection
N12 G261	Connecting straight line
N13 G263 R6 I0 J0	Circular CCW with tangential connection
N14 G263 R24	Circular CCW with tangential connection
N15 G263 R6 I12 J0	Circular CCW with tangential connection
N16 G269 I1=2	Selecting a solution for the course of the contour
N17 G262 R1.5	Circular CW with tangential connection
N18 G262 R36 I44 J-10	Circular CW with tangential connection
N19 G269 I1=2	Selecting a solution for the course of the contour

N20 G263 R5	Circular CCW with tangential connection
N21 G261 X110 Y15 B1=0	Straight line
N22 G251 B1=-90	Straight line
N23 G251 X65 B1=180 L41=30 L43=20	Straight line
N24 G266 R5	Rounding arc
N25 G251 X65 Y-25 B1=-90	Straight line
N26 G253 R50 I65 J-75	Circular CW
N27 G262 R65	Circular CW with tangential connection
N28 G269 I1=1	Select the first solution from those proposed
N29 G262 R40 Y0 I0 J0	Circular CW with tangential connection
N30 G269 I1=4	Selecting a solution for the course of the contour
N31 G0 X-70	Retract the tool
N32	Leave the contour at a tangent
N33 G40	Canceling tool compensation
N34 G0 Z20	Retract the tool
N35 M30	Program end

7.4 G251 Free Linear Movement

Function G251 defines a straight line without tangential connection. MillPlus moves the tool in a straight line from its current position to the straight line end point. The starting point is the end point of the preceding block.



The contour programming feature can only be used for programming contour elements that lie in the 2D working plane.

Address description

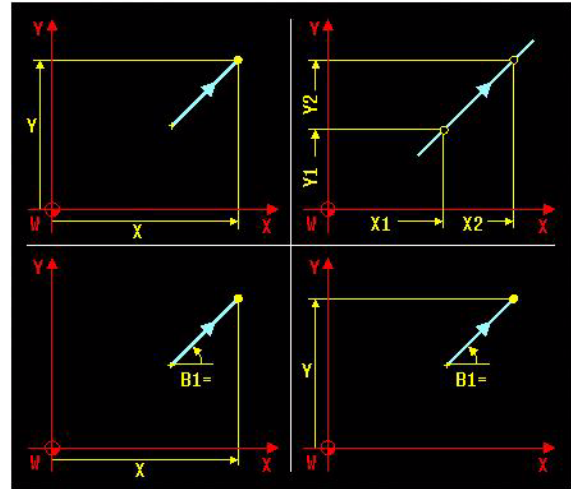
- ▶ **X, Y, Z end point coordinates** Addresses X, Y, Z define an end point.
- ▶ **X91=, Y91=, Z91= end point, incremental**
- ▶ **X1=, Y1=, Z1= 1. auxiliary point contour element**
- ▶ **X2=, Y2=, Z2= 2. auxiliary point contour element**
- ▶ **X4=, Y4=, Z4= auxiliary point beside contour element**
- ▶ **L4= parallel shift** Always together with X4=, Y4=, Z4=
- ▶ **L41= incremental parallel shift**
- ▶ **I5= start(1)/end(-1) closed contour** 1: start of contour -1: end of contour.
- ▶ **B1= angle**
- ▶ **F feed**
- ▶ **B11= incremental angle**
- ▶ **B2= polar angle**
- ▶ **B21= incremental polar angle**
- ▶ **L1= path length**
- ▶ **L2= polar length**
- ▶ **L21= incremental polar length**

Example

Defining a straight line

G251 X65 Y-25 B11=-90

G251 Define a straight line



7.5 G252 Free Circular Movement, CW

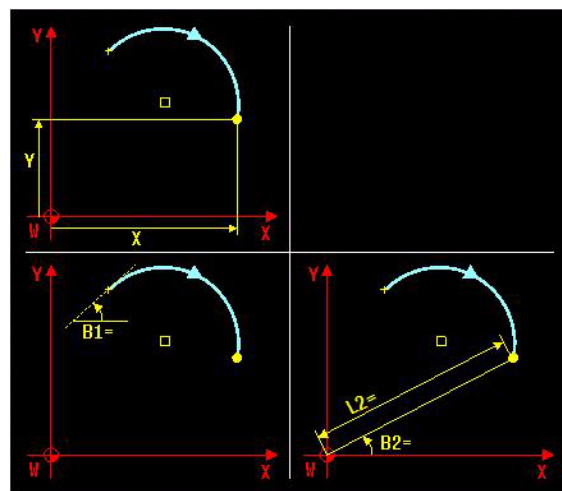
Function G252 defines a circular arc in a clockwise direction (CW). The tool moves on a circular path with the radius R.



The contour programming feature can only be used for programming contour elements that lie in the 2D working plane.

Address description

- ▶ **X, Y, Z end point coordinates** Addresses X, Y, Z define an end point.
- ▶ **X91=, Y91=, Z91= end point, incremental**
- ▶ **R circle radius**
- ▶ **I, J, K circle center points (X,Y,Z..)**
- ▶ **I91=, J91=, K91= incremental center points**
- ▶ **X1=, Y1=, Z1= 1. auxiliary point contour element**
- ▶ **X2=, Y2=, Z2= 2. auxiliary point contour element**
- ▶ **X3=, Y3=, Z3= 3. auxiliary point contour element (circle)**
- ▶ **X4=, Y4=, Z4= auxiliary point beside contour element**
- ▶ **L4= parallel shift** Always together with X4=, Y4=, Z4=
- ▶ **I5= start(1)/end(-1) closed contour** 1: start of contour -1: end of contour.
- ▶ **B5= angle of arc**
- ▶ **F feed**
- ▶ **L1 = chord length of the arc**
- ▶ **B1= gradient angle of the entry tangent**
- ▶ **B11= incremental angle**
- ▶ **B2= polar angle**
- ▶ **B21= incremental polar angle**
- ▶ **L2= polar length**
- ▶ **L21= incremental polar length**
- ▶ **B3= polar angle for center**
- ▶ **B31= incremental polar angle center**
- ▶ **L3= polar length for center**
- ▶ **L31= incremental polar length center**



Application



The starting and end points of the arc must lie on the circle.

Input tolerance: up to 0.016 mm (selected via the "circleDeviation" machine parameter)

Example

Defining a circle

```
G252 R40 I0 J0
```

R4 Radius is executed clockwise.

7.6 G253 Free Circular Movement, CCW

Function G253 defines a circular arc in a counter-clockwise direction (CCW). The tool moves on a circular path with the radius R.

Address description

Identical to addresses of G252, (see "Address description" on page 308)

Application



The starting and end points of the arc must lie on the circle.

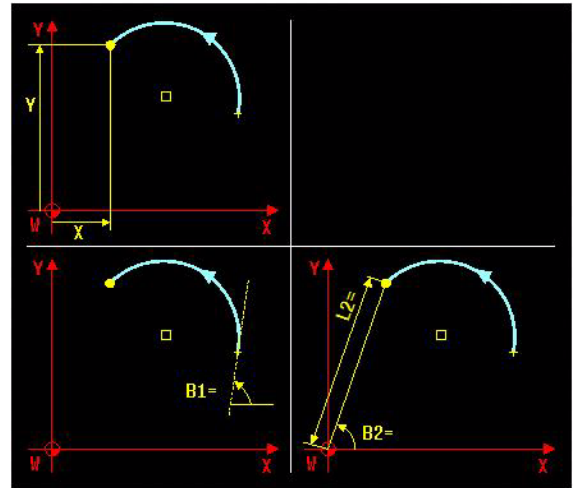
Input tolerance: up to 0.016 mm (selected via the "circleDeviation" machine parameter)

Example

Defining a circle

```
G252 R40 I0 J0
```

R4 Radius is executed counter-clockwise.



7.7 G261 Free Linear Movement, Tangential

Function G261 defines a straight line with a tangential connection. MillPlus moves the tool in a straight line from its current position to the straight line end point. The starting point is the end point of the preceding block.

Address description

Identical to addresses of G251, (see "Address description" on page 307)

Application

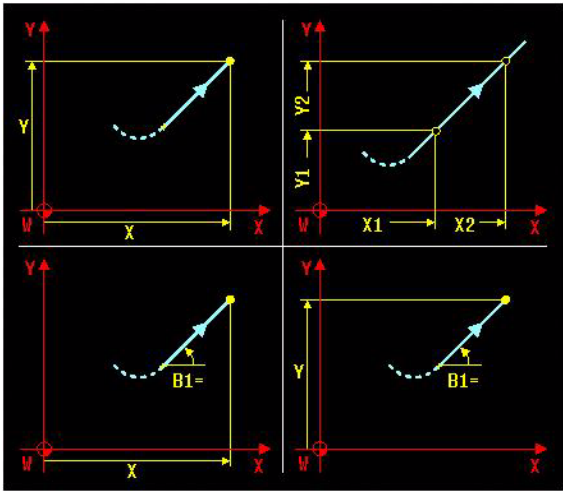
A transition between two contour elements is called "tangential" when there is no kink or corner at the intersection between the two contours, i.e. the transition is smooth. The contour element to which the tangential arc connects must be programmed directly before the G261 block. This requires at least two positioning blocks.

Example

Defining a straight line

G261 X65 Y-25 B1=-90

G261 Define a straight line



7.8 G262 Free Circular Movement, CW, Tangential

Function G262 defines a circular arc in a clockwise direction with a tangential connection. The tool moves on an arc that starts tangentially to the previously programmed contour element.

Address description

Identical to addresses of G252, (see "Address description" on page 308)

Application

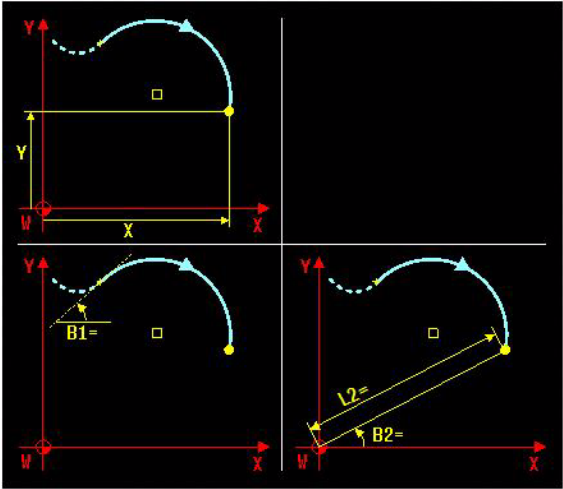
A transition between two contour elements is called "tangential" when there is no kink or corner at the intersection between the two contours, i.e. the transition is smooth. The contour element to which the tangential arc connects must be programmed directly before the G262 block. This requires at least two positioning blocks.

Example

Defining a circle

```
G262 R40 Y0 I0 J0
```

G262 Radius is executed clockwise.



7.9 G263 Free Circular Movement, CCW, Tangential

Function G263 defines a circular arc in a counter-clockwise direction with a tangential connection. The tool moves on an arc that starts tangentially to the previously programmed contour element.

Address description

Identical to addresses of G252, (see "Address description" on page 308)

Application

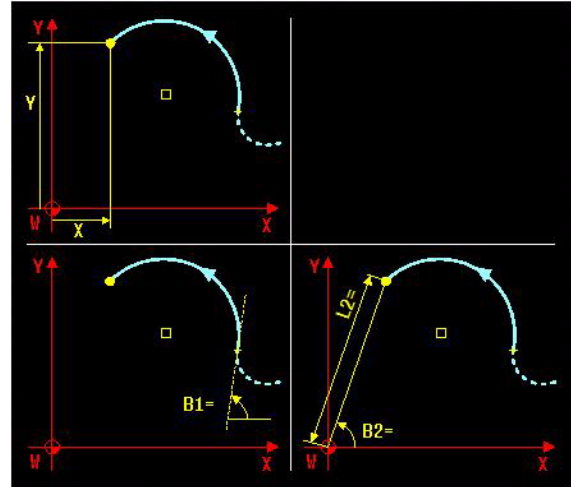
A transition between two contour elements is called "tangential" when there is no kink or corner at the intersection between the two contours, i.e. the transition is smooth. The contour element to which the tangential arc connects must be programmed directly before the G263 block. This requires at least two positioning blocks.

Example

Defining a circle

```
G263 R40 Y0 I0 J0
```

G263 Radius is executed counter-clockwise.



7.10 G265 Free Chamfer

The chamfer enables you to cut off corners at the intersection of two straight lines. Function G265 defines the chamfer.

Address description

► L chamfer length

Application

The chamfer must be machinable with the current tool.



You cannot start a contour with a chamfer.

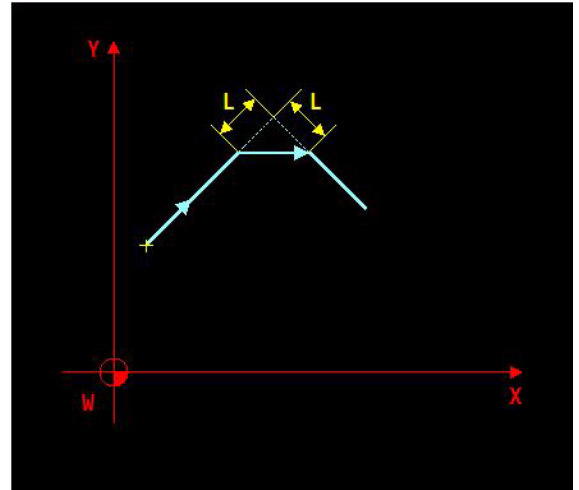
The corner point is cut off by the chamfer and is not part of the contour.

Example

Defining a chamfer

G265 L4

L4 Length of the chamfer.



7.11 G266 Free Rounding

The G269 function is used for rounding off corners. The tool moves on an arc that is tangentially connected to both the preceding and subsequent contour elements.

Address description

► R rounding radius

Application

The rounding arc must be machinable with the called tool.



The corner point is cut off by the rounding arc and is not part of the contour.

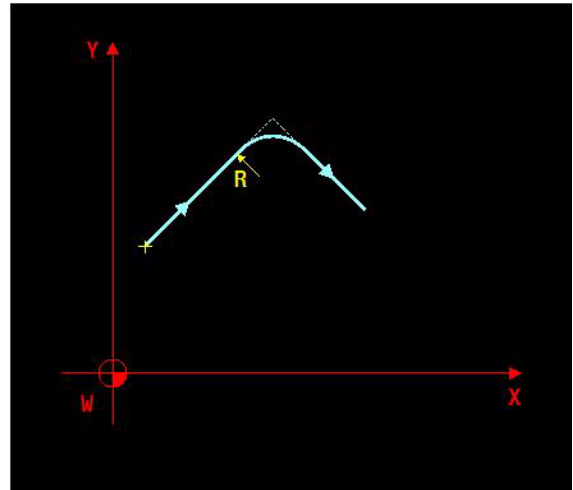
In the preceding and subsequent contour elements, both coordinates must lie in the plane of the rounding arc. If you machine the contour without tool-radius compensation, you must program both coordinates in the working plane.

Example

Defining a rounding

G266 R4

R4 Select the fourth solution from those proposed.



7.12 G269 Free Contour Selection

You use function G269 to select a solution.

Address description

- **I1= selection of solution** Number of the solution from those proposed and calculated by MillPlus.

Application

Parameter I1= is selected with the graphic function while the program is being formatted. G269 is always in a block after the element definition.

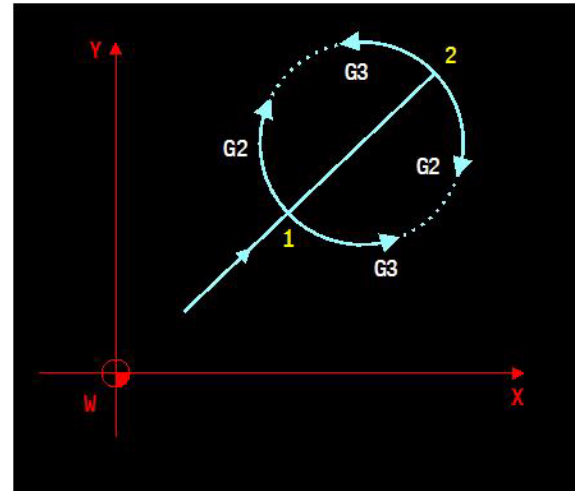
Example

Selecting a contour

```
...  
G262 R40 Y0 I0 J0  
G269 I1=2  
...
```

I1=2 Select the second solution from those proposed in the block after the definition element

For application within a program, (see "Example: Contour programming with auxiliary points near the contour" on page 304)



7.13 G270 Disables Limit Planes

Deactivates all defined limit planes.

Address description

► I1= disable and/or delete limit planes

- **I1=0** temporarily disables the defined limit planes. G271 can be used to re-activate the same limit planes.
- **I1=1** deletes the definitions of the limit planes and disables the planes. This function is executed with M30.

Default setting

I1=0

Application

Modality

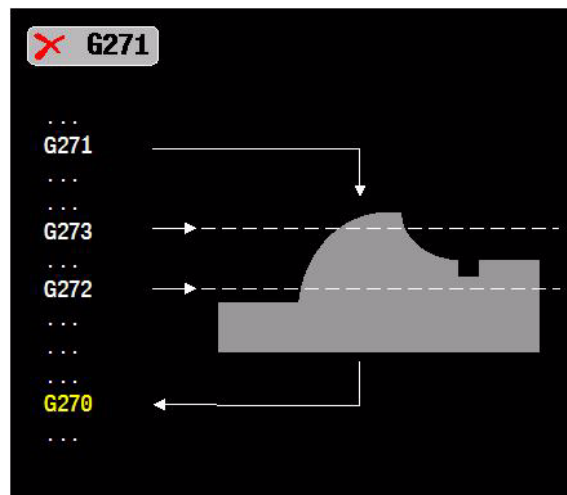
G270 is modal with G271.

Associated functions

G271, G272, G273.

Procedure

The limit planes defined by G272 or G273 are disabled. The limit plane definition remains active and can be re-enabled with G271.



7.14 G271 Enables Defined Limit Planes

Activates the defined limit plane.

Address description

► I1= limit plane

- I1=1 lower limit plane is activated (G272)
- I1=2 upper limit plane is activated (G273).

Default settings

I1=1

Application

Modality

G271 is modal with G270.

Associated functions

G270, G272, G273.

Limit plane definition

The limit plane to be enabled must be defined beforehand in the NC program (G272 or G273). Otherwise, an error message is issued.

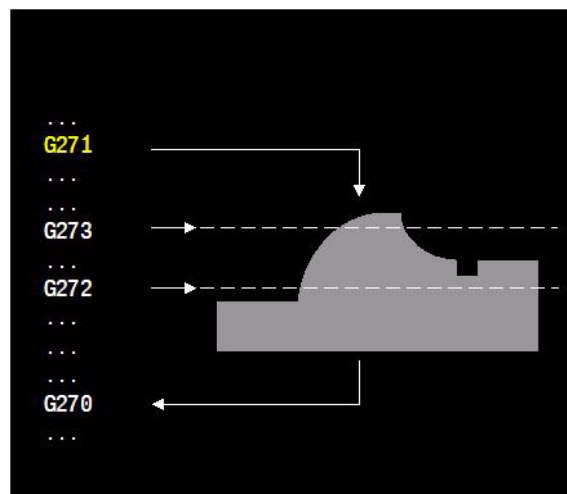
Procedure

The limit planes defined by G272 and/or G273 are enabled.

NC program execution is restricted by means of up to two limit planes. Only movements between the G272 lower limit plane and the G273 upper limit plane are executed according to the NC program. The movements outside of the limit planes are skipped or executed projected onto the limit plane.

Only one of the limit planes needs to be defined.

G270 can be used to disable the limit planes again.

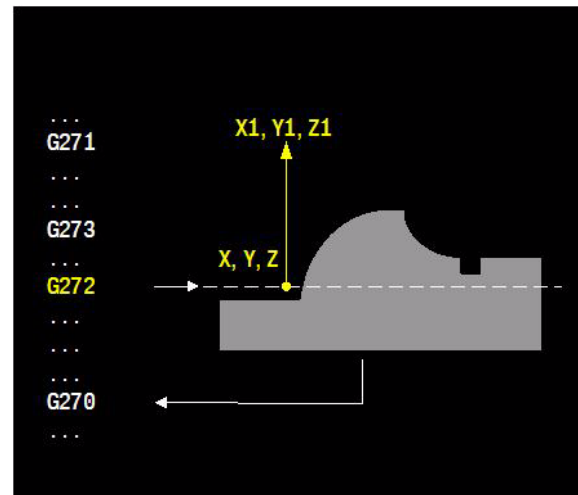


7.15 G272 Definition of Lower Limit Plane

Defines the lower limit plane during machining.

Address description

- ▶ **X, Y, Z limit plane point** Addresses X, Y, Z define a point. The limit plane passes through this point. The point is defined in relation to the workpiece zero point W.
- ▶ **X1=, Y1=, Z1= limit plane normal vector** Defines the normal direction of the limit plane. In conjunction with the point (X,Y,Z), this defines the limit plane. The normalized vector points to the top of the plane. Basic setting (0, 0, 1).
- ▶ **I1= behavior on other side of limit plane**
 - **I1=1** machine normally. The plane is thus inactive.
 - **I1=2** machine along the projected path. (Default setting).
 - **I1=3** explicitly defined direction (X3=, Y3=, Z3=)
- ▶ **I2= kind of limit plane projection (I1=2)** To be defined when I1 = 2. The movements below the plane are projected onto the limit plane. The direction of this projection can be programmed:
 - **I2=1** normalized vector of the plane.
 - **I2=2** tool direction (default setting).
 - **I2=3** explicitly defined direction (X2=, Y2=, Z2=).
- ▶ **X2=, Y2=, Z2= limit plane projection vector (I2=3)** To be defined when I2 = 3. Defines the projection direction of the non-executed movements below the limit plane at the limit plane.
- ▶ **I3= kind of limit plane aux. movements (I1=3)** To be defined when I1 = 3. The movements below the plane are skipped by auxiliary movements. The direction of the auxiliary movements can be programmed:
 - **I3=1** normalized vector of the plane.
 - **I3=2** tool direction (default).
 - **I3=3** explicitly defined direction (X3=, Y3=, Z3=)
- ▶ **X3=, Y3=, Z3= aux. movements vector (I3=3)** To be defined when I3=3. Defines the direction of the auxiliary movements for exit and approach.
- ▶ **L1= exit and approach distance** This distance is traversed in feed mode.
- ▶ **L2= safety distance (I1=3)** To be defined when I1=3. Defines the clearance height at which the movements are traversed below the plane. The tool moves to this position (height) in rapid traverse.
- ▶ **F6= approach feed** Defines the feed rate at which the distance L1= is traversed on approach. The default is normal feed rate.



Default setting

I1=2, I2=2, I3=2, L1=0, L2=0, F6=F

Application

Associated functions

G270, G271, G273.

Deleting

The limit plane definition is deleted at the end of the main program.

Procedure

NC program execution is restricted by means of two limit planes. Only movements between the G272 lower limit plane and the G273 upper limit plane are executed according to the NC program. The movements programmed outside of the two limit planes are skipped or executed projected onto the limit plane.

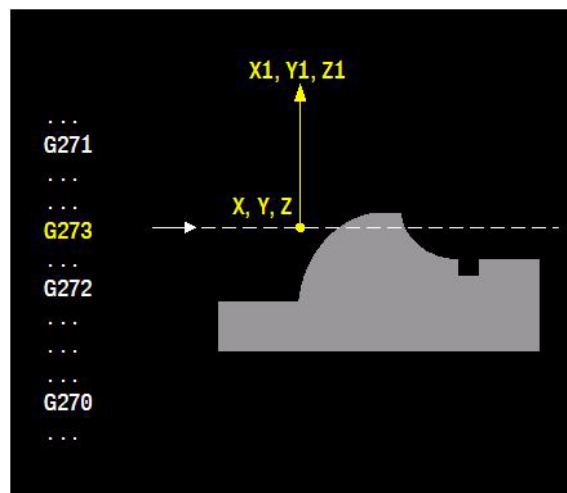
Only one of the limit planes needs to be defined.

7.16 G273 Definition of Upper Limit Plane

Defines the upper limit plane during machining.

Address description

- ▶ **X, Y, Z limit plane points** Addresses X, Y, Z define a point. The limit plane passes through this point. The point is defined in relation to the workpiece zero point W.
- ▶ **X1=, Y1=, Z1= limit plane normal vector** Defines the normal direction of the limit plane. In conjunction with the point (X,Y,Z), this defines the limit plane. The normalized vector points to the top of the plane. Basic setting (0, 0, 1).
- ▶ **I1= behavior on other side of limit plane**
 - **I1=1** machine normally. The plane is thus inactive.
 - **I1=2** machine along the projected path. (Default setting).
 - **I1=3** auxiliary movement to clearance plane.
- ▶ **I2= kind of limit plane projection (I1=2)** To be defined when I1 = 2. The movements below the plane are projected onto the limit plane. The direction of this projection can be programmed:
 - **I2=1** normalized vector of the plane.
 - **I2=2** tool direction (default setting).
 - **I2=3** explicitly defined direction (X2=, Y2=, Z2=).
- ▶ **X2=, Y2=, Z2= limit plane projection vector (I2=3)** To be defined when I2 = 3. Defines the projection direction of the non-executed movements below the limit plane at the limit plane.
- ▶ **I3= kind of limit plane aux. movements (I1=3)** To be defined when I1 = 3. The movements below the plane are skipped by auxiliary movements. The direction of the auxiliary movements can be programmed:
 - **I3=1** normalized vector of the plane.
 - **I3=2** tool direction (default).
 - **I3=3** explicitly defined direction (X3=, Y3=, Z3=)
- ▶ **X3=, Y3=, Z3= aux. movements vector (I3=3)** To be defined when I3=3. Defines the direction of the auxiliary movements for exit and approach.
- ▶ **L1= exit and approach distance** This distance is traversed in feed mode. (Default setting=0)
- ▶ **L2= safety distance (I1=3)** To be defined when I1=3. Defines the clearance height at which the movements are traversed below the plane. The tool moves to this position (height) in rapid traverse. (Default setting=0)
- ▶ **F6= approach feed** Defines the feed rate at which the distance L1= is traversed on approach. The default is normal feed rate.



Default setting

I1=2, I2=2, I3=2, L1=0, L2=0, F6=F

Application**Associated functions**

G270, G271, G272.

Deleting

The limit plane definition is deleted at the end of the main program.

Procedure

NC program execution is restricted by means of two limit planes. Only movements between the G272 lower limit plane and the G273 upper limit plane are executed according to the NC program. The movements programmed outside of the two limit planes are skipped or executed projected onto the limit plane.

Only one of the limit planes needs to be defined.

7.17 G275 Zoning Planes: Disable

Disables the defined zoning plane.

Address description

- ▶ **I1= zoning planes: disables and/or undefines**
 - **I1=0** temporarily disables the defined limit planes. G276 can be used to re-activate the same limit planes.
 - **I1=1** deletes the definitions of the limit planes and disables the planes. This function is executed with M30. Zoning planes: deletes the definitions and disables the planes

Default setting

I1=0

Application

Modality

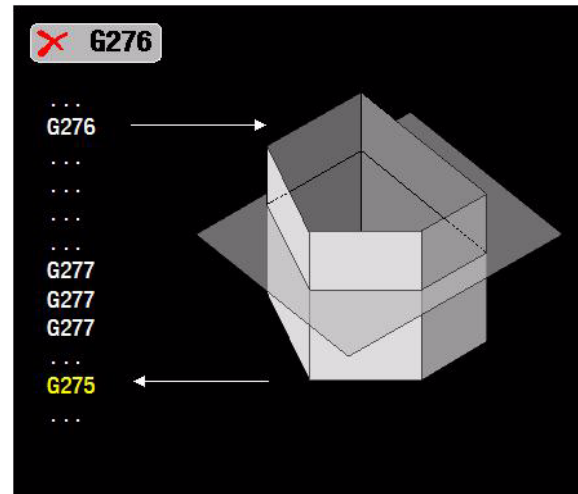
G275 is modal with G276.

Associated functions

G276, G277.

Procedure

The zoning plane defined with G277 is disabled. The zoning plane definition remains active and can be re-enabled with G276.



7.18 G276 Zoning Planes: Enable

Enables the defined zoning plane.

Address description

No specific addresses.

Application

Modality

G276 is modal with G275.

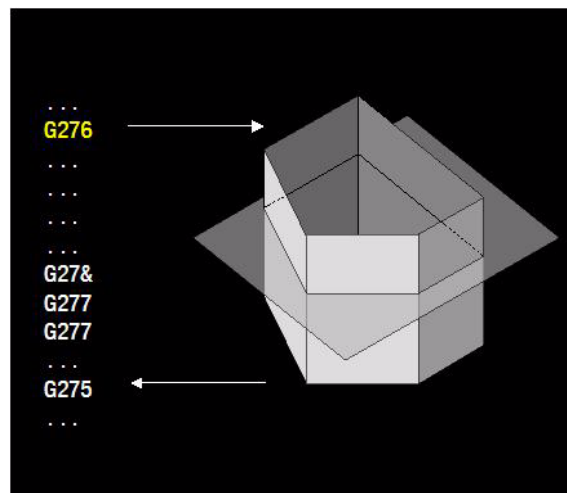
Associated functions

G275, G277.

Procedure

In addition to the limit planes (G271), there are lateral limitations known as zoning planes. The movements programmed outside of a zoning plane are skipped at clearance height or executed projected onto the zoning plane.

Only one zoning plane can be defined.



7.19 G277 Zoning Planes: Define

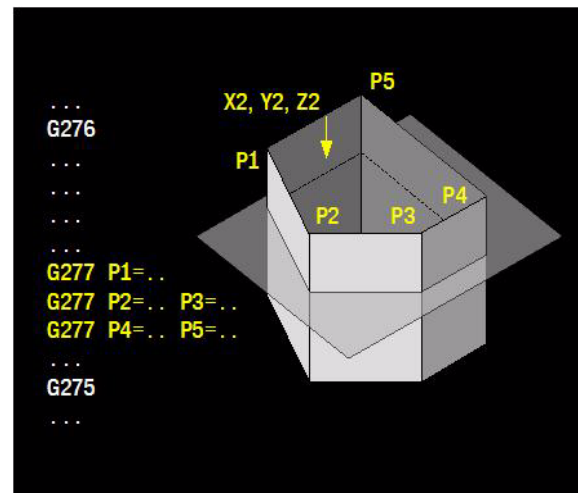
Defines the zoning planes (fences) during machining.

A zoning plane is defined by a sequence of points (polygon), a projection direction, and a machining type.

The points polygon is defined either by up to four points from the points table (Pn=) or by an unlimited number of points programmed consecutively with G277 I1=.

Address description

- ▶ **I zoning plane number** Defines the zoning plane. Multiple zoning planes can be defined. All zoning planes have the same projection direction.
- ▶ **P1=, P2=, P3=, P4= zoning plane polygon point numbers** Up to four point numbers can be programmed for defining the points polygon. Index n of Pn= defines the sequence. The polygon can be closed with I1=4. The points are defined in relation to the workpiece zero point W. This definition cannot be programmed simultaneously to (X, Y, Z)
- ▶ **I1= polygon point sequence number** If the polygon points are defined by (X, Y, Z), multiple G277 must be programmed consecutively. The sequence of G277 commands defines the sequence of the points polygon. I1= defines whether this is the first, last, or an intermediate point:
 - 1 = first point
 - 2 = intermediate point (multiple)
 - 3 = last point of an open polygon
 - 4 = last point that joins the polygon with the first point
 - If the zoning plane remains open (I1=3), the first and last sides are "infinitely extended". Additional information such as projection direction etc. is to be defined for the first point. There is no limit to the number of intermediate points. This definition cannot be used at the same time as Pn=.
- ▶ **X, Y, Z, or P zoning plane polygon point** The addresses X, Y, Z, or P define a point of the polygon. The point is defined in relation to the workpiece zero point W. This definition cannot be programmed simultaneously to Pn=.
- ▶ **I2= projection vector**
 - **I2=1** machining plane direction (G17, G18, G19, G7) (Default setting)
 - **I2=2** tool direction
 - **I2=3** explicitly defined vector (X2=, Y2=, Z2=)
In conjunction with the polygon points, this defines the zoning plane.
 - In conjunction with the polygon points, this defines the zoning plane.



- ▶ **X2=, Y2=, Z2= zoning plane projection vector (I2=3)** To be defined when I2=3. Defines the projection direction of the zoning plane.
- ▶ **I3= aux. movements vector**
 - **I3=1** projection direction (default)
 - **I3=2** horizontal of the intersection between the zoning plane and limit plane
 - **I3=3** explicitly defined vector (X3=, Y3=, Z3=).
- ▶ **X3=, Y3=, Z3= aux. movements vector (I3=3)** To be defined when I3=3. Defines the direction of the auxiliary movements for exit and approach.
- ▶ **L1= 1 exit and approach distance** This distance is traversed in feed mode. Default setting = 0.
- ▶ **L2= safety distance** To be defined when I3=3. Defines the clearance height at which the auxiliary movements are traversed. The tool moves to this position (height) in rapid traverse. Default setting = 0.
- ▶ **F6= approach feed** Defines the feed rate at which the distance L1= is traversed on approach. The default is normal feed rate.

Default setting

I=1, I1=2, I2=2, I3=2, L1=0, L2=0, F6=F

Application

Associated functions

G275, G276.

Permitted range

The zoning plane can be closed or open. The permitted range is defined by: left of the plane in the running direction of the polygon points viewed from "above" (opposite to the projection direction)

Deleting

The zoning plane definition is deleted at the end of the main program.

Procedure

NC program execution is laterally restricted by means of a zoning plane. Only movements "left" of the zoning plane are executed according to the NC program. The movements programmed "right" of the zoning plane are skipped at clearance height or executed projected onto the zoning plane. Multiple zoning planes can be defined.

7.20 G280-G286 Contour Milling Cycles

Contour milling cycles and the contour formula enable you to form complex contours by combining subcontours (pockets or islands). You define the individual subcontours (geometry data) as separate programs. In this way, any subcontour can be used any number of times. MillPlus calculates the complete contour from the selected subcontours, which you link together through a contour formula.



The memory capacity for programming a contour milling cycle (all contour description programs) is limited to **128 contours**. The number of possible contour elements depends on the type of contour (inside or outside contour) and the number of contour descriptions. You can program up to **16384** contour elements.

The contour milling cycles with contour formulas presuppose a structured program layout and enable you to save frequently used contours in individual programs. Using the contour formula, you can connect the subcontours to a complete contour and define whether it applies to a pocket or island.

The contour milling cycles conduct comprehensive and complex internal calculations as well as the resulting machining operations. For safety reasons, always run a graphical program test before machining! This is a simple way of establishing whether the operation calculated by MillPlus will produce the desired results.



The complete contour is machined with functions G283 to G286

The machining dimension (such as the milling depth, finishing allowance, and safety clearance) are entered as CONTOUR DATA in cycle G283.

Properties of the subcontours

- By default, MillPlus assumes that the contour is a pocket. Do not program a radius compensation. In the contour formula, you can convert a pocket to an island by making it negative.
- Feeds F and additional functions M are not permitted.
- Coordinate conversion is not permitted
- Geometry calculation (G64/G63) is permitted
- Although the subprograms can contain coordinates in the spindle axis, such coordinates are ignored.
- The working plane is defined in the first coordinate block of the subprogram.

Properties of the fixed cycles

- MillPlus automatically positions the tool to the safety clearance before a cycle.
- Each level of infeed depth is milled without interruptions since the cutter traverses around islands instead of over them.
- The radius of "inside corners" can be programmed - the tool keeps moving to prevent surface blemishes at inside corners (this applies for the outermost pass in the roughing and side finishing cycles).
- The contour is approached in a tangential arc for side finishing.
- For floor finishing, the tool again approaches the workpiece on a tangential arc (for tool axis Z, for example, the arc may be in the Z/X plane).
- The contour is machined throughout in either climb or up-cut milling.

Entering a contour formula

You can use a mathematical function to interlink various contours in a mathematical formula.

Mathematical function
Intersected with e. g. EC10 = EC1 & EC5
Joined with e. g. EC25 = EC7 EC18
Joined, but without intersection e. g. EC12 = EC5 ^ EC25
Intersected with complement of e. g. EC25 = EC1 \ EC2
Parentheses e. g. EC10 = (EC1 \ EC2) \ EC3 \ EC4
Defining a single contour e. g. EC12 = EC1

Superimposed contours

By default, MillPlus considers a programmed contour to be a pocket. With the functions of the contour formula, you can convert a contour from a pocket to an island.

Pockets and islands can be overlapped to form a new contour. You can thus enlarge the area of a pocket by another pocket or reduce it by an island.

Subprograms: overlapping pockets



The following programming examples are contour description programs that are defined in a contour definition program. The contour definition program is called via the **G282** function in the actual main program.

Pockets A and B overlap.

The pockets are programmed as full circles.

Contour description program 1: pocket A

```
'POCKET_A.MM
```

```
G1 X65 Y65
```

```
G2 I65 J50
```

Contour description program 2: pocket B

```
'POCKET_B.MM
```

```
E1=35 E2=50 E3=25
```

```
G1 X=E1 Y=E2-E3
```

```
G3 I=E1 J=E2
```

Area of inclusion (joined with)

Both surfaces A and B are to be machined, including the overlapping area:

- The surfaces A and B must be entered in separate programs without radius compensation.
- In the contour formula, the surfaces A and B are processed with the "joined with" function

Contour definition program:

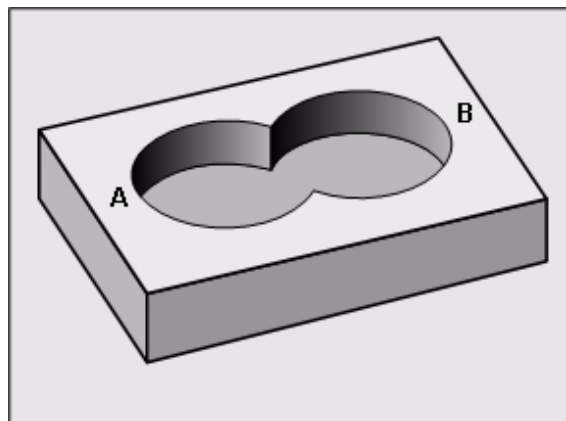
...

EC1="POCKET_A.MM"

EC2="POCKET_B.MM"

EC10= EC1 | EC2

...



Area of difference (intersected with complement of)

Surface A is to be machined without the portion overlapped by B:

- The surfaces A and B must be entered in separate programs without radius compensation.
- In the contour formula, the surface B is subtracted from the surface A with the "intersected with complement of" function

Contour definition program:

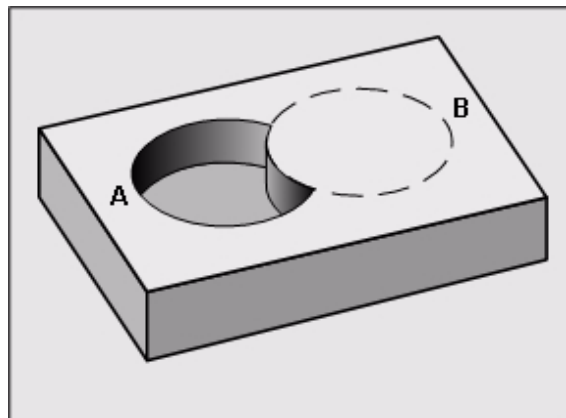
...

EC1="POCKET_A.MM"

EC2="POCKET_B.MM"

EC10= EC1 \ EC2

...



Area of intersection (intersected with)

Only the area where A and B overlap is to be machined. (The areas covered by A or B alone are to be left unmachined.)

- The surfaces A and B must be entered in separate programs without radius compensation.
- In the contour formula, the surfaces A and B are processed with the "intersected with" function

Contour definition program:

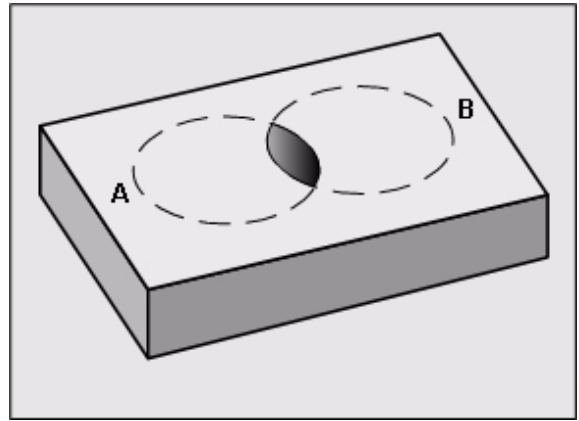
...

EC1="POCKET_A.MM"

EC2="POCKET_B.MM"

EC10= EC1 & EC2

...



Area of inclusion without intersection (joined with but without intersection)

Both surfaces A and B are to be machined, excluding the area overlapped by A and B:

- The surfaces A and B must be entered in separate programs without radius compensation.
- In the contour formula, the surfaces A and B are processed with the "joined with but without intersection" function

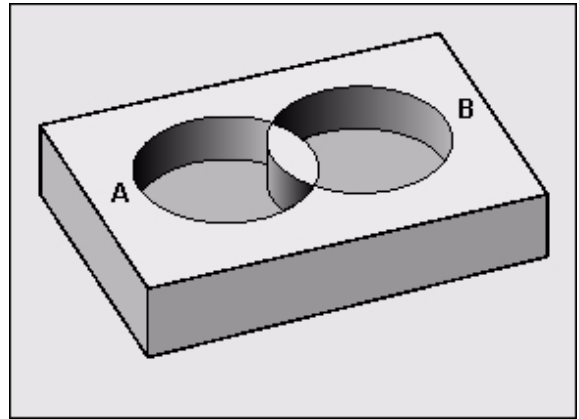
Contour definition program:

...

EC1="POCKET_A.MM"

EC2="POCKET_B.MM"

EC10= EC1 ^ EC2



G285 T10=0 C1=10 F=350 F2=250	Contour roughing
T10001 M6	Call tool: roughing cutter
G179	Call milling cycle: contour roughing
G286 B3=1 C1=10 F=400 I1=1 I2=0	Contour finishing
T10002 M6	Call tool: finishing cutter
G179	Call milling cycle: contour finishing
G280	Contour milling cycle end
M30	Program end

Contour definition program

'MODEL.MM	Contour definition program
EC1="CIRCLE1.MM"	Definition of subprogram "Circle1"
EC2="CIRCLE2.MM"	Definition of subprogram "Circle2"
EC3="TRIANGLE.MM"	Definition of subprogram "Triangle"
C4="QUADRANGLE.MM"	Definition of subprogram "Quadrangle"
EC10=(EC1 EC2) \ EC3 \ EC4	Contour formula

Contour description programs

'CIRCLE1.MM	Contour description program: circle at right
G1 X65 Y25	
G2 I65 J50	

'CIRCLE2.MM	Contour description program: circle at left
E1=35 E2=50 E3=25	
G1 X=E1 Y=E2-E3	
G3 I=E1 J=E2	

TRIANGLE	Contour description program: triangle at right
G1 X57 Y42	
G1 X73 Y42	
G1 X65 Y58	
G1 X57 Y42	

QUADRANGLE	Contour description program: square at left
G1 X27 Y58	
G1 X43	
G1 X42	
G1 X27	
G1 Y58	

7.21 G280 End Contour Milling

This function ends the contour milling cycle description.

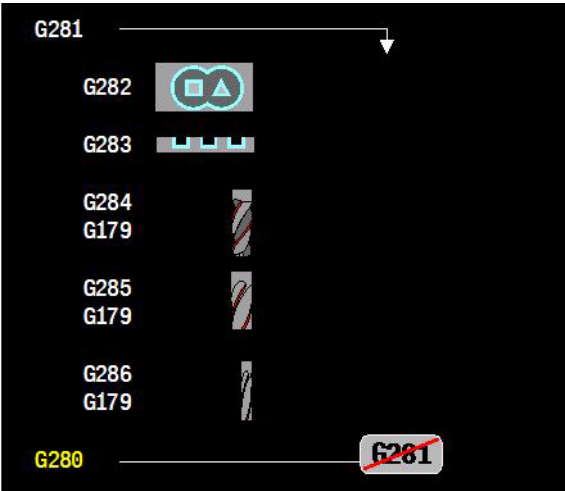
Address description

No specific addresses.

Application

Modality

Function G280 ends the contour milling cycle description started with G281.



7.22 G281 Begin Contour Milling

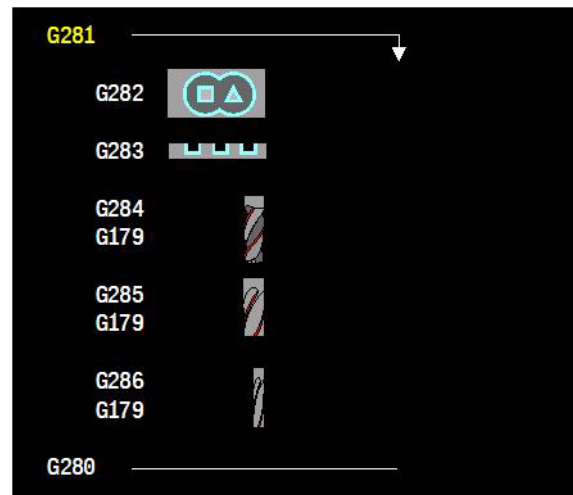
Function G281 begins the contour milling cycle description.

Contour milling cycles enable you to form complex contours by combining subcontours (pockets or islands). MillPlus calculates a complete contour from the list of subcontours.

Application

Modality

G281 is modal with G280.



7.23 G282 Contour Definition Program

Function G282 enables you to select a program with contour definitions, from which MillPlus takes the contour descriptions:

Address description

- **N= contour program name** The selected contour program can be called with the file name, with or without file path. In the N= parameter, the complete contour program name (without or without file path and including <.mm>) can be programmed between double quotation marks <">. e.g. N="Contour-1.mm". No other file name extensions are permitted.
- **N5= folder** The selected contour program can be located in a different directory. The path to this directory must be enclosed in double quotation marks <"> and should be entered separately in the N5= parameter, or should be before the file name in the N= parameter. The path must be entered complete and absolute, e.g. N5="%URS%\nc_prog\Part1\Programs\" or N="%USR%\nc_prog\Part1\Programs\Contour-1.mm"

Application

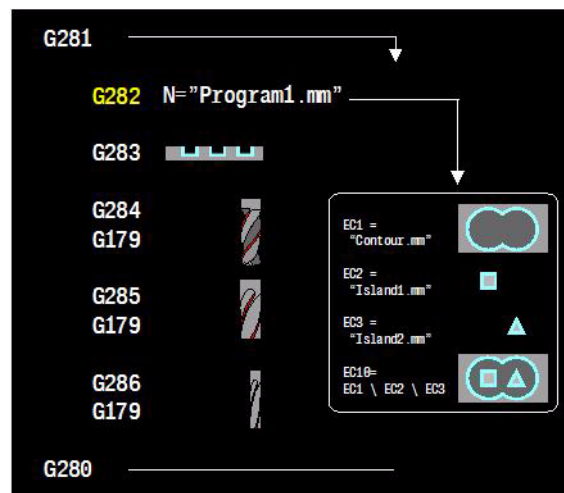
Function G282 remains active up to G280.

The bottom of the pocket must lie parallel to the machining plane.

The pocket edges must be perpendicular to the bottom of the pocket

Modality

Program function G282 before functions G284-G286.



7.24 G283 Contour Data Definition

Function G283 is used to enter machining data for the subprograms with the subcontours.

Address description

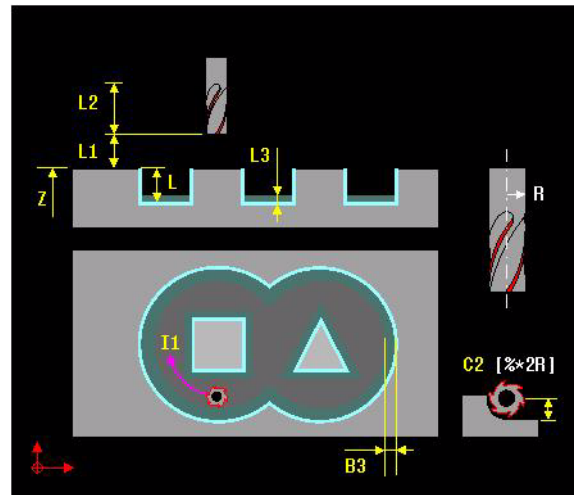
- ▶ **Z workpiece surface coordinate** Absolute coordinate of the workpiece surface
- ▶ **L depth** (incremental): Distance between workpiece surface and bottom of pocket.
- ▶ **L1= 1st setup clearance** (incremental): Distance between tool front face and workpiece surface
- ▶ **L2= 2nd setup clearance** (incremental)
- ▶ **L3= finishing allowance bottom** (incremental): Finishing allowance for bottom.
- ▶ **B3= finishing allowance sides** (incremental): Finishing allowance in the working plane.
- ▶ **C2= proportional cutting width**
- ▶ **R rounding radius** Rounding radius at inside "corners"; entered value refers to the tool midpoint path
- ▶ **I1= milling 1=climb -1=conventional**

Application

Function G283 remains active up to G280.

The machining data defined with G283 is applicable for functions G284 to G286.

If you program Depth = 0, MillPlus will not execute the function G283.



7.25 G284 Contour Pilot Drilling

Function G284 is used to pilot drill one (or multiple) cutter infeed point(s). For the cutter infeed points, MillPlus takes the side and bottom finishing allowances as well as the radius of the roughing tool into account. The cutter infeed points also serve as starting points for roughing.

Address description

- ▶ **T10= roughing tool number** Number of the roughing tool
- ▶ **T12= roughing tool offset index**
- ▶ **C1= plunging depth** Dimension by which the tool plunges in each infeed
- ▶ **F feed for plunging** Traversing speed in mm/min for drilling

Application

Function G284 is executed with function G179.

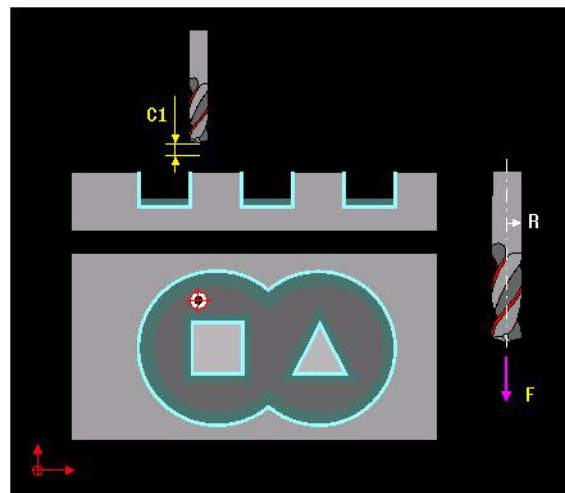


Before programming, note the following

Program a positioning block for the starting point (hole center) in the working plane with radius compensation G40.

Procedure

- 1 MillPlus positions the tool in the tool axis at rapid traverse to the safety distance above the workpiece surface.
- 2 The tool drills to the first plunging depth at the programmed feed rate F.
- 3 When it reaches the first plunging depth, the tool retracts at rapid traverse to the starting position and advances again to the first plunging depth minus the advanced stop distance.
- 4 The tool then advances by another infeed depth at the programmed feed rate F.
- 5 MillPlus repeats this process (3 to 4) until the programmed depth is reached.
- 6 The tool is retracted from the hole bottom to the set-up clearance or - if programmed - to the 2nd safety distance at rapid traverse.



Example

Contour pilot drilling

G284 T10=1 C1=5 F100

T2 M6

G179

G284	Define contour pilot drilling
T2 M6	Call tool: drill
G179	Perform contour pilot drilling

7.26 G285 Contour Roughing

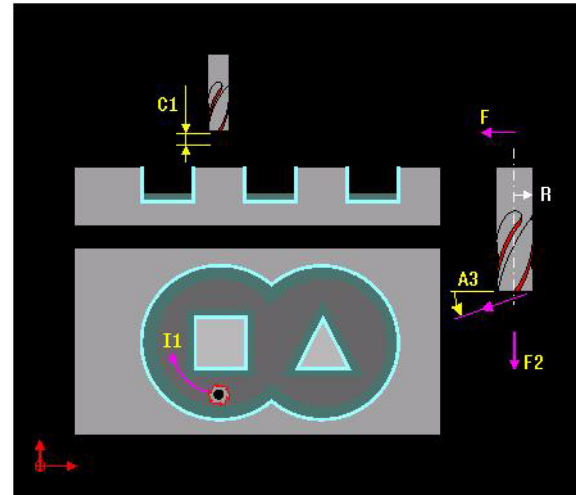
MillPlus uses function G285 to rough-out a pre-defined contour pocket.

Address description

- ▶ **T10= coarse roughing tool number** Number of the tool with which the TNC has already coarse-roughed the contour. If coarse roughing was performed, enter "0". If you enter a value other than zero, MillPlus will only rough-out the portion that could not be machined with the coarse roughing tool.
If the portion that is to be roughed cannot be approached from the side, MillPlus will mill in a reciprocating plunge-cut. For this purpose, you must enter the tool length LCUTS in the tool table TOOL.T and define the maximum plunging ANGLE of the tool. MillPlus will otherwise generate an error message.
- ▶ **T12= roughing tool offset index**
- ▶ **C1= plunging depth**(incremental): Dimension by which the tool plunges in each infeed.
- ▶ **A3= plunging angle** Angle (0..90°) at which the tool can plunge into the workpiece. It only plunges vertically at 90°. A3 is only permitted if "ANGLE" = 0 or "ANGLE" > 0
- ▶ **F feed for milling** Milling feed rate in mm/min
- ▶ **F2= feed for plunging**: Plunge feed rate in mm/min

Default setting

A3=90, F2=0.5*F for vertical plunging and F2=F for oblique plunging.



Application

Function G285 is executed with function G179.



Before programming, note the following

If you define the plunge angle A3 at between 0.1° and 89.999°, MillPlus will mill in a reciprocating plunge-cut at the specified A3 to the coarse roughing depth.

The plunge angle (A3 or ANGLE) should be greater than 0.1°

If the portion to be roughed cannot be approached from the side, MillPlus will not traverse to the relevant roughing depth with a helix movement, but will mill in a reciprocating plunge-cut



If plunge angle A3 is not programmed, the plunge angle ANGLE of the tool table is used. If ANGLE is not entered into the tool table, the default ANGLE=90° is used.

Procedure

- 1 MillPlus positions the tool over the cutter infeed point, taking the side finishing allowance into account.
- 2 In the first plunging depth, the tool mills the contour from the inside outward at the milling feed rate F.
- 3 The island contours are milled out with a movement toward the pocket contour.
- 4 MillPlus then rough-mills the pocket contour and retracts the tool to the clearance height.

Example

Contour roughing

G285 T10=0 C1=10 F=350 F2=250

T2 M6

G179

G285	Define contour roughing
T2 M6	Call tool: roughing cutter
G179	Perform contour roughing

7.27 G286 Contour Finishing

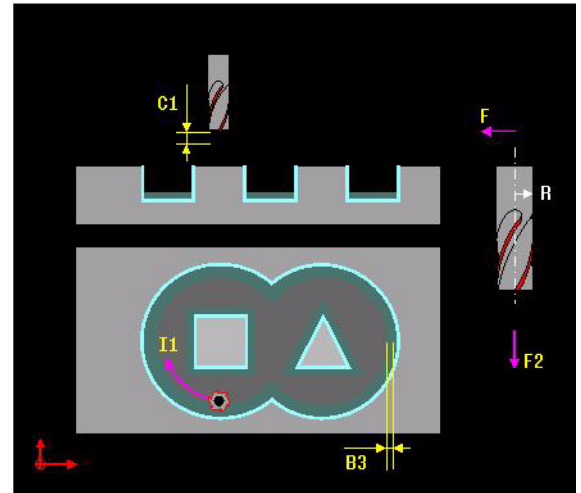
Machining data for finishing is entered in cycle G286.



Before programming, note the following The sum of the side finishing allowance (B3=) and the radius of the finish mill must be smaller than the sum of side finishing allowance (B3= from G283) and the radius of the rough mill.

MillPlus automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket.

The subcontours are approached and exited on a tangential arc. Each subcontour is finished separately.



Address description

- ▶ **B3= finishing allowance sides**(incremental): Allowance for multiple finish operations. If you enter B3 = 0, the remaining finishing allowance will be cleared.
- ▶ **C1= plunging depth**(incremental): Dimension by which the tool plunges in each infeed.
- ▶ **I1= milling 1=climb -1=conventional**
- ▶ **I2= finishing 0=complete 1=sides**
 - 0: finishing of side and bottom
 - 1: finishing of side only
- ▶ **F feed for milling** Milling feed rate in mm/min
- ▶ **F2= feed for plunging**:Plunge feed rate in mm/min

Application

Function G286 is executed with function G179.

- ▶ **R1= proportional helix radius** Percentage of the tool radius to be used as the helix radius (>0) for plunging.

Example

Contour finishing

G286 B3=1 C1=10 F=400 I1=1 I2=0

T2 M6

G179

G286	Define contour finishing
T2 M6	Call tool: finishing cutter
G179	Perform contour finishing

8

**G300-G399 G-Codes for
Macros**

8.1 Specific G Codes for Macros

Overview of G codes for macros

Error message functions

- G300 Program error call

Read functions

- G319 Read actual technology data
- G320 Read actual G data
- G321 Read tool data
- G322 Read machine constant memory
- G324 Read G group
- G326 Read actual position
- G327 Read operation mode

Overview of G codes for installation purposes

The control contains specific G codes for **installation purposes**.

However, these functions are intended for the sole use of the OEM and can result in machine damage and dangerous situations if used incorrectly.

Synchronize CNC-PLC

- G303 M19 with programmable direction
- G305 Synchronize CNC and PLC
- G338 Write IPLC marker or I/O

Read functions

- G323 Read cycle data
- G328 Read IPLC marker or I/O
- G329 Read offset from kinematic model

Write functions

- G331 Write tool data
- G333 Write cycle macro
- G339 Write offset in kinematic model

8.2 G300 Program Error Call

Activation of error messages when executing universal programs or macros.

Address description

► **D P error message number** Programming error messages (P).

Application

Interruption of program or macro execution by means of a programmed error message.

Procedure

The defined error message is set. Program execution is stopped according to the error class of the called error.

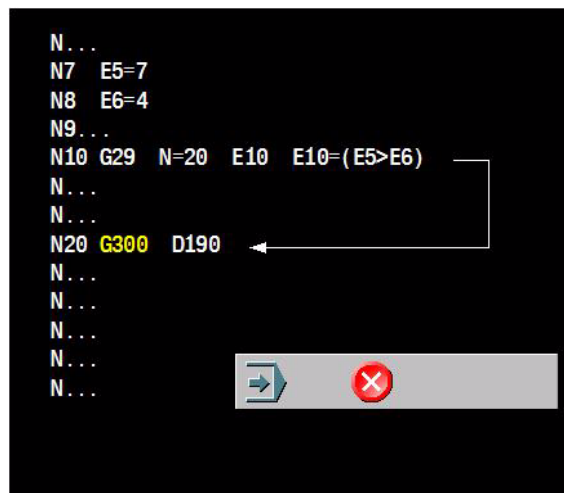
Example

Setting an error message

G29 I1 E30 N=180 E30=(E4>360)

N180 G300 D190 (PROGRAMMED R VALUE>MAXIMUM VALUE)

G29	Check whether E4>360 degrees. If yes, jump to N180.
G300	Error message: programmed value > minimum value Program execution is interrupted. Program has to be ended to enter an altered value.



8.3 G303 M19 with Programmable Direction

Spindle orientation in a programmed direction.

Address description

- ▶ **D** angle oriented spindle stop
- ▶ **I2=** direction 3=CW 4=CCW

Application

Spindle orientation in a programmable direction, for example, to avoid a collision.

Procedure

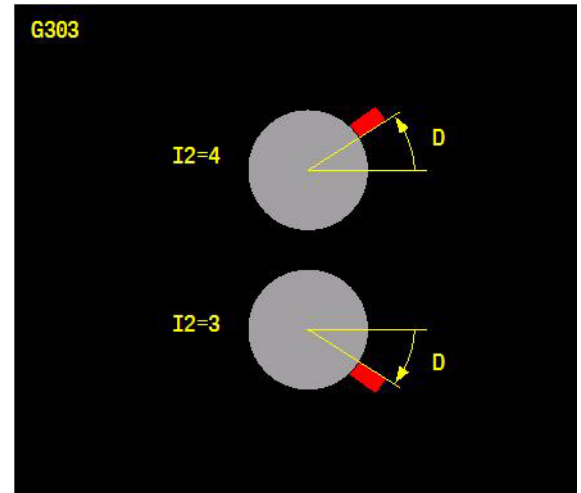
The spindle stops and orientates itself in the direction programmed with I2= to the end angle D.

Example

Spindle orientation in programmed direction

G303 M19 D15 I2=3

G303 Spindle orientation with M19 D15 in clockwise direction



8.4 G305 Synchronize CNC and PLC

Wait until the IPLC has set a defined IPLC signal.

Address description

- ▶ N5= IPLC marker or I/O number
- ▶ E E parameter

Application

Wait until the IPLC has set a defined IPLC signal.

Procedure

G305 does not perform any actions until the movement in the preceding block is finished. When the signal condition is satisfied, machining will continue.

Example

Waiting until an IPLC marker is set

```
N1004 G305 N5="MS_FUNCTION_M10 < 1"
N1005 G305 N5="ACHSPOS::M_ACHSPOS_INIT > 0"
```

G305	Wait until IPLC marker MS_Function_M10 is less than 1
G305	Wait until IPLC marker M_ACHSPOS_INIT in module ACHSPOS.MOD is greater than 0

```
G305
N...
N10 G305 N5="MS_Funktion_M10 < 1" E10
N...
```

8.5 G319 Read Actual Technology Data

Read actual value of F (feed rate), S (speed), S1 (cutting speed/ rotational speed), or T (tool number)

Address description

- ▶ **G read actual technology data**
- ▶ **I1= 1-7 (F,S,T,S1,F1,F3,F4)**
 - **I1=1** F feed
 - **I1=2** S speed
 - **I1=3** T tool number
E= (I2=0: without sister tool number)
 - **I1=4** S1= cutting speed for turning
 - **I1=5** F1= constant cutting feed (F1= with G41/G42)
E= (0=none, 1=inside only, 2=inside and outside, 3=outside only)
 - **I1=6** F3= in depth feed
 - **I1=7** F4= in plane feed
- ▶ **I2= 0= programmed value** (optional)
 - **I2=0** programmed value
- ▶ **E E parameter**

Default setting

I2=0

Application



If an address has no value, the specified E parameter has no value.

Changes to V5xx

- See "G319_I2=1" on page 519.

Example

Exporting the active feed rate and saving the value.

G319 I1=1 E10

G319 I1=1 read feed rate value.
 E10 contains the feed rate value

```
G319 I1=1 I2=0 E8

I1=  1  F Feed
     2  S Speed
     3  T Tool number
     4  S1= Cutting speed for turning
     5  F1= Constant cutting feed
     6  F3= In depth feed
     7  F4= In plane feed

I2=   0  Programmed value
```

8.6 G320 Read Actual G Data

Read the address values of current modal G codes and save these values to the E-parameter provided.

Address description

- ▶ **G read actual G data**
- ▶ **E E parameter**
- ▶ **I1= selection number**
- ▶ **G7 tilting working plane**
E= (-180 - 180)[degrees]
 - **I1=1** solid angle of A axis
 - **I1=2** solid angle of B axis
 - **I1=3** solid angle of C axis
- ▶ **Result of G17, G18, G19, G180, and G182**
E= (1=X, 2=Y, 3=Z, 4=A, 5=B, 6=C)
 - **I1=10** main axis (1-3)
 - **I1=11** parallel axis (1-6)
 - **I1=12** tool axis (1-3)
- ▶ **G25/G26 enable/disable feed/spindle override**
E= (0=F and S active, 1=F=100%, 2=S=100%, 3=F and S=100%)
 - **I1=13** feed/speed override (0-3)
- ▶ **G27/G28 reset/activate positioning functions**
 - **I1=16** positioning logic (I5=0 or 1)
E= (0=with positioning logic, 1=without positioning logic)
 - **I1=17** reduced acceleration (I6=) **E=** (5-100)[%]
 - **I1=18** contour accuracy (I7=)
E= (0-10.000) [mm|inch].
- ▶ **G39 tool offset change**
E= ([mm|inch])
 - **I1=19** additional tool compensation (L)
 - **I1=20** additional tool compensation (R)

G320 I1=... E...		
I0=	Description	Range
1-3	Solid angle of A/B/C-axis in G7	<-180,180>
10	Main axis	{1,2,3}
11	Parallel axis	{1,2,...,6}
12	Tool axis	{1,2,3}
13	Feed / Speed override	
19	Additional tool compensation L	
20	Additional tool compensation R	
27-33	G54 ZPS in X...B4-axis	
34-40	G92/G93 ZPS in X...B4-axis	
48	Scale factor in machining axis	<0,100>
49	Scale factor in tool axis	<0,100>
50-55	Mirror factor in X...C-axis	<-1,1>
56-61	System axis number of X...C	{0,1,...,6}
70	G100 I1=	{0,1}

- ▶ **G54 activate zero point shift**
E= ([mm]|[Inch])[degrees]
 - **I1=27** ZPS in X axis
 - **I1=28** ZPS in Y axis
 - **I1=29** ZPS in Z axis
 - **I1=30** ZPS in A axis
 - **I1=31** ZPS in B axis
 - **I1=32** ZPS in C axis
 - **I1=33** angle of rotation
- ▶ **G92/G93 zero point shift incr./abs.**
E= ([mm]|[Inch])[degrees]
 - **I1=34** ZPS in X axis
 - **I1=35** ZPS in Y axis
 - **I1=36** ZPS in Z axis
 - **I1=37** ZPS in A axis
 - **I1=38** ZPS in B axis
 - **I1=39** ZPS in C axis
 - **I1=40** angle of rotation
- ▶ **G72/G73 cancel/activate mirror image and scaling**
 - **I1=48** scale factor [%|factor] in machining axis (A4=)
 [%|factor] dependent on machine parameter dimension
 - **I1=49** scale factor [%|factor] in tool axis (A4=)
 [%|factor] dependent on machine parameter dimension
 - **I1=50** mirror factor in X axis
 - **I1=51** mirror factor in Y axis
 - **I1=52** mirror factor in Z axis
 - **I1=53** mirror factor in A axis
 - **I1=54** mirror factor in B axis
 - **I1=55** mirror factor in C axis
E= (-1=mirroring active, 1=mirroring not active)
- ▶ **System axis number** (determined by index of machine parameter)
E= (0=not active, 1–6 axis number)
 - **I1=56** X axis
 - **I1=57** Y axis
 - **I1=58** Z axis
 - **I1=59** A axis
 - **I1=60** B axis
 - **I1=61** C axis
- ▶ **G7 G106 and G108 kinematic calculation: OFF/ON**
 - **I1=70** values of **I1= address G108**
E= (0=G106 active, 1=G108 active in the head and in the table if applicable)

Application

Changes to V5xx

- See "G320_I1" on page 520.

Example

Reading current G data and saving the value in an E parameter.

G320 I1=10 E11

G320 I1=11 E12

G320 I1=12 E13

G320 I1=10 Read main axis
 E11 contains the result
 E11=1 X axis is the main axis.

G320 I1=11 Read parallel axis
 E12 contains the result
 E12=2 Y axis is the parallel axis.

G320 I1=12 Read tool axis
 E13 contains the result
 E13=3 Z axis is the tool axis.

8.7 G321 Read Tool Data

Read values from the tool table.

Address description

- ▶ **G** read tool data
- ▶ **T** tool number
- ▶ **T2=** sister tool index (optional)
- ▶ **E** E parameter
- ▶ **I1=** tool address (1=L .. 37=LCUTS)
 - I1=1 L tool length
 - I1=2 R tool radius
 - I1=3 R2 tool corner radius
 - I1=4 DL length allowance
 - I1=5 DR radius allowance
 - I1=8 CUT number of tool teeth
 - I1=9 DIRECT cutting direction
 - I1=10 ANGLE plunge angle
 - I1=11 PTYP tool type for magazine table
 - I1=12 TS tool status
 - I1=13 TIME1 tool life (time unit is minutes)
 - I1=14 CUR_TIME tool life (passed cutting time)
 - I1=16 LBREAK breakage tolerance: length
 - I1=24 LTOL wear tolerance: length
 - I1=25 RTOL wear tolerance: radius
 - I1=26 L-OFFS measuring offset: length
 - I1=27 R-OFFS measuring offset: radius
 - I1=31 DR2 tool corner radius offset
 - I1=32 TL tool locked
 - I1=37 LCUTS cut length in the tool axis

G321 T100 T2=1 I1=1 E8

```

I1= 1  ..  5 = L      R      R2      DL      DR
      8  .. 12 = CUT   DIRECT ANGLE   PTYP   TS
     13 .. 14 = TIME1  CUR_TIME
     16      = LBREAK
     24 .. 27 = LTOL   RTOL   L_OFFS  R_OFFS
     31      = DR2
     36      = TL
     37      = LCUTS


```

Application

Tool number and position

The tool number (T) must be known. The position (P) in the tool table cannot be read.

Reading a tool table address without a value



If the address in the tool table is empty, the specified E parameter has no value.

Changes to V5xx

■ See "G321" on page 522.

Example

Program blocks for reading the tool table.

G321 T10 I1=1 E1
G321 T10 I1=2 E10
G321 T10 I1=3 E20
G321 T10 I1=4 E2
G321 T10 I1=5 E11
E3=E1+E2
E12=E10+E11

G321	Read request T tool number I1= tool address data E1 E parameter number. (L) tool length is set in E parameter 1
G321	R (tool radius) is set in E parameter 10
G321	R2 (tool corner radius) is set in E parameter 20 (if R2 has no value, E20 is cleared)
G321	DL (length allowance) is set in E parameter 2
G321	DR (radius allowance) is set in E parameter 11 The correct tool length (E3) is L+DL (E1+E2) The correct tool radius (E12) is R+DR (E10+E11)

8.8 G322 Read Machine Constant Memory

Read a machine parameter (value or string) and save its contents in the E-parameter provided (value or string).

Address description

- ▶ **G** read machine constant memory
- ▶ **N5=** name of machine parameter
- ▶ **01=** E parameter for numerical value (optional)
- ▶ **02=** E parameter for string value (optional)

Application

Machine parameter (N5=)

The machine parameter is defined by a path. The various elements in the path are separated with <:>. The path is specified by defining the element in the corresponding CFG file. The current value of the relevant machine parameter is returned as a numerical value or "string". The path is case-sensitive.

E parameter number

O1= defines the number of the E parameter to which the numerical result is written, while O2= defines the number of the E parameter to which the "string" is written.

Reading machine parameters without value

If an unavailable machine parameter is read, an error message is issued. The E parameter is not modified.

Changes to V5xx

- See "G322" on page 523.

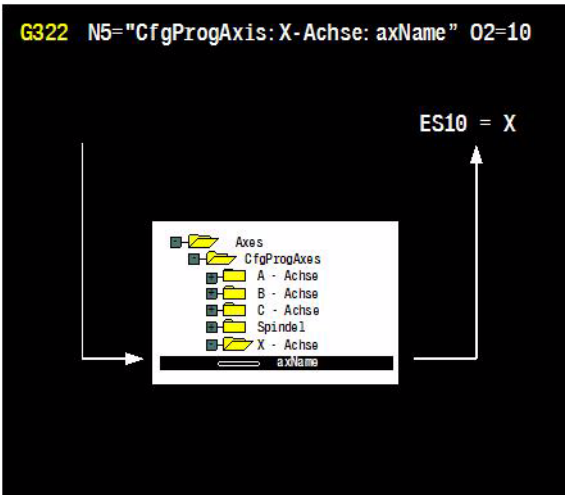
Example

Reading a numerical value and a string

G322 N5="CfgUnitOfMeasure:unitOfMeasure" O1=3

G322 N5="CfgProgAxis:X-Axis:axName" O2=10

G322 O1=3 returns E3 = 0
G322 O2=10 returns E10 = "X"



8.9 G323 Read Cycle Data


Read cycle data from the internal memory. The data is used to execute and display the correct cycle macro.

Address description

- ▶ **G read cycle data**
- ▶ **01= E parameter subprogram number** (optional)
- ▶ **02= E parameter G number of cycle** (optional)
- ▶ **03= first E parameter for cycle definition** (optional)
- ▶ **04= last E parameter for cycle definition** (optional)

Application

E parameter number



If an address has no value, the specified E parameter has no value.

01= defines the number of the E parameter to which the macro number is written.

02= defines the number of the E parameter to which the cycle number is written.

03= defines the number of the first E parameter to which the saved cycle definition is written.

04= defines the number of the last E parameter to which the saved cycle definition is written.

Changes to V5xx

- See "G323" on page 524.

Example

Reading a numerical value and a string

G81 X0 Y0 Z10
G323 O1=10

G81	Drilling cycle
G323	O1=10 returns E10 = 81

G323	O1=...	O2=...	O3=...	O4=...
Description				
O1=	E-par. macro ID	(999xxxx)		
O2=	E-par. cycle number	(modal G81-89, G781-799)		
O3=	First E-par. for cycle definition			
O4=	Last E-par. for cycle definition			

8.10 G324 Read G Group

Read a current modal G code and save this value to the E-parameter provided.

Address description

► **G** read G group

► **E** E parameter

► **I1=** G group

E= number of G code

- **I1=1** G0, G1, G2, G3, G6, G31, G33
- **I1=2** G17, G18, G19
- **I1=3** G40, G41, G42, G43, G44, G141
- **I1=4** G53, G54, G54_I, G55, G56, G57, G58, G59
- **I1=5** G63, G64
- **I1=7** G70, G71
- **I1=8** G90, G91
- **I1=10** G94, G95
- **I1=11** G96, G97 (rotation only)
- **I1=12** G36, G37 (rotation only)
- **I1=13** G72, G73
- **I1=14** G66, G67
- **I1=15** Off, G39
- **I1=16** G51, G52
- **I1=17** G196, G199
- **I1=19** G27, G28
- **I1=20** G25, G26
- **I1=22** G202, G201
- **I1=24** G180, G182
- **I1=26** Off, G141
- **I1=27** Off, G7
- **I1=28** Off, G8

G324 I1=... E...

I1 = ...

1 = G0 G1 G2 G3 G6 G31 G33	16 = G51 G52
2 = G17 G18 G19	17 = G196 G199
3 = G40 G41 G42 G43 G44 G141	18 = G61 G62
4 = G53 G54 G54_I G55 G56 G57 G58 G59	19 = G27 G28
5 = G64 G63	20 = G25 G26
7 = G70 G71	21 = G9
8 = G90 G91	22 = G201 G202
10 = G94 G95	24 = G180 G182 G180_XZC
11 = G96 G97	26 = G141
12 = G36 G37	27 = G7
13 = G72 G73	28 = G8
14 = G66 G67	
15 = G39	

Application

Reading a group without a value

If the group or the G code does not exist, the E parameter is not modified.

Results

Generally, the result is equal to the value of the modal G code. For example: When G40 is active, G324 I1=3 returns the value 40 as the result.

Exceptions are:

- Off returns the value 0.
- G26_S, G26_F_S returns 26.
- G54_I returns 54.nn, where nn is the index.
- G180_XYZ returns 180.

Changes to V5xx

- See "G324_I1" on page 524.

Example

Reading G code (I1=2) and saving the value to the E parameter 10.

G324 I1=2 E10

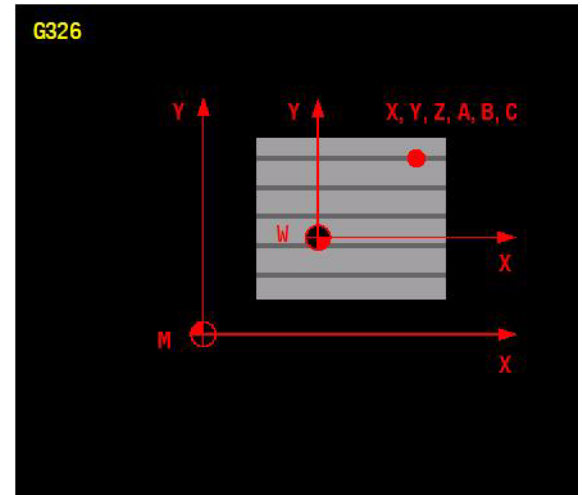
G324	I1=2 Read G code group 2 E10 contains the result E10 =17 G17 is active
------	---

8.11 G326 Read Actual Position

Read a current position value and save this value to the E-parameter provided.

Address description

- ▶ **G read actual position**
- ▶ **I1= 0=workpiece, 1=machine**
I1=0 position to workpiece zero point (default)
I1=1 position to machine zero point
- ▶ **I2= 0=programmed, 1=actual**
I2=0 programmed position (default)
I2=1 actual position
- ▶ **I3= 0=current, 1=cycle pattern home position**
I3=0 current position (default)
I3=1 cycle pattern home position
Returns the requested position, but compensated for the home position of the cycle pattern. If the actual position is identical to the cycle pattern end position (G336 I2=1), the cycle pattern home position (G336 I2=0) is returned. Comment: With incremental programming, for example, this function enables the program to be continued from the cycle pattern home position instead of the actual position.
- ▶ **X7= E parameter for X position**
- ▶ **Y7= E parameter for Y position**
- ▶ **Z7= E parameter for Z position**
- ▶ **A7= E parameter for A position**
- ▶ **B7= E parameter for B position**
- ▶ **C7= E parameter for C position**
- ▶ **D7= E parameter for S position**
- ▶ **U7= E parameter for U position**
- ▶ **V7= E parameter for V position**
- ▶ **W7= E parameter for W position**



Application

Reading unavailable axes



If the axis is not available, the specified E parameter has the value -999999999.

Reading with graphic simulation

With graphic simulation, the X, Y, and Z axes are read correctly. The rotary axes remain at zero.

Changes to V5xx

- See "G326" on page 524.

Example

Program continuation after the contour milling cycle.

G280

G326 I1=0 I2=0 X7=20 Y7=21

G29 E1 N=90 E1=E20 >100

G29 E1 N=90 E1=E20 <-100

G0 X-110 Y100

N90

G280	End contour milling
G326	Unknown current end position of X and Y.
G29	When actual X position >100, jump to N90
G29	When actual X position <-100, jump to N90
G0	G0 movement after X-110, if the current X position lies between 100 and -100. This allows an obstacle to be avoided, for example.

8.12 G327 Read Operation Mode

Read the current operation mode and save this value to the E-parameter provided.

Address description

► **G read operation mode**

► **I1= active mode (1-6)**

I1=0 not active
I1=1 free entry
I1=2 single block
I1=3 graphics
I1=5 search
I1=6 demo

► **E E parameter**

Application

Changes to V5xx

■ See "G327" on page 525.

Example

Reading the operating mode (I1=1) and saving the value to the E parameter 10.

G327 I1=1 E10

G327 Check I1=1 to establish whether free entry is active.
 E10 contains the result: 0= not active, 1= active

G327 I1=... E...

1 = Free entry / Freie Eingabe
2 = Single block / Einzelsatz
3 = Graphics / Grafik
5 = Search / Suchen
6 = Demo

8.13 G328 Read IPLC Marker or I/O

Read an IPLC marker or input/output and save this value to the E parameter provided.

Address description

- ▶ **N5= signal name** Defines the symbolic name of the read PLC signal.
- ▶ **O1= E parameter for PLC signal numerical value** Defines the E parameter to which the read value of the PLC signal is written when program execution continues.
- ▶ **O2= E parameter for PLC signal string value** Defines the E parameter to which the read value of the PLC text is written when program execution continues.

Application

Reading of IPLC values for use during program or macro execution.

Changes to V5xx

- See "G328" on page 525.

Procedure

G328 does not perform any actions until the movement in the preceding block is finished. The PLC signal defined with N5= is read and written to the E parameter.

```
G328 N5="..." O1=... O2=...
```

```
G328 N5="M9586" O1=...
G328 N5="MS_Funktion_M10" O1=...
G328 N5="MS_Programmunterbrechung[10]" O1=...
G328 N5="ACHSPOS::M_ACHSPOS_INIT" O1=...
```

Example

Different ways of reading an IPLC marker

```
N1002 G328 N5="M9586" O1=6
N1003 G328 N5="MS_FUNCTION_M10" O1=7
N1004 G328 N5="MS_PROGRAMINTERRUPT[10]" O1=8
N1005 G328 N5="ACHSPOS::M_ACHSPOS_INIT" O1=16
N1110 IF (E8=0) THEN
N1126 END IF
```

- G328 Read IPLC marker M9586
- G328 Read IPLC marker MS_Function_M10
- G328 Read 10th IPLC marker of array MS_PROGRAMINTERRUPT
- G328 Read IPLC marker M_ACHSPOS_INIT in module ACHSPOS.MOD
- IF...END IF Blocks N1110 to N1126 are executed only if E8 is equal to zero.

8.14 G329 Read Offset from Kinematic Model

Read a kinematic element and save this value to the E parameter provided.

Address description

- ▶ **I1= read mode** Defines how the kinematic model is read.
 - 0 = directly via "key" (default)
 - 1 = read sequential: 1st element
 - 2 = read sequential: next element
 - 3 = linear shift of a rotary axis
 - 4 = total linear shift of a rotary axis. In table: from machine base. In head: from spindle nose.
 - 5 = rotary axis presence
- ▶ **I2= rotary axis (4=A,5=B,6=C)** Effective only for I1=3, 4, or 5; used in conjunction with I3=. In the kinematic model, the linear shifts are specified for each rotary axis. In the individual elements, the shifts are defined in an X, Y, and Z direction. Multiple elements, e. g. one for the basic shift and a second for a compensation shift, can be defined for each direction X, Y, or Z. I2= defines the rotary axis from which the linear shift(s) in one direction are read. Unless programmed, the value belonging to the rotary axis described first is returned.
 - 4 = A axis
 - 5 = B axis
 - 6 = C axis
- ▶ **I3= linear axis direction (1=X,2=Y,3=Z)** Effective only for I1=3 or 4; used in conjunction with I2= and defines the linear shift direction that is read in the rotary axis defined (with I2=).
 - 1 = X direction
 - 2 = Y direction
 - 3 = Z direction
- ▶ **N5= key** Effective only for I1=0 (read via "key"). Defines the "key" of the kinematic element being read. Note: the key is case-sensitive.



- ▶ **01= E parameter for status**
 - 0 = no error
 - 1 = error: unknown "key"
 - 2 = warning: no rotary axis
 - 3 = error: other errors
 - 4-9 reserved for errors
 - 10 = warning: end of model. Possible only with I1=2 (sequential read)
- ▶ **02= E parameter for element type** 1 = CfgKinSimpleTrans, 2 = CfgKinSimpleAxis, 3=CfgKinAnchor.
- ▶ **03= ES parameter for Key** String in "key" of the read element.
- ▶ **04= E parameter for direction** Defined only for element type O2=1 or 2 (CfgKinSimpleTrans or CfgKinSimpleAxis). Value in "dir" of the read element. 1= X direction 2=Y, 3=Z
- ▶ **05= E parameter for value** Defined only for element type O2=1 (CfgKinSimpleTrans). Value in "val" of the read element.
 - Depending on the:
 - - Read mode I1= (sequential, individual, or total)
 - - Model configuration (basic and compensation element)
 - the linear shifts of multiple elements can be returned in summated form.
- ▶ **06= E parameter for machine axis** Defined only for element type O2=2 (CfgKinSimpleAxis). Value in "axisRef" of the read element.
 - 1 = X axis
 - 2 = Y axis
 - 3 = Z axis
 - 4 = A axis
 - 5 = B axis
 - 6 = C axis
- ▶ **07= E parameter for head/table** Effective only for I1=5 (read "axis type")
 - 0 = not present
 - 1 = axis present in tool head
 - 2 = axis present in table
- ▶ **08= E parameter for angle of rotary axis** Effective only for I1=5 (read "axis type").

Application

Function G329 (and G339) can only be used for the "simple" kinematic model of MillPlus from version V600. This kinematic model is described using the Cfg entities:

- CfgKinComposeModel
- CfgKinSimpleModel
- CfgKinSimpleTrans
- CfgKinSimpleAxis
- CfgKinAnchor

Freely-definable "keys" are used to define the various elements of the model and specify the precise sequence. See the Technical Manual for an additional description

Changes to V5xx

- See "G329" on page 525.

Procedure

The values of the kinematic model can be read in various ways via G329. All values of the read kinematic element are written to E parameters.



In the case of multiple kinematic models, the value is read from the active model.

8.15 G331 Write Tool Data

Write values to tool table.

Address description

- ▶ **G** write tool data
- ▶ **T** tool number
- ▶ **T2=** sister tool index
- ▶ **E** E parameter
- ▶ **I1=** tool address (1=L .. 37=LCUTS)
- ▶ **I1=** tool address (1=L .. 37=LCUTS)
 - I1=1 L tool length
 - I1=2 R tool radius
 - I1=3 R2 tool corner radius
 - I1=4 DL length allowance
 - I1=5 DR radius allowance
 - I1=8 CUT number of tool teeth
 - I1=9 DIRECT cutting direction
 - I1=10 ANGLE plunge angle
 - I1=11 PTYP tool type for magazine table
 - I1=12 TS tool status
 - I1=13 TIME1 tool life (time unit is minutes)
 - I1=14 CUR_TIME tool life (passed cutting time)
 - I1=16 LBREAK breakage tolerance: length
 - I1=24 LTOL wear tolerance: length
 - I1=25 RTOL wear tolerance: radius
 - I1=26 L-OFFS measuring offset: length
 - I1=27 R-OFFS measuring offset: radius
 - I1=31 DR2 tool corner radius offset
 - I1=36 TL tool locked
 - I1=37 LCUTS cut length in the tool axis

The tool comment, however, cannot be changed.

G321	T100	I1=1	E8					
I1= 1	..	5	=	L	R	R2	DL	DR
8	..	12	=	CUT	DIRECT	ANGLE	PTYP	TS
13	..	14	=	TIME1	CUR_TIME			
16			=	LBREAK				
24	..	27	=	LTOL	RTOL	L_OFFS	R_OFFS	
31			=	DR2				
36			=	TL				
37			=	LCUTS				

Application

Tool number and position

The tool number (T) must be known. The position (P) in the tool table cannot be modified.

Activating new information

The modified tool information must be reactivated after the write operation. (T... M67).

Tool life

If M (G331 I1=13 E...) is written to the tool memory, M1= is also written to the tool memory simultaneously (G331 I1=14 E...). The time unit is minutes.

Changes to V5xx

■ See "G331" on page 526.

Example.

E5=100 (TOOL LENGTH)	L (tool length) is set in E parameter 5.
E6=10 (TOOL RADIUS)	R (tool radius) is set in E parameter 6.
E7=3 (TOOL CORNER RADIUS)	R2 (tool corner radius) is set in E parameter 7.
E8=0 (LENGTH ALLOWANCE)	DL (length allowance) is set in E parameter 8.
E9=0 (RADIUS ALLOWANCE)	DR (radius allowance) is set in E parameter 9.
G331 T10 I1=1 E5	L (tool length) Write E parameter 5 to the tool table.
G331 T10 I1=2 E6	L (tool radius) Write E parameter 6 to the tool table.
G331 T10 I1=3 E7	R2 (tool corner radius) Write E parameter 7 to the tool table.
G331 T10 I1=4 E8	DL (length allowance) Write E parameter 8 to the tool table.
G331 T10 I1=5 E9	DR (radius allowance) Write E parameter 9 to the tool table.
T10 M67	Tool must be activated with the modified information.

E8=0.3 (LENGTH ALLOWANCE)	DL (length allowance) E parameter 8 is set to 0.3.
G331 T10 I1=4 E8	DL (length allowance) Write E parameter 8 to the tool table.
T10 M67	Tool must be reactivated with the modified information.

8.16 G338 Write IPLC Marker or I/O

Set IPLC marker or input/output.

Address description

- ▶ **N5= signal name** Defines the symbolic name of the set PLC signal.
- ▶ **E= E parameter PLC signal value** Defines the E parameter to which the value of the set PLC is written.

Application

Setting of IPLC values for use during program or macro execution.

Changes to V5xx

- See "G328" on page 525.

Procedure

G338 does not perform any actions until the movement in the preceding block is finished. The PLC signal defined with N5= is set with the value written to the E parameter.

Example

Different ways of setting an IPLC marker

N1002	G338	N5="M9586"	E6
N1003	G338	N5="MS_FUNCTION_M10"	E7
N1004	G338	N5="MS_PROGRAMINTERRUPT[10]"	E8
N1005	G338	N5="ACHSPOS::M_ACHSPOS_INIT"	E16

G338	Set IPLC marker M9586
G338	Set IPLC marker MS_Function_M10
G338	Set 10th IPLC marker of array MS_PROGRAMINTERRUPT
G338	Set IPLC marker M_ACHSPOS_INIT in module ACHSPOS.MOD

```
G338 N5="..." E...

G338 N5="M9586" E...
G338 N5="MS_Funktion_M10" E...
G338 N5="MS_Programmunterbrechung[10]" E...
G338 N5="ACHSPOS: :M_ACHSPOS_INIT" E...
```

8.17 G339 Write Offset in Kinematic Model

Write kinematic element from the E parameter provided.

Address description

- ▶ **I1= write mode** Defines how the kinematic model is written.
 - 0 = in configuration (on hard drive) (default)
 - 1 = intermittently (lost after controller is switched off)
- ▶ **I4= 0=absolute, 1=incremental** Defines whether the value overwrites the previous value or whether it is added to the existing value
 - 0 = absolute. The previous value is overwritten
 - 1 = incremental. Value is added (default)
- ▶ **I5= value** Value is written to "val" of element type CfgKinSimpleTrans with "key" from N5=
- ▶ **N5= key** Effective only for I1=0 (read via "key"). Defines the "key" of the kinematic element being written. Note: the key is case-sensitive.
- ▶ **O1= E parameter for status**
 - 0 = no error
 - 1 = error: unknown "key"
 - 2 = error: "key" must not be modified (probably incorrect Cfg element)
 - 3 = error: other errors



Application

Function G339 (and G329) can only be used for the "simple" kinematic model of MillPlus from version V600. This kinematic model is described using the Cfg entities:

- CfgKinComposeModel
- CfgKinSimpleModel
- CfgKinSimpleTrans
- CfgKinSimpleAxis
- CfgKinAnchor

Freely-definable "keys" are used to define the various elements of the model and specify the precise sequence. See the Technical Manual for an additional description

Procedure

The value of a kinematic element can be written via G339.



In the case of multiple kinematic models, the value is written from the active model.

8.18 G380 Protection Zones

Writing of protection zones to limit the traverse range. The axes are only allowed within the defined range; otherwise an error message is issued.

Address description

- ▶ **I1= activation (0=override, 1=add)** Defines the protection zones.
 - 0 = write mode: override (default) First, the active protection zones of all axes are cancelled. Then the newly programmed protection zones are activated.
 - 1 = write mode: add Enabled protection zones remain active. The newly programmed protection zones are activated only for the programmed axes.
- ▶ **X1=,Y1=,Z1=,A1=,B1=,C1= positive limit value** The programmed positions are relative to the reference point and must lie within the range of the SW limit switches.
- ▶ **X2=,Y2=,Z2=,A2=,B2=,C2= negative limit value** The programmed positions are relative to the reference point and must lie within the range of the SW limit switches.

Default setting

G380 I1=0

Application

Cancelation

Active protection zone monitoring G380 is canceled by:

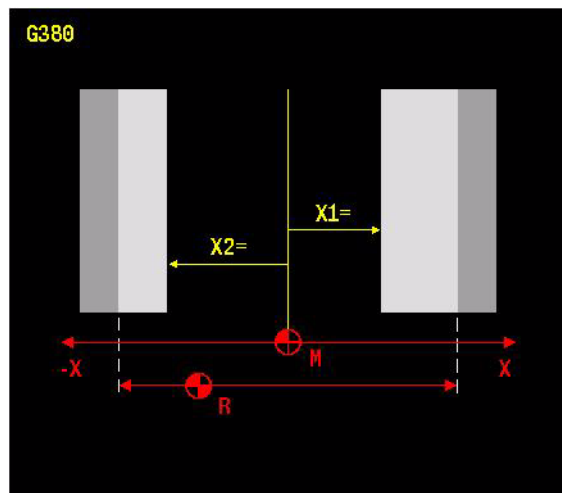
- G380 without address
- Controller activation

G380 is not canceled by:

- M30
- Cancel program



The new protection zones are NOT added to the active protection zones, but overwrite the old values.



9

**G600-G699 Measuring
Cycles**

9.1 Tool Measuring Cycles for Laser Measurements

General notes and usage

Laser measurement is complemented by the following G codes

G951	Calibrate
G953	Measure tool length
G954	Tool length, tool radius
G955	Tool edge monitoring SF
G956	Tool breakage monitoring
G957	Tool edge monitoring KF
G958	Tool measurements: length, radius, corner radius

For a description of these G codes, refer to the Blum Manual.

Availability

The machine and control must be prepared for the measuring system by the machine manufacturer. If your machine does not feature all the G codes described here, refer to your Machine Manual.

Programming

Any rotary axes are neither taken into account nor positioned.

Free working plane G7 must not be active

Machine parameters

The G code and associated functions are activated via machine parameters.

9.2 Tool Measuring Cycles for Tool Touch Probe Measuring Systems



TT stands for "Tisch-Taster" (German for "tool touch probe"), e.g. TT130 or a similar device.

General Notes on Tool Touch Probe Measuring Systems

Availability

The machine and CNC must be prepared for the measuring system by the machine manufacturer. If your machine does not feature all the G codes described here, refer to your Machine Manual.

Programming

Before any of the G600-G609 functions are called, an M24 (switch on measuring devices) must be programmed. It sets the measuring devices to the correct measuring position. To retract the measuring devices at the end of the operation, an M28 (switch off measuring devices) must be programmed.

Machine parameters

The G code and associated functions are activated via machine parameters.

9.3 Measuring Cycles

Introduction to measuring cycles

Measuring cycles in the main plane

G620	Angle measurement
G621	Position measurement
G622	Corner outside measurement
G623	Corner inside measurement
G626	Datum outside rectangle
G627	Datum inside rectangle
G628	Circle measurement outside
G629	Circle measurement inside

Special measuring cycles

G633	Angle measurement 2 holes
G634	Measurement center 4 holes

Definition

Cycle definition is independent of the machining plane (G17, G18, G19, and G7).

Axes and machining plane

The cycles are executed in the current main plane G17, G18, G19 or in the inclined plane G7

	G17	G18	G19
Principal axis	X	X	Y
Minor axis	Y	Z	Z
Working axis	XY	XZ	YZ
Tool axis	Z	Y	X or -X (G66/ G67)

In some cycles, the direction of measurement is determined by the address (I1=).

Zero point

The measured values ($I5 > 0$) can be saved in the zero point shift table for the shift that is currently active and/or in an E parameter.



If G7 is active, the measured angle cannot be set using G620 or G633 with $I5 = 2$ in the zero point. Program G620 and G633 with $I5 = 0$ $O3 = ..$ and use the relevant E parameter in an incremental G7 shift, e.g. G7 C6=E10 L1=1.

Comments

Comments are not allowed in a block with a machining cycle.

Results of activating a measuring cycle:

- G91 is deactivated.
- Radius correction is deactivated (G40 is active).
- Scaling with G72 is deactivated.
- L and R in G39 are zeroed.
- $A40 =$, $B40 =$, $C40 =$, R for calculating the feed rate of the axes.

Functions that are not allowed when a measuring cycle is called

- G36, rotations ($B4 =$) in G92/G93 and G182.
- G7 must not be active if the measured values are saved in a zero point shift ($I5 > 0$).
- Tool T0 is not allowed.



Pre-position the tool such that no collision can occur between the workpiece and clamping devices.

Explanation of addresses

The addresses described here are used in most cycles. Specific addresses are described in the relevant cycle.

- **X, Y, Z starting point** Starting point of the measuring movement. The measuring cycle is executed from here. If all the starting point coordinates are not entered, the current position of the touch probe is used.
 - Unlike a milling cycle, a measuring cycle is executed directly from the starting point (X, Y, Z).
 - The touch probe moves to the first starting point (X, Y, Z) in rapid traverse and, depending on G28, using positioning logic.

- ▶ **C1= measuring distance** Maximum distance between the start and end points of the measuring movement. (Default 10). Movement stops once the wall of the workpiece or the end of the measuring range is reached.
 - Note: If there is no contact with material within the measuring range (C1=), an error message is issued.
- ▶ **L2= safety distance** During (if I3=1) and at the end of the measurement, the touch probe moves to the safety clearance (default 0 for measurement on the outside of the workpiece or 1 mm for measurements in pockets and holes). The safety clearance (L2=) is based on the relevant starting point X, Y, Z.
- ▶ **B3= distance to corner** The distance in the principal axis between the first starting point and the corner of the workpiece. If address B4= is absent, B3= is also the distance to the next measurement around the workpiece corner. The path traced by the touch probe around the corner of the workpiece to the starting point of the 2nd measurement is the same length in both directions. For each direction, the distance is the sum of B3= and the first measuring range travelled.
- ▶ **B4= distance to corner in minor axis** The distance in the minor axis between the first starting point and the corner of the workpiece.
- ▶ **I1= measuring direction from touch probe to workpiece** I1=principal axis, I1= 2 minor axis, I1= 3 tool axis. The angular reference axis is always perpendicular to the scanning direction.
- ▶ **I3= movement between measuring movements** I3= is used to determine whether the positioning movement between measurements takes place at measuring height or at the safety clearance (L2=). With I3=0, the positioning movement between measuring movements is at measuring height and parallel to the principal axis. In the case of circular movement, the positioning movement is circular and at the feed rate. I3=1 The positioning movement between measuring movements is at the safety clearance and in a line between measurement points.
- ▶ **I4= corner number (1-4)** Specifies the corner at which the first measurement is to take place (default 1). The first measurement is always perpendicular to the principal axis. The second measurement is always perpendicular to the minor axis.
- ▶ **O1= to O7= save measured values** The measured values can be saved in the E parameters. The number of the E parameter must be entered. If no number is entered, nothing is saved. Example: O1=10 means that the result is saved in E parameter 10.
- ▶ **F measuring feed** The default is PROBE_FEED.

9.4 G620 Angle Measurement

Measurement of the inclined position of a clamped workpiece.

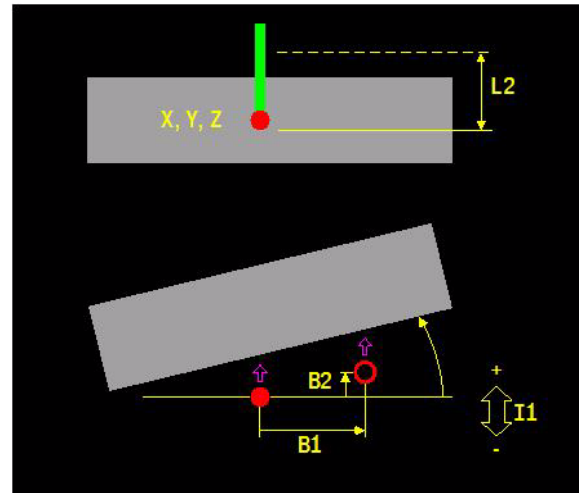
Address description

- ▶ **I1= meas.dir. $\pm 1/\pm 2/-3$ =main/minor/tl**
- ▶ **X,Y,Z starting point**
- ▶ **B1= dist. meas. positions main axis** If I1= ± 2 , B1= must be programmed (B1= must not equal zero). If I1=-3, B1= and B2= must not be programmed at the same time.
- ▶ **B2= dist. meas. positions par. axis** If I1= ± 1 , B2= must be programmed (B2= must not equal zero). If I1=-3, B1= and B2= must not be programmed at the same time. Not permitted: B1= B2= 0. On saving, the measured values are added to the active zero point shift.
- ▶ **C1= measuring distance**
- ▶ **L2= safety distance**
- ▶ **I3= 2nd measurem. via L2 0=no 1=yes**
- ▶ **I5= G5x offset 0=no 1=B4 2=A/B/C**
 - **I5=0** Do not save.
 - **I5=1** Save in the active zero point shift in the angle of rotation (G54 B4=).
 - **I5=2** Save in the active zero point shift in the rotary axis (A/B/C).
- ▶ **A1= target value angle** If the measured angle is saved in the active zero point shift (I5>0), it is used to calculate the target value. The measured position is thus given the target value for subsequent programming.
- ▶ **03= E par. measured angle**
- ▶ **F2= measuring feed**

For a description of the additional addresses, see "Explanation of addresses" on page 379.

Default setting

B1=0, B2=0, C1=20, L2=0, I3=0, I5=0, A1=0, F2=PROBE_FEED.



Application

Measuring direction

Depending on the plane selected (G17, G18 or G19), the parameter I1= determines the direction of measurement and this defines the meaning of B1= and B2=.

G17			
Measuring direction	I1= ±1	I1= ±2	I1= 3 B1= B2=
Angle plane	XY	XY	XZ YZ
Rotary axis	C	C	B A

G18			
Measuring direction	I1= ±1	I1= ±2	I1= 3 B1= B2=
Angle plane	XZ	XZ	XY ZY
Rotary axis	B	B	C A

G19			
Measuring direction	I1= ±1	I1= ±2	I1= 3 B1= B2=
Angle plane	YZ	YZ	YX ZX
Rotary axis	A	A	C B

Setting the zero point shift



If G7 is active, the measured angle cannot be set using G620 I5=2 in the zero point. Program G620 O3=.. and use the relevant E parameter in an incremental G7 shift, e.g. G7 C6=E10 L1=1.

Procedure

- 1 Rapid traverse to first starting point (X, Y, Z). If X, Y, Z are not programmed, the current position is used as the starting point.
- 2 First measurement with measurement feed (F2=) until the workpiece or the maximum measuring range (C1=) is reached.
- 3 Rapid traverse back to starting point. An error message is issued if the touch probe has not switched within the maximum measuring range (C1=).
- 4 Rapid traverse to the starting point of the 2nd measurement; depending on the value of I3=, the movement is performed at the safety clearance (L2=).
- 5 Second measurement (as described in points 2 and 3).
- 6 At the end, a rapid traverse to the safety clearance (L2=) is executed.
- 7 The measured value is saved in accordance with I5=.

Example

Aligning a workpiece

G17
G54 I3
G620 X-50 Y-50 Z-5 I1=2 B1=100 L2=10 I3=1 I5=2
G0 C0

G17	Set plane
G54	Set zero point
G620	Define and execute measuring cycle. G54 I3 is recalculated after the cycle.
G0	Rotary table is positioned at zero. (G17).

9.5 G621 Position Measurement

Measurement of a coordinate on the wall of a workpiece.

Address description

- ▶ **I1= meas.dir. $\pm 1/\pm 2/-3$ =main/minor/tl**
- ▶ **X,Y,Z starting point**
- ▶ **C1= measuring distance**
- ▶ **L2= safety distance**
- ▶ **I2= probe orientat. -1=auto 0=no**
 - **I2=-1** Measure with automatic orientation. For an all-round transmitter, orientation is in the scanning direction. In the case of a two-layer touch probe, two measurements are performed with a 180° difference in orientation.
 - **I2=0** Measure without probe orientation.
- ▶ **I5= G5x offset 0=no 1=X/Y/Z**
 - **I5=0** Do not save.
 - **I5=1** Save in the active zero point shift in the linear axes (X/Y/Z).
On saving, the measured values are added to the active zero point shift.
- ▶ **B1= target position** When the measured coordinate is saved in the active zero point shift (I5>0), it is used to calculate the nominal value. The measured coordinate is assigned the target value for further programming.
- ▶ **O1= E par. for measured position**
- ▶ **F2= measuring feed**

For a description of the additional addresses, see "Explanation of addresses" on page 379.

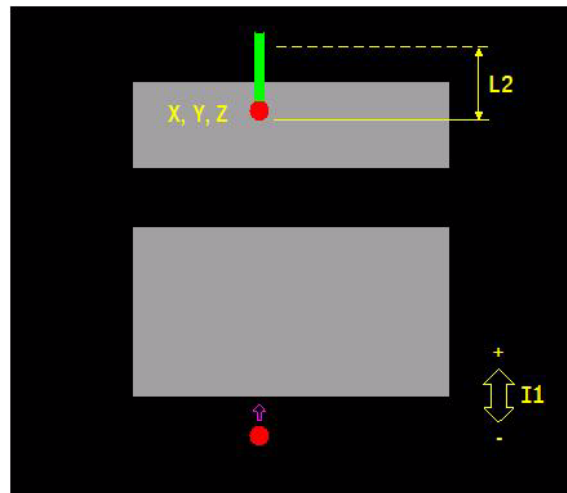
Default setting

C1=20, L2=0, I2=-1, I5=0, B1=0, F2=PROBE_FEED.

Application

Measuring direction

Depending on the plane selected (G17, G18, or G19), address I1= determines the measuring direction.



Procedure

- 1 Rapid traverse to first starting point (X, Y, Z). If X, Y, Z are not programmed, the current position is used as the starting point.
- 2 First measurement with measurement feed (F2=) until the workpiece or the maximum measuring range (C1=) is reached.
- 3 Rapid traverse back to starting point. An error message is issued if the touch probe has not switched within the maximum measuring range (C1=).
- 4 At the end, a rapid traverse to the safety clearance (L2=) is executed.
- 5 The measured value is saved in accordance with I5=.

Example

Measuring a position

```
G621 X40 Y40 Z-5 I1=2 L2=20 O1=300
```

G621	Define and execute measuring cycle. After the measuring cycle, the result is written to E parameter (E300).
------	---

9.6 G622 Corner Outside Measurement

Measurement of the corner position (outside) of an aligned workpiece.

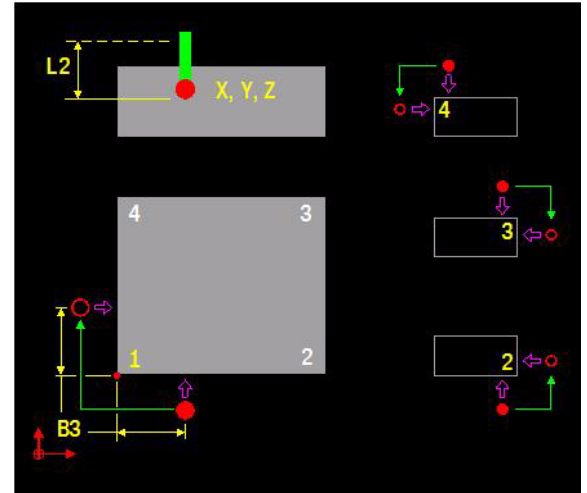
Address description

- ▶ **I4= corner number**
- ▶ **X,Y,Z starting point**
- ▶ **B3= distance to corner**
- ▶ **C1= measuring distance**
- ▶ **L2= safety distance**
- ▶ **I2= probe orientat. -1=auto 0=no**
 - **I2=-1** Measure with automatic orientation. For an all-round transmitter, orientation is in the scanning direction. In the case of a two-layer touch probe, two measurements are performed with a 180° difference in orientation.
 - **I2=0** Measure without probe orientation.
- ▶ **I3= 2nd measurem. via L2 0=no 1=yes**
- ▶ **I5= G5x offset 0=no 1=X/Y/Z**
 - **I5=0** Do not save.
 - **I5=1** Save in the active zero point shift in the linear axes (X/Y/Z).
On saving, the measured values are added to the active zero point shift.
- ▶ **O1= E par. meas. position main axis**
- ▶ **X1=, Y1=, Z1= target position corner** When the measured coordinate is saved in the active zero point shift (I5>0), it is used to calculate the nominal value. The measured coordinate is assigned the target value for further programming.
- ▶ **O1= E par. meas. position minor axis**
- ▶ **F2= measuring feed**

For a description of the additional addresses, see "Explanation of addresses" on page 379.

Default setting

I4=1, B3=10, C1=20, L2=0, I2=-1, I3=0, I5=0, X1=0, Y1=0, Z1=0, F2=PROBE_FEED.




Application

Note

- The sides must be parallel to the axes.
- The workpiece angle must be 90 degrees.
- The measured plane is perpendicular to the tool axis.

Direction of measurements

- The first measurement is always perpendicular to the principal axis.
- The second measurement is always perpendicular to the minor axis.



The support picture is in G17. The picture is not correct for a machine with exchanged axes (G18). Angle 1 must be replaced with 2, and 3 with 4.

Procedure

- 1 Rapid traverse to first starting point (X, Y, Z). If X, Y, Z are not programmed, the current position is used as the starting point.
- 2 First measurement with measurement feed (F2=) until the workpiece or the maximum measuring range (C1=) is reached.
- 3 Rapid traverse back to the first starting point. An error message is issued if the touch probe has not switched within the maximum measuring range (C1=).
- 4 Rapid traverse to the starting point of the 2nd measurement; depending on the value of I3=, the movement is performed at the safety clearance (L2=).
- 5 Second measurement (as described in points 2 and 3).
- 6 At the end, a rapid traverse to the safety clearance (L2=) is executed.
- 7 The measured value is saved in accordance with I5=.

Example

Aligning the outside corner of a workpiece

G1 X... Y... Z-5

G54 I3

G622 L2=20 B3=25 I3=1 I5=1 X1=-50 Y1=-50

G1	Position the touch probe 10 mm to the right of corner 1 and 8 mm from the front.
G54	Set zero point
G622	Define and execute measuring cycle. After the measuring cycle, the zero point shift is overwritten so that the coordinates of corner 1 are equal to X1= and Y1=

9.7 G623 Corner Inside Measurement

Measurement of the corner position (inside) of an aligned workpiece.

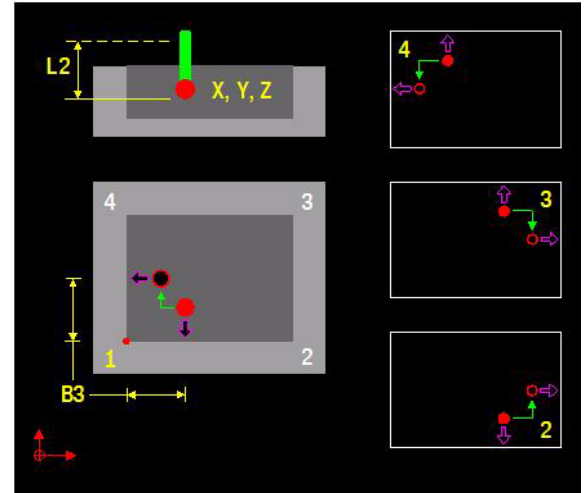
Address description

- ▶ **I4= corner number**
- ▶ **X,Y,Z starting point**
- ▶ **B3= distance to corner**
- ▶ **C1= measuring distance**
- ▶ **L2= safety distance**
- ▶ **I2= probe orientat. -1=auto 0=no**
 - **I2=-1** Measure with automatic orientation. For an all-round transmitter, orientation is in the scanning direction. In the case of a two-layer touch probe, two measurements are performed with a 180° difference in orientation.
 - **I2=0** Measure without probe orientation.
- ▶ **I3= 2nd measurem. via L2 0=no 1=yes**
- ▶ **I5= G5x offset 0=no 1=X/Y/Z**
 - **I5=0** Do not save.
 - **I5=1** Save in the active zero point shift in the linear axes (X/Y/Z).
On saving, the measured values are added to the active zero point shift.
- ▶ **X1=, Y1=, Z1= target position corner** When the measured coordinate is saved in the active zero point shift (I5>0), it is used to calculate the nominal value. The measured coordinate is assigned the target value for further programming.
- ▶ **O1= E par. meas. position main axis**
- ▶ **O1= E par. meas. position minor axis**
- ▶ **F2= measuring feed**

For a description of the additional addresses, see "Explanation of addresses" on page 379.

Default setting

I4=1, B3=10, C1=20, L2=0, I2=-1, I3=0, I5=0, X1=0, Y1=0, Z1=0, F2=PROBE_FEED.



Application

Note

- The sides must be parallel to the axes.
- The workpiece angle must be 90 degrees.
- The measured plane is perpendicular to the tool axis.

Direction of measurements

- The first measurement is always perpendicular to the principal axis.
- The second measurement is always perpendicular to the minor axis.



The support picture is in G17. The picture is not correct for a machine with exchanged axes (G18). Angle 1 must be replaced with 2, and 3 with 4.

Procedure

- 1 Rapid traverse to first starting point (X, Y, Z). If X, Y, Z are not programmed, the current position is used as the starting point.
- 2 First measurement with measurement feed (F2=) until the workpiece or the maximum measuring range (C1=) is reached.
- 3 Rapid traverse back to the first starting point. An error message is issued if the touch probe has not switched within the maximum measuring range (C1=).
- 4 Rapid traverse to the starting point of the 2nd measurement; depending on the value of I3=, the movement is performed at the safety clearance (L2=).
- 5 Second measurement (as described in points 2 and 3).
- 6 At the end, a rapid traverse to the safety clearance (L2=) is executed.
- 7 The measured value is saved in accordance with I5=.

Example

Aligning the inside corner of a workpiece

G1 X... Y... Z-5
G54 I3
G623 L2=20 B3=25 I3=1 I5=1 X1=-50 Y1=-50

G1	Position the touch probe 10 mm to the right of corner 1 and 8 mm from the front.
G54	Set zero point
G623	Define and execute measuring cycle. After the measuring cycle, the zero point shift is overwritten so that the coordinates of corner 1 are equal to X1= and Y1=

9.8 G626 Datum Outside Rectangle

Measurement of the center point of a paraxial rectangle.

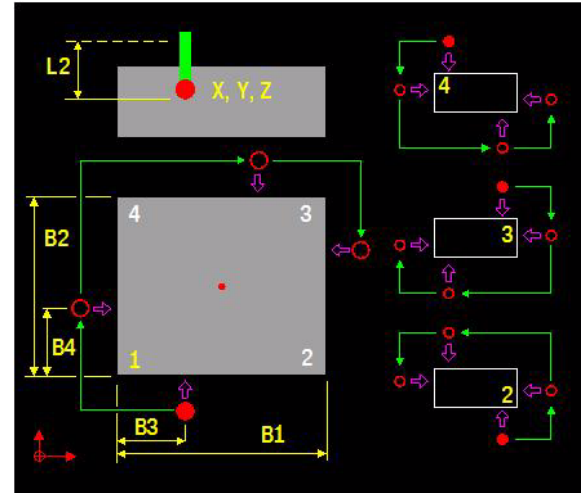
Address description

- ▶ **I4= corner number**
- ▶ **X,Y,Z starting point**
- ▶ **B1=,B2= side length**
- ▶ **B3=,B4= distance to corner** If B4= is not entered, B4=B3 is used.
- ▶ **C1= measuring distance**
- ▶ **L2= safety distance**
- ▶ **I2= probe orientat. -1=auto 0=no**
 - **I2=-1** Measure with automatic orientation. For an all-round transmitter, orientation is in the scanning direction. In the case of a two-layer touch probe, two measurements are performed with a 180° difference in orientation.
 - **I2=0** Measure without probe orientation.
- ▶ **I3= 2nd measurem. via L2 0=no 1=yes**
- ▶ **I5= G5x offset 0=no 1=X/Y/Z**
 - **I5=0** Do not save.
 - **I5=1** Save in the active zero point shift in the linear axes (X/Y/Z).
On saving, the measured values are added to the active zero point shift.
- ▶ **X1=, Y1=, Z1= target center point** When the measured coordinate is saved in the active zero point shift (I5>0), it is used to calculate the nominal value. The measured coordinate is assigned the target value for further programming.
- ▶ **O1=,O2= E par. meas. center**
- ▶ **O4=,O5= E par. meas. length**
- ▶ **F2= measuring feed**

For a description of the additional addresses, see "Explanation of addresses" on page 379.

Default setting

I4=1, B3=10, B4=B3, C1=20, L2=0, I2=-1, I3=0, I5=0, X1=0, Y1=0, Z1=0, F2=PROBE_FEED.



Application

Measurement


Two opposing tool corners are measured (1+3 or 2+4).

Direction of first corner measurement

- The first measurement is always perpendicular to the principal axis.
- The second measurement is always perpendicular to the minor axis.

Direction of second corner measurement

- Clockwise from corner number 1 ± 3 or 3 ± 1.
- Counter-clockwise from corner number 2 ± 4 or 4 ± 2.



The support picture is in G17. The picture is not correct for a machine with exchanged axes (G18). Angle 1 must be replaced with 2, and 3 with 4.

Procedure

- 1 Rapid traverse to the first starting point (X, Y, Z). If X, Y, Z are not programmed, the current position is used as the starting point.
- 2 First measurement with measurement feed (F2=) until the workpiece or the maximum measuring range (C1=) is reached.
- 3 Rapid traverse back to the first starting point. An error message is issued if the touch probe has not switched within the maximum measuring range (C1=).
- 4 Rapid traverse to the starting point of the 2nd measurement; depending on the value of I3=, the movement is performed at the safety clearance (L2=).
- 5 Second measurement (as described in points 2 and 3).
- 6 The opposite corner is measured by means of a 3rd and 4th measurement (as described in points 2 and 3).
- 7 At the end, a rapid traverse to the safety clearance (L2=) is executed.
- 8 The measured value is saved in accordance with I5=.

Example: Saving the center point of a rectangle in the zero point shift .

G54 I3

G626 X-45 Y-3 Z-5 B1=100 B2=20 B3=5 I3=1 I5=1

G54	Set zero point
G626	Define and execute measured cycle (B4=B3). Once the measuring cycle is complete, X and Y are recalculated in G54 I3.

9.9 G627 Datum Inside Rectangle

Measurement of the center point of a paraxial rectangular hole.

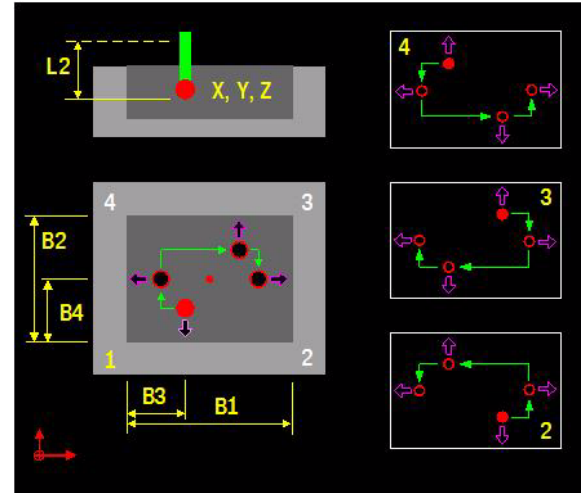
Address description

- ▶ **I4= corner number**
- ▶ **X,Y,Z starting point**
- ▶ **B1=,B2= side length**
- ▶ **B3=,B4= distance to corner** If B4= is not entered, B4=B3 is used.
- ▶ **C1= measuring distance**
- ▶ **L2= safety distance**
- ▶ **I2= probe orientat. -1=auto 0=no**
 - **I2=-1** Measure with automatic orientation. For an all-round transmitter, orientation is in the scanning direction. In the case of a two-layer touch probe, two measurements are performed with a 180° difference in orientation.
 - **I2=0** Measure without probe orientation.
- ▶ **I3= 2nd measurem. via L2 0=no 1=yes**
- ▶ **I5= G5x offset 0=no 1=X/Y/Z**
 - **I5=0** Do not save.
 - **I5=1** Save in the active zero point shift in the linear axes (X/Y/Z).
On saving, the measured values are added to the active zero point shift.
- ▶ **X1=, Y1=, Z1= target center point** When the measured coordinate is saved in the active zero point shift (I5>0), it is used to calculate the nominal value. The measured coordinate is assigned the target value for further programming.
- ▶ **O1=,O2= E par. meas. center**
- ▶ **O4=,O5= E par. meas. length**
- ▶ **F2= measuring feed**

For a description of the additional addresses, see "Explanation of addresses" on page 379.

Default setting

I4=1, B3=10, B4=B3, C1=20, L2=0, I2=-1, I3=0, I5=0, X1=0, Y1=0, Z1=0, F2=PROBE_FEED.



Application

Measurement

Two opposite corners of the workpiece are measured (1+3 or 2+4)

Direction of first corner measurement

- The first measurement is always perpendicular to the principal axis.
- The second measurement is always perpendicular to the minor axis.

Direction of second corner measurement

- Clockwise from corner number 1 ± 3 or 3 ± 1.
- Counter-clockwise from corner number 2 ± 4 or 4 ± 2.



The support picture is in G17. The picture is not correct for a machine with exchanged axes (G18). Angle 1 must be replaced with 2, and 3 with 4.

Procedure

- 1 Rapid traverse to the first starting point (X, Y, Z). If X, Y, Z are not programmed, the current position is used as the starting point.
- 2 First measurement with measurement feed (F2=) until the workpiece or the maximum measuring range (C1=) is reached.
- 3 Rapid traverse back to the first starting point. An error message is issued if the touch probe has not switched within the maximum measuring range (C1=).
- 4 Rapid traverse to the starting point of the 2nd measurement; depending on the value of I3=, the movement is performed at the safety clearance (L2=).
- 5 Second measurement (as described in points 2 and 3).
- 6 The opposite corner is measured by means of a 3rd and 4th measurement (as described in points 2 and 3).
- 7 At the end, a rapid traverse to the safety clearance (L2=) is executed.
- 8 The measured value is saved in accordance with I5=.

Example: Saving the center point of a rectangle in the zero point shift .

```
G54 I3
G627 X-45 Y-3 Z-5 B1=100 B2=20 B3=5 I3=1 I5=1
```

G54 Set zero point
 G627 Define and execute measured cycle (B4=B3). Once the measuring cycle is complete, X and Y are recalculated in G54 I3.

9.10 G628 Circle Measurement Outside

Measurement of the center point of a circle.

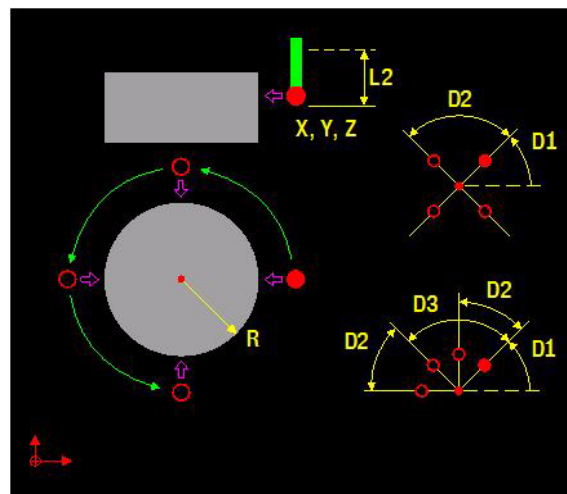
Address description

- ▶ **X,Y,Z starting point**
- ▶ **R circle radius**
- ▶ **D1= starting angle** Angle shift of the circle measurement, relative to the principal axis.
- ▶ **D2= second angle** Angle between first and second measurement and between third and fourth measurement. The smallest entry value is 5°.
- ▶ **D3= third angle** Angle between the first and third measurement. D3 must be at least 5° greater than D2. If D3 and D2 are identical, a 3-point measurement is performed.



The greatest accuracy is achieved with a symmetrical measurement with standard values D2=90 and D3=180.

- ▶ **C1= measuring distance**
- ▶ **L2= safety distance**
- ▶ **I2= probe orientat. -1=auto 0=no**
 - **I2=-1** Measure with automatic orientation. For an all-round transmitter, orientation is in the scanning direction. In the case of a two-layer touch probe, two measurements are performed with a 180° difference in orientation.
 - **I2=0** Measure without probe orientation.
- ▶ **I3= 2nd measurem. via L2 0=no 1=yes**
- ▶ **I5= G5x offset 0=no 1=X/Y/Z**
 - **I5=0** Do not save.
 - **I5=1** Save in the active zero point shift in the linear axes (X/Y/Z). On saving, the measured values are added to the active zero point shift.
- ▶ **X1=, Y1=, Z1= target center point** When the measured coordinate is saved in the active zero point shift (I5>0), it is used to calculate the nominal value. The measured coordinate is assigned the target value for further programming.
- ▶ **R1= minimum circle radius** The smallest permitted radius of the circle. The measured radius must be at least greater than or equal to R1, otherwise an error message is issued.



- **R1= maximum circle radius** The largest permitted radius of the circle. The measured radius must be at least smaller than or equal to R2, otherwise an error message is issued.
- **01=,02= E par. meas. center**
- **06= E par. measured diameter**
- **07= E par. radius difference** The difference between the measured radius and the programmed circular radius R is saved to an E parameter. The number of the E parameter must be entered. If no number is entered, nothing is saved.
- **F2= measuring feed**

For a description of the additional addresses, see "Explanation of addresses" on page 379.

Default setting

D1=0, D2=90, D3=180, C1=20, L2=0, I2=-1, I3=0, I5=0, X1=0, Y1=0, Z1=0, F2=PROBE_FEED.

Application

Starting point

The starting point of the circle measurement must be selected such that the first measurement moves as precisely as possible in the direction of the circle center.

Measuring direction

The circle measurement is executed counter-clockwise.

Procedure

- 1 Rapid traverse to the first starting point (X, Y, Z). If X, Y, Z are not programmed, the current position is used as the starting point.
- 2 First measurement with measurement feed (F2=) until the workpiece or the maximum measuring range (C1=) is reached.
- 3 Rapid traverse back to the first starting point. An error message is issued if the touch probe has not switched within the maximum measuring range (C1=).
- 4 Rapid traverse to the starting point of the 2nd measurement; depending on the value of I3=, the movement is performed at the safety clearance (L2=).
- 5 Second measurement (as described in points 2 and 4).
- 6 At the end, a rapid traverse to the safety clearance (L2=) is executed.
- 7 The measured value is saved in accordance with I5=.

Example

Saving the center point of a circular stud in the zero point shift

```
G54 I3
G628 X-45 Y-3 Z-5 R50 I3=1 I5=1
```

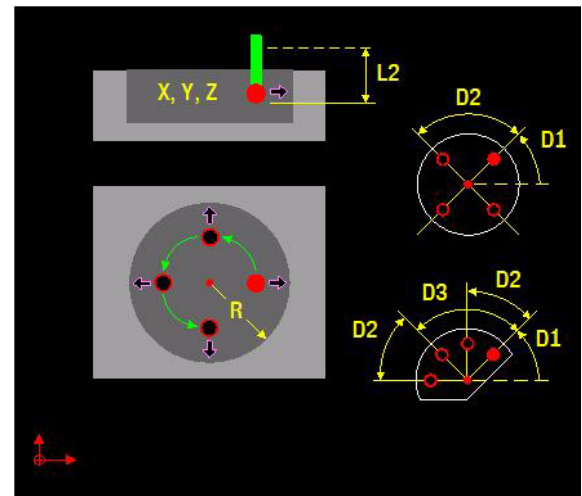
G54 Set zero point
G628 Define and execute measuring cycle.
 Once the measuring cycle is complete, X and Y are
 recalculated in G54 I3.

9.11 G629 Circle Measurement Inside

Measurement of the center point of a circular hole.

Address description

- ▶ **X,Y,Z starting point**
- ▶ **R circle radius**
- ▶ **D1= starting angle** Angle shift of the circle measurement, relative to the principal axis.
- ▶ **D2= second angle** Angle between first and second measurement and between third and fourth measurement. The smallest entry value is 5°.
- ▶ **D3= third angle** Angle between the first and third measurement. D3 must be at least 5° greater than D2. If D3 and D2 are identical, a 3-point measurement is performed.
- ▶ **C1= measuring distance**
- ▶ **L2= safety distance**
- ▶ **I2= probe orientat. -1=auto 0=no**
 - **I2=-1** Measure with automatic orientation. For an all-round transmitter, orientation is in the scanning direction. In the case of a two-layer touch probe, two measurements are performed with a 180° difference in orientation.
 - **I2=0** Measure without probe orientation.
- ▶ **I3= 2nd measurem. via L2 0=no 1=yes**
- ▶ **I5= G5x offset 0=no 1=X/Y/Z**
 - **I5=0** Do not save.
 - **I5=1** Save in the active zero point shift in the linear axes (X/Y/Z). On saving, the measured values are added to the active zero point shift.
- ▶ **X1=, Y1=, Z1= target center point** When the measured coordinate is saved in the active zero point shift (I5>0), it is used to calculate the nominal value. The measured coordinate is assigned the target value for further programming.
- ▶ **R1= minimum circle radius** The smallest permitted radius of the circle. The measured radius must be at least greater than or equal to R1, otherwise an error message is issued.
- ▶ **R1= maximum circle radius** The largest permitted radius of the circle. The measured radius must be at least smaller than or equal to R2, otherwise an error message is issued.
- ▶ **01=,02= E par. meas. center**
- ▶ **06= E par. measured diameter**



- **07= E par. radius difference** The difference between the measured radius and the programmed circular radius R is saved to an E parameter. The number of the E parameter must be entered. If no number is entered, nothing is saved.
- **F2= measuring feed**

For a description of the additional addresses, see "Explanation of addresses" on page 379.



The greatest accuracy is achieved with a symmetrical measurement with standard values D2=90 and D3=180.

Default setting

D1=0, D2=90, D3=180, C1=20, L2=10, I2=-1, I3=0, I5=0, X1=0, Y1=0, Z1=0, F2=PROBE_FEED.

Application

Starting point

The starting point of the circle measurement must be selected so that the first measurement moves as precisely as possible in the direction of the circle center.

Measuring direction

The circle measurement is executed counter-clockwise.

Procedure

- 1 Rapid traverse to first starting point (X, Y, Z). If X, Y, Z are not programmed, the current position is used as the starting point.
- 2 First measurement with measurement feed (F2=) until the workpiece or the maximum measuring range (C1=) is reached.
- 3 Rapid traverse back to the first starting point. An error message is issued if the touch probe has not switched within the maximum measuring range (C1=).
- 4 Rapid traverse to the starting point of the 2nd measurement; depending on the value of I3=, the movement is performed at the safety clearance (L2=).
- 5 Second measurement (as described in points 2 and 4).
- 6 At the end, a rapid traverse to the safety clearance (L2=) is executed.
- 7 The measured value is saved in accordance with I5=.

Example

Saving the center point of a circle in the zero point shift

G54 I3

G629 X-45 Y-3 Z-5 R50 I3=1 I5=1

G54	Set zero point
G629	Define and execute measuring cycle. Once the measuring cycle is complete, X and Y are recalculated in G54 I3.

9.12 G631 Measure Inclined Plane

Measurement of the inclination of a workpiece plane (G7) by means of a 3-point measurement.

Address description

- ▶ I1= meas.dir. $\pm 1/\pm 2/-3$ =main/minor/t1
- ▶ X,Y,Z starting point (meas. point 1)
- ▶ X1=Y1=,Z1= measuring point 2
- ▶ X2=,Y2=,Z2= measuring point 3
- ▶ O1= E par. for absolute spatial angle A5=
- ▶ O2= E par. for absolute spatial angle B5=
- ▶ O3= E par. for absolute spatial angle C5=
- ▶ C1= measuring distance
- ▶ L2= **safety distance** The safety distance is based on the starting point of each measurement and lies in the measuring direction.
- ▶ I3= 2nd measurem. via L2 0=no 1=yes
- ▶ F2= measuring feed

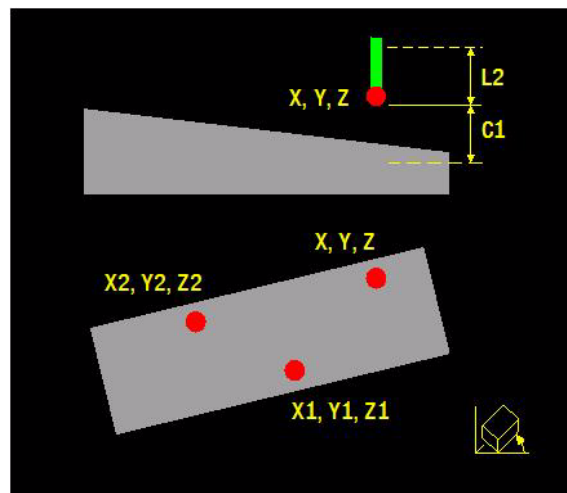
For a description of the additional addresses, see "Explanation of addresses" on page 379.

Default setting

C1=20, L2=0, I3=0, F2=PROBE_FEED.

Application

The measured inclination can be leveled with the G7 function.



Procedure

Rapid traverse movements are always performed with positioning logic in the active machining plane (which may already be tilted).

- 1 Rapid traverse to first starting point (X, Y, Z).
- 2 First measurement with measurement feed (F2=) until the workpiece or the maximum measuring range (C1=) is reached.
- 3 Rapid traverse back to starting point. An error message is issued if the touch probe has not switched within the maximum measuring range (C1=).
- 4 Movement to the starting point of the 2nd measurement; depending on the value of I3=, the movement is performed at the safety clearance (L2=).
- 5 Second and third measurement (as described in points 2 to 4).
- 6 At the end, a rapid traverse to the safety clearance (L2=) is executed.
- 7 The measured values are saved.

Example

Aligning and rotating the machining plane

G54 I3
G0 X50 Y20 Z100
G631 X18 Y0 Z-16 X1=18 Y1=10 Z1=-16 X2=10 Y2=0 Z2=-6 C1=15 L2=20 O1=10 O2=11 O3=12 F2=150
G0 Z100
G7 A5=E10 B5=E11 C5=E12 L1=1

G54	Set zero point.
G0	Move to the first position with the touch probe.
G631	Measure inclination of plane.
G0	Move to safe height (G17).
G7	Rotate machining plane.

9.13 G633 Angle Measurement 2 Holes

Measurement of the inclined position of a clamped workpiece.

The probe measures the centers points of two holes. MillPlus then calculates the angle between the principal axis of the working plane and the line connecting the center point of the hole.

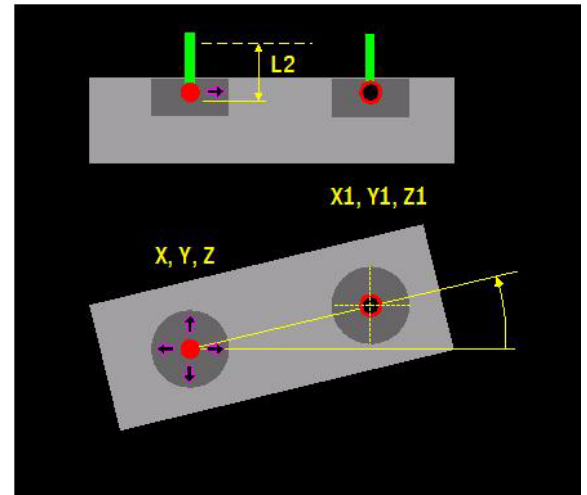
Address description

- ▶ **X, Y, Z starting point (meas. point 1)** Starting point for measuring the 1st hole (or current position).
- ▶ **X1=, Y1=, Z1= measuring point 2** Starting point for measuring the 2nd hole (all 3 coordinates must be entered).
- ▶ **C1= measuring distance**
- ▶ **L2= safety distance**
- ▶ **G5x offset 0=no 1=B4 2=A/B/C** Save measured values in a zero point shift. On saving, the measured values are added to the active zero point shift.
 - **I5=0** Do not save.
 - **I5=1** Save in the active zero point shift of the rotation angle (B4=).
 - **I5=2** Save in the active zero point shift in the rotary axis (A/B/C).
- ▶ **A1= target value angle** If the measured angle is saved in the active zero point shift (I5>0), it is used to calculate the target value. The measured position is thus given the target value for subsequent programming.
- ▶ **03= E par. measured angle** Number of the E parameter in which the angle is saved.
- ▶ **F2= measuring feed**

For a description of the additional addresses, see "Explanation of addresses" on page 379.

Default setting

C1=20, I5=0, A1=0, F2=PROBE_FEED.



Application

Starting position

The starting position must be programmed inside the hole.

Setting the zero point shift



If G7 is active, the measured angle cannot be set using G633 I5=2 in the zero point. Program G633 O3=.. and use the relevant E parameter in an incremental G7 shift, e.g. G7 C6=E10 L1=1.

Procedure

- 1 Rapid traverse to first starting point (X, Y, Z) in the 1st hole. If X, Y, Z are not programmed, the current position is used as the starting point.
- 2 Measurement with measurement feed (F2=), until the hole wall or the maximum measuring range (C1=) is reached. The center point is first measured roughly and then precisely.
- 3 Rapid traverse back to starting point. An error message is issued if the touch probe has not switched within the maximum measuring range (C1=). Retraction to the safety clearance (L2=).
- 4 Rapid traverse, over the safety clearance (L2=), to the starting point in the 2nd hole.
- 5 The hole is measured at the new position in the same way.
- 6 Steps 4 and 5 are repeated for the 3rd and 4th hole measurements.
- 7 At the end, a rapid traverse to the safety clearance (L2=) is executed.
- 8 The measured value is saved in accordance with I5=.

Example

Aligning a workpiece

G54 I3

G633 X-100 Y-50 Z-5 X1=10 Y1=-50 Z1=-5 L2=30 I5=2

G0 C0

G54	Set zero point.
G633	Define measuring cycle with starting point for 1st hole. Starting point for 2nd hole. Save safety distance=30 and measured value in zero point shift of rotary axis (C).
G0	Rotary table is positioned at zero (G17).

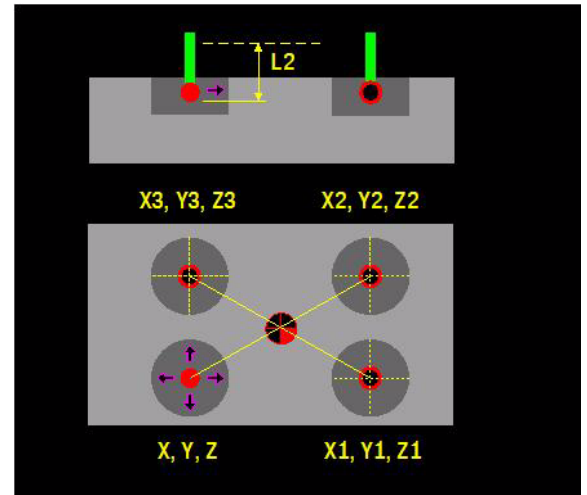
9.14 G634 Measurement Center 4 Holes

This probe cycle calculates the intersection point of two lines, each connecting two hole centers, and sets this intersection as a reference point. If desired, MillPlus can also enter the intersection point in a zero point table.

Address description

- ▶ **X, Y, Z starting point (meas. point 1)** Starting point for measuring the 1st hole (or current position).
- ▶ **X1=, Y1=, Z1= measuring point 2** Starting point for measuring the 2nd hole (all 3 coordinates must be entered).
- ▶ **X2=, Y2=, Z2= measuring point 3** Starting point for measuring the 3rd hole (all 3 coordinates must be entered).
- ▶ **X3=, Y3=, Z3= measuring point 4** Starting point for measuring the 4th hole (all 3 coordinates must be entered).
- ▶ **C1= measuring distance**
- ▶ **L2= safety distance**
- ▶ **I2= probe orientat. -1=auto 0=no**
 - **I2=-1** Measure with automatic orientation. For an all-round transmitter, orientation is in the scanning direction. In the case of a two-layer touch probe, two measurements are performed with a 180° difference in orientation.
 - **I2=0** Measure without probe orientation.
- ▶ **I5= G5x offset 0=no 1=X/Y/Z** Save measured values in a zero point shift. On saving, the measured values are added to the active zero point offset.
 - **I5=0** Do not save.
 - **I5=1** Save in the active zero point shift in the linear axes (X/Y/Z).
- ▶ **X4=, Y4=, Z4= target center point** When the measured coordinate is saved in the active zero point shift (I5>0), it is used to calculate the target value. The measured coordinate is assigned the target value for further programming.
- ▶ **01= E par. meas. center main axis** Number of the E parameter in which the measured center point of the principal axis is saved
- ▶ **02= E par. meas. center minor axis** Number of the E parameter in which the measured center point of the minor axis is saved
- ▶ **F2= measuring feed**

For a description of the additional addresses, see "Explanation of addresses" on page 379.



Default setting

C1=20, L2=-1, L5=0, F2=PROBE_FEED.

Application

Starting position

The starting position must be programmed inside the hole.

Procedure

- 1 Rapid traverse to first starting point (X, Y, Z) in the 1st hole. If X, Y, Z are not programmed, the current position is used as the starting point.
- 2 Measurement with measurement feed (F2=), until the hole wall or the maximum measuring range (C1=) is reached. The center point is first measured roughly and then precisely.
- 3 Rapid traverse back to starting point. An error message is issued if the touch probe has not switched within the maximum measuring range (C1=). Retraction to the safety clearance (L2=).
- 4 Rapid traverse, over the safety clearance (L2=), to the starting point in the 2nd hole.
- 5 The hole is measured at the new position in the same way.
- 6 At the end, a rapid traverse to the safety clearance (L2=) is executed.
- 7 The measured value is saved in accordance with L5=.

Example

Determining the center point of 4 holes in a workpiece

```
G54 I3
G634 X-10 Y-20 Z-5 X1=-100 Y1=-40 Z1=-5 X2=-100 Y2=-100
Z2=-5 X3=-10 Y3=-120 Z3=-5 L2=30 I5=1
```

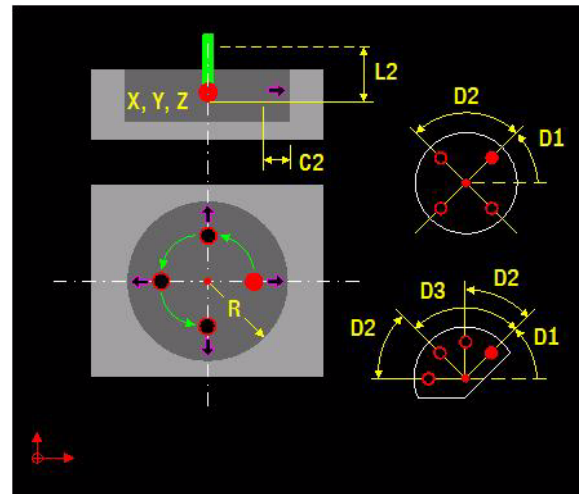
- G54 Set zero point.
- G634 Define measuring cycle with:
 - Starting point for 1st hole
 - Starting point for 2nd hole
 - Starting point for 3rd hole
 - Starting point for 4th hole
 - Safety clearance=30
 - Once the measuring cycle is complete, X and Y are recalculated in G54 I3.

9.15 G636 Circle Measurement Inside (CP)

Measurement of the center point of a hole.

Address description

- ▶ **R circle radius**
- ▶ **X, Y, Z circlecenter point** Theoretical center point of the circle to be measured.
- ▶ **D1= starting angle** Angle shift of the circle measurement, relative to the principal axis.
- ▶ **D2= second angle** Angle between first and second measurement and between third and fourth measurement. The smallest entry value is 5°.
- ▶ **D3= third angle** Angle between the first and third measurement. D3 must be at least 5° greater than D2. If D3 and D2 are identical, a 3-point measurement is performed.
- ▶ **C2= pre distance meas. point** The distance between the starting point of the measuring movement and the theoretical circle radius. The default is SAFETY_DIST.
- ▶ **L2= safety distance**
- ▶ **I2= probe orientat. -1=auto 0=no**
 - **I2=-1** Measure with automatic orientation. For an all-round transmitter, orientation is in the scanning direction. In the case of a two-layer touch probe, two measurements are performed with a 180° difference in orientation.
 - **I2=0** Measure without probe orientation.
- ▶ **I3= 2nd measurem. via L2 0=no 1=yes**
- ▶ **01= E par. meas. center main axis**
- ▶ **02= E par. meas. center minor axis**
- ▶ **06= E par. measured diameter**
- ▶ **07= E par. radius difference** The difference between the measured radius and the programmed circular radius R is saved to an E parameter. The number of the E parameter must be entered. If no number is entered, nothing is saved.
- ▶ **R1= minimum circle radius** The smallest permitted radius of the circle. The measured radius must be at least greater than or equal to R1, otherwise an error message is issued.
- ▶ **R2= maximum circle radius** The largest permitted radius of the circle. The measured radius must be at least smaller than or equal to R2, otherwise an error message is issued.



- **F2= measuring feed**
- **F5= feed circular movement** Feed of the circular movements between measurements. The default is RAPID_FEED.

For a description of the additional addresses, see "Explanation of addresses" on page 379.



The greatest accuracy is achieved with a symmetrical measurement with standard values D2=90 and D3=180.

Default setting

D1=0, D2=90, D3=180, C2=SAFETY_DIST, L2=0, I2=-1, I3=0, F2=PROBE_FEED, F5=RAPID_FEED

Application

Starting point

The starting point of the circle measurement must be selected such that the first measurement moves as precisely as possible in the direction of the circle center.

The starting point of the measuring movement is determined from the circle center point, the pre-measurement distance, and the starting angle. The measuring cycle is executed from here. If not all coordinates of the center point are entered, the current position of the touch probe is used.

Measuring direction

The circle measurement is executed counter-clockwise.

Procedure

- 1 Rapid traverse to the starting point calculated from X, Y, Z, R, and C2. If X, Y, Z are not programmed, the current position is used as the starting point.
- 2 First measurement with measurement feed (F2=), until the workpiece or the maximum measuring range (C2+MEAS_RANGE) is reached.
- 3 Rapid traverse back to the first starting point. An error message is issued if the touch probe has not switched within the maximum measuring range (C2+MEAS_RANGE).
- 4 Rapid traverse to the starting point of the 2nd measurement; depending on the value of I3=, the movement is performed at the safety clearance (L2=) or with a circular movement.
- 5 Second measurement (as described in points 2 and 4).
- 6 At the end, a rapid traverse to the safety clearance (L2=) is executed.

Example: Saving the center point and diameter of a circle in an E parameter

```
G636 X-45 Y-3 Z-5 R5 O1=1 O2=2 O6=3 O7=4
```

G636 Define and execute measuring cycle. Once the measuring cycle is complete, E-parameters E1, E2, E3, and E4 are recalculated.

9.16 G638 Touch Probe Calibration on Ball

Calibration of length, orientation angle, radius and, oriented radius of a touch probe using a ball.

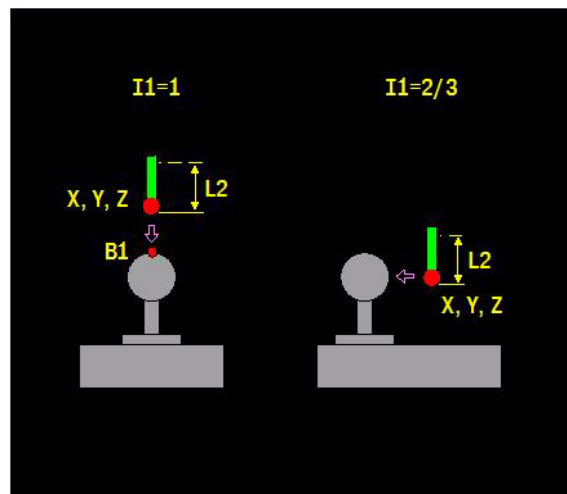
Address description

- ▶ **I1=** 1=L 2=R 3=CAL_ANG + R
- ▶ **X,Y,Z** starting point
- ▶ **C1=** measuring distance
- ▶ **B1= target position** If I1 = 1, the measured coordinate is compared with the target position. The difference is offset in the new probe length.
- ▶ **R ball radius** If I1 = 2 or 3, the ball radius must be entered.
- ▶ **L2=** safety distance
- ▶ **01=** E par. L
- ▶ **02=** E par. R
- ▶ **03=** E par. CAL_ANG

A description of the other addresses is provided under "Introduction to measuring cycles".

Default setting

C1=20, L2=0.



Application

General information

The touch probe must be calibrated when:

- Being used for the first time
- The touch probe pin is replaced
- The touch probe pin is bent

Calibrating the touch probe length

To calibrate the touch probe length, a target position must be entered for Address B1. The new touch probe length is saved under address L in the tool table.

Calibrating the touch probe radius

When a calibration ring is calculated, the center touch probe radius R is determined and automatically saved to the tool table. If the touch probe has an all-round emitter, the oriented probe radius is also saved to address R.

Calibrating the probe orientation angle

The orientation angle is determined by 120 measurements and is automatically saved in CAL_ANG. Available only if the touch probe has an all-round transmitter.

Procedure for calibrating the touch probe length (I1=1)

- 1 Rapid traverse to starting point (X, Y, Z). If X, Y, Z are not programmed, the current position is used as the starting point.
- 2 Measurement in tool axis until the ball or maximum measuring distance (C1=) is reached.
- 3 Rapid traverse back to starting point. An error message is issued if the touch probe has not switched within the maximum measuring range (C1=).
- 4 At the end, a rapid traverse to the safety clearance is executed (L2=).

Procedure for calibrating the probe radius (I1=2)

- 1 Rapid traverse to starting point (X, Y, Z). If X, Y, Z are not programmed, the current position is used as the starting point.
- 2 Rough measurement of center point. An error message is issued if the touch probe has not switched within the maximum measuring range (C1=).
- 3 Precise measurement of center point.
- 4 Only in the case of an all-round transmitter: oriented measurement to determine R
- 5 Non-oriented measurement to determine R.
- 6 At the end, a rapid traverse to the safety clearance is executed (L2=).

Procedure for calibrating the orientation angle and touch probe radius (I1=3)

- 1 Rapid traverse to starting point (X, Y, Z). If X, Y, Z are not programmed, the current position is used as the starting point.
- 2 Rough measurement of center point. An error message is issued if the touch probe has not switched within the maximum measuring range (C1=).
- 3 Precise measurement of center point.
- 4 120 measurements to measure the orientation angle.
- 5 Only in the case of an all-round transmitter: oriented measurement to determine R
- 6 Non-oriented measurement to determine R.
- 7 At the end, a rapid traverse to the safety clearance (L2=) is executed.

Example

Calibrating the touch probe orientation angle and probe radius

```
G54 X0 Y0 Z0
G638 R10 I1=3 X-45 Y-3 Z342.651 C1=20
```

G54 Delete zero point shift
G638 Calibrate orientation angle (CAL_ANG) and touch probe radius (R). CAL_ANG and R are automatically recalculated.

9.17 G639 Touch Probe Calibration

Calibration of the length, orientation angle, radius, and oriented radius of a touch probe.

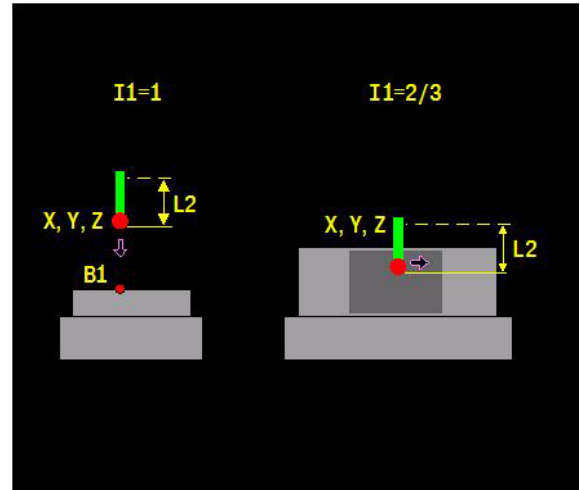
Address description

- ▶ **I1=** 1=L 2=R 3=CAL_ANG + R
- ▶ **X,Y,Z** starting point
- ▶ **C1=** measuring distance
- ▶ **B1= target position** If I1= 1, the measured coordinate is compared with the target position. The difference is offset in the new probe length.
- ▶ **R ball radius** If I1= 2 or 3, the ball radius must be entered.
- ▶ **L2=** safety distance
- ▶ **01=** E par. L
- ▶ **02=** E par. R
- ▶ **03=** E par. CAL_ANG

A description of the other addresses is provided under "Introduction to measuring cycles".

Default setting

C1=20, L2=0.



Application

General information

The touch probe must be calibrated when:

- Being used for the first time
- The touch probe pin has been replaced
- The touch probe pin is bent

Calibrating the touch probe length

To calibrate the touch probe length, a target position must be entered for Address B1. The new touch probe length is saved under address L in the tool table.

Calibrating the touch probe radius

When a calibration ring is calculated, the center touch probe radius R is determined and automatically saved to the tool table. If the touch probe has an all-round emitter, the oriented probe radius is also saved to address R.

Calibrating the probe orientation angle

The orientation angle is determined by 120 measurements and is automatically saved in CAL_ANG. Available only if the touch probe has an all-round transmitter.

Procedure for calibrating the touch probe length (I1=1)

- 1 Rapid traverse to starting point (X, Y, Z). If X, Y, Z are not programmed, the current position is used as the starting point.
- 2 Measurement in the tool axis until the table (or measured block) or the maximum measuring distance (C1=) is reached.
- 3 Rapid traverse back to starting point. An error message is issued if the touch probe has not switched within the maximum measuring range (C1=).
- 4 At the end, a rapid traverse to the safety clearance is executed (L2=).

Procedure for calibrating the probe radius (I1=2)

- 1 Rapid traverse to starting point (X, Y, Z) in calibration ring. If X, Y, Z are not programmed, the current position is used as the starting point.
- 2 Rough measurement of center point. An error message is issued if the touch probe has not switched within the maximum measuring range (C1=).
- 3 Precise measurement of center point.
- 4 Only in the case of an all-round transmitter: oriented measurement to determine R
- 5 Non-oriented measurement to determine R.
- 6 At the end, a rapid traverse to the safety clearance is executed (L2=).

Procedure for calibrating the orientation angle and touch probe radius (I1=3)

- 1 Rapid traverse to starting point (X, Y, Z). If X, Y, Z are not programmed, the current position is used as the starting point.
- 2 Rough measurement of center point. An error message is issued if the touch probe has not switched within the maximum measuring range (C1=).
- 3 Precise measurement of center point.
- 4 120 measurements to measure the orientation angle.
- 5 Only in the case of an all-round transmitter: oriented measurement to determine R
- 6 Non-oriented measurement to determine R.
- 7 At the end, a rapid traverse to the safety clearance (L2=) is executed.

Example

Calibrating the touch probe length

G54 X0 Y0 Z0

G639 I1=1 X-45 Y-3 Z342.651 C1=20 B1=309.769

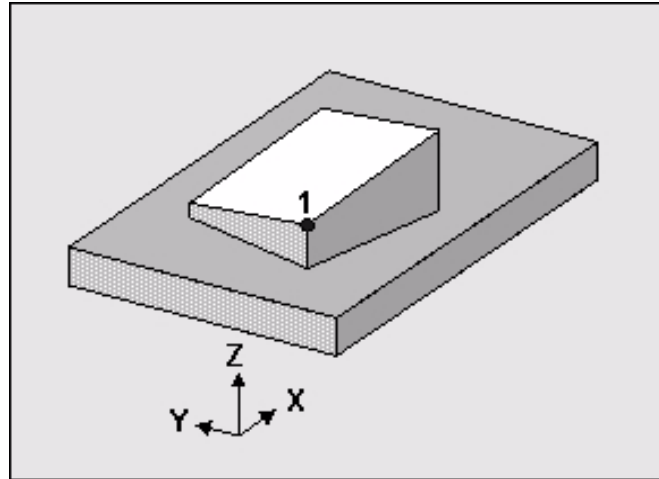
G54	Delete zero point shift
G639	Calibrate touch probe length (L). Address L in the tool table is automatically recalculated.

Example: setting a reference point in the corner and top edge of an inclined, rectangular surface

In this example, a reference point is set in the corner and at the top edge of an inclined, rectangular surface by means of several measuring cycles. The measured angles are used in a G7 shift. The measured positions, however, are used in a zero point shift.

Procedure:

- Measure inclination of plane.
- Position plane perpendicular to the tool.
- Measure angle between rectangle and X axis.
- Move X axis parallel to rectangle.
- Measure corner 1 of right angle (see fig.).
- Measure upper edge of workpiece.



G17	Activate the G17 plane.
G54 I1 X0 Y0 Z0 B0 C0 B4=0	Reset and activate zero point.
T1 M6	Call tool.
G631 I1=-3 X100 Y5 Z1 X1=165 Y1=5 Z1=15 X2=100 Y2=45 Z2=3 O1=10 O2=11 O3=12	Measure inclination of plane and save absolute spatial angles A5=, B5=, and C5= in E10, E11, and E12.
G0 X100 Y5 Z100	Position in rapid traverse.
G7 A5=E10 B5=E11 C5=E12 L1=1	Position plane perpendicular to the tool.
G620 I1=2 X0 Y0 Z10 B1=20 B2=5 C1=10 L2=100 I5=0 O3=14 F2=150	Measure the angle between the long side of the rectangle and the X axis and save in E14.
G7 C6=E14 L1=1	Move X axis parallel to the long side of the rectangle.
G622 I4=1 X12 Y1 Z18 B3=20 C1=10 L2=100 I5=1 O1=16 O2=17 F2=150	Measure corner 1 of the rectangle, set the corner in the zero point, and save in E16 and R17.
G621 I1=-3 X10 Y10 Z22 C1=10 L2=100 I5=1 O1=18 F2=150	Measure the top edge of the workpiece, set it in the zero point, and save in E18.
M30	End of the program.

10

**G700-G799 Milling
Cycles**

10.1 Machining and Positioning Cycles

The machining cycle defines a machining process. A separate positioning cycle defines the execution of the machining cycle at a specific position.

Overview of machining and positioning cycles

Special Cycle

1	G700	Face turning (only in DIN/ISO)
2	G730	Multipass milling
3	G740	Thread milling inside
4	G741	Thread milling outside

Positioning cycles (patterns) (only in MDI)

1	G771	Operation on line
2	G772	Operation on quadrangle
3	G773	Operation on grid
4	G777	Operation on circle, enhancement of G77
5	G779	Operation at position, enhancement of G79

Drilling cycles

1	G781	Drilling/centring enhancement of G81
2	G782	Deep-hole drilling, enhancement of G83
3	G783	Deep-hole drill. add. chip break. enhancement of G83 (only in DIN/ISO)
4	G784	Tapping, enhancement of G84 (only in MDI)
5	G785	Reaming, enhancement of G85
6	G786	Boring, enhancement of G86
7	G790	Back-boring
8	G794	Tapping, interpolated, enhancement of G84 (only in MDI)

Milling cycles

1	G787	Pocket milling, enhancement of G87
2	G788	Key-way milling, enhancement of G88
3	G789	Circular pocket milling, enhancement of G89
4	G797	Pocket finishing
5	G798	Key-way finishing
6	G799	Circular pocket finishing

Introduction

Machining plane

Cycle programming is independent of the machining plane (G17, G18, G19, and G7).

Tool axis and machining plane

The cycles are executed in the current main plane G17, G18, G19 or in the inclined plane G7. The working direction of the cycle is determined by the tool axis. The direction of the tool axis can be reversed with G67.

Procedure in DIN

The new machining cycles (special, drilling, and milling cycles) are only executed by positioning cycle G79 at one position. Points (P1-P4) are not allowed.

Positioning logic

In rapid traverse and, depending on G28, using positioning logic, the tool moves to the 1st setup clearance via the position (X, Y, Z,) defined by the positioning cycle.

Mirroring and scaling

Mirroring and scaling must not be activated between a drilling/milling cycle and a positioning cycle.

Deleting cycle data

Cycle data is deleted by M30, the CANCEL PROGRAM softkey, the CNC RESET softkey, or by defining a new cycle.

Spindle activation

The spindle must be switched on for the cycle start. F and S can be overwritten in the cycle definition.

Mirroring

If you are only mirroring one axis, the direction of rotation of the tool changes. This does not apply during machining cycles.

Comments

Comments are not allowed in a block with a machining cycle. Before calling up the cycle, you must program radius compensation G40.



Pre-position the tool so that there can be no collision with the workpiece or clamping devices.

Explanation of addresses

The addresses described here are used in most cycles. Specific addresses are described in the relevant cycle.

- ▶ **X, Y, Z position of the defined machining geometry** Machining is executed at this position. If X, Y or Z is not entered, the current tool position is used.
 - The tool moves to the starting point in rapid traverse and, depending on G28, using positioning logic. If X, Y, Z are not programmed, the current position is used as the starting point. The first setup clearance (L1=) is taken into account in the tool axis. With multipass milling (G730), the other axes are also displaced.
- ▶ **L depth (greater than 0)** With multipass milling (G730), this is the machining height: distance between programmed workpiece surface and blank surface.
- ▶ **R radius of circular pocket**
- ▶ **L1= 1st setup clearance at start of cycle**
- ▶ **L2= 2nd setup clearance** Height above the 1st setup clearance. At the end of the cycle, the tool moves to the 2nd setup clearance (if entered).
- ▶ **C1= cutting/plunging depth (> 0)** Dimension by which the tool plunges in each infeed. The depth (L) or machining depth (L) does not have to be a multiple of the feed depth (C1=). The CNC moves to the depth in one work pass if the feed depth is the same as or greater than the depth (C1=>L-L3).
 - If a feed depth (C1=) is programmed for milling or machining operations, this usually results in a residual cut that is smaller than the programmed feed depth. For drilling, the last 2 cuts are distributed equally if the residual cut > 0. This avoids a very small last cut.
- ▶ **D3= dwell** Number of revolutions for which the tool stays at the bottom of the hole for free cutting. (Minimum is 0 and maximum is 9.9.)
- ▶ **F2= rapid plunging** Traversing speed of tool when moving from the setup clearance to the milling depth.
- ▶ **F5= rapid retraction** Traversing speed of tool when retracting from the hole.
- ▶ **F and S feed and spindle speed** The addresses F and S are not available in machining cycles within MDI. They must be programmed in the FST menu.

10.2 G700 Face Turning

The face turning cycle executes a single flat or conical turning operation.

Address description

- ▶ **X** radius
- ▶ **F2=** feed [mm/rev|inch/rev]
- ▶ **L** tool axis displacement
- ▶ **I1=** uncouple 0=no 1=yes
- ▶ **S** speed

The following addresses in the tool memory are used by the cycle:

- ▶ **R** **adjustment radius** Is automatically overwritten with the current radius after face turning.
- ▶ **A1=** **orientation angle for engaging** Is automatically overwritten with the current angle (0-359.999 degrees) after face turning.
- ▶ **R1=** **minimum diameter (optional)**
- ▶ **R2=** **maximum diameter (optional)**

For a description of the additional addresses, see "Explanation of addresses" on page 420.

Default setting

L0, I1=0

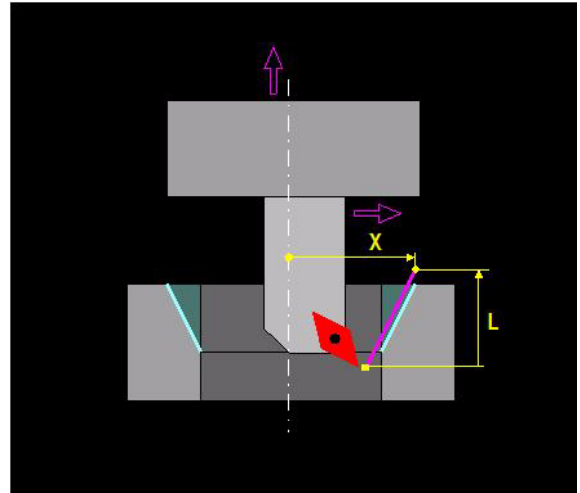
Application

G700 must not be programmed if:

- G36 and/or G182 are active.
- Tool T0 is programmed.
- Spindle orientation at an angle must not be zero.

Resetting the radial facing slide

The maximum permitted speed can be used to reset the radial facing slide to the starting diameter.



Actual diameter reached

The programmed diameter is rounded so that it exactly matches one of the 72 indexing positions of the clamp. The maximum deviation that this causes is $< (\text{feed}/72)/2$, i.e. a 0.001 mm deviation for a feed rate of 0.15 mm/rev.

Comment

G40, G72, G90, and G94 remain active after G700.

Mid-program startup

In the case of mid-program start-up, the head must be in the correct position before the start of a G700 cycle. Therefore, the radius R and angle A1 must be correctly entered in the tool table.

Speed and feed correction switch

The speed correction switch is not active. The feed correction switch is active.

Display

During tool movement, the speed is displayed in the current S field. At the end, the spindle position is always displayed in the range of 0-359.999 degrees.

The programmed feed remains unchanged. The current feed displays the value zero or the feed of the traverse path in the tool axis.

The indexing of inward and outward movements is automatically performed by the cycle.

M82 indexing of outward movement (in facing head). M80 indexing of inward movement

Facing head

The facing head can be used as a boring head after being inserted into the spindle. The bracket is fixed by the indexing device built into the machine and, at the same time, the locking device between the bracket and facing head is released. When the spindle rotates, the radial facing slide is moved by a mechanical gearing of e.g. 0.1 mm/ causes. The traversed path is determined by the rotary speed of the spindle. Synchronized movement of the spindle and tool axis (Z) enables cones and chamfers to be turned. The spindle is rotated counter-clockwise to reset.

Procedure

- 1 Set facing head adjustment radius and enter it into the tool memory.
- 2 Insert the facing head into the spindle (check the engagement angle at first insertion).
- 3 Check the orientation and indexing and retract if necessary.
- 4 Spindle turns, thus executing a facing operation.
- 5 Tool moves to angle positions in multiples of 5 degrees.
- 6 The adjustment radius and angle of orientation are automatically written to the tool memory.

Example

Face turning

G700 X50 L5 F=0.05 S600

G700 X70

G0 Z100

G700 X40 I1=1 S1200

	Tool memory: tool radius R20
	Tool memory: orientation angle A1=0
G700	Chamfer 5 mm from diameter 40 to 50
G700	Facing movement at diameter 70
G0	Lift
G700	Reset to diameter 40 and disengage

10.3 G730 Multipass Milling

Definition of a multipass milling cycle in a single program block.

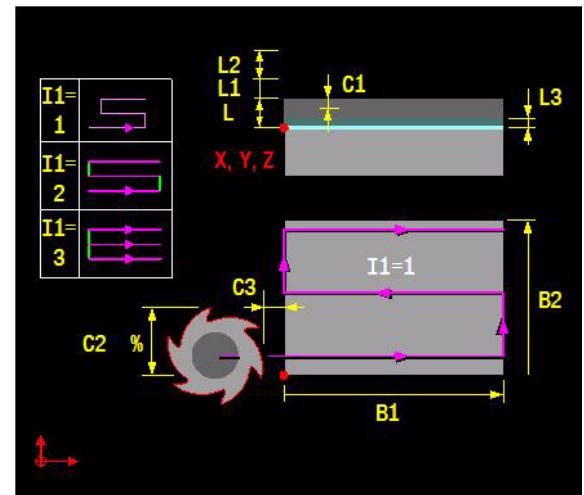
Address description

- ▶ **B1= B2= side length**
- ▶ **L height** Machining height >0
- ▶ **L1=, L2= setup clearance**
- ▶ **L3= finishing allowance**
- ▶ **C1= plunging depth**
- ▶ **C2= proportional cutting width** Maximum percentage of the tool diameter to be used as the cutting width on each pass. The total width is divided into equal sections. The last cut goes 10% of the mill diameter over the edge of the material.
- ▶ **C3= radial setup clearance**
- ▶ **I1= 1=meander 2=M.+rapid 3=parallel** Method:
 - I1=1 meander
 - I1=2 meander and transverse movement outside material
 - I1=3 machining in same direction. The directions of B1= and B2= determine whether climb or up-cut milling is used.
- ▶ **F2= rapid for plunging**

For a description of the additional addresses, see "Explanation of addresses" on page 420.

Default setting

L1=1, L2=0, L3=0, C1=L-L3, C2=67%, C3=5, I1=1



Procedure

Method: meander

- 1 Rapid traverse to the 1st setup clearance above workpiece surface. The starting point is the radius of the tool plus the radial setup clearance (C3=) in addition to the programmed position.
- 2 Rapid plunging movement (F2=) by the feed depth (C1=) to the next depth.
- 3 The tool then mills one pass in the principal axis. The end point of this movement is by the cutting width (C2= maximum 50% of the milling radius) in the material. In the last cut, the tool travels outside the material by the amount of the radial setup clearance.
- 4 The tool moves to the starting point of the next pass with transverse milling feed. In the last pass, it moves by 10% of the milling cutter radius outside the material.
- 5 Repeat steps 3 and 4 until the specified surface has been machined in full.
- 6 Repeat steps 1 to 6 until the depth (L) has been reached.
- 7 At the end, a rapid traverse movement to the 1st plus 2nd setup clearance (L1= plus L2=) is performed.

Method: meander and transverse movement outside of material

- 1 With this method, the end point of each pass is located outside the material by the amount of the radial setup clearance. The tool executes the transverse movement in rapid traverse.

Method: milling in same direction.

- 1 With this method, the tool mills in the same direction on each pass (climb or up-cut milling). The end point of each pass is outside the material by the amount of the radial setup clearance. The CNC retracts the tool by the 1st setup clearance (L1=) at the end of a pass. The tool then moves back along the principal axis in rapid traverse before executing the rapid transverse movement.

Example

Multipass milling

```
G730 I1=2 B1=100 B2=80 L10 L1=5 C1=3 C2=73 C3=1 F100
G79 X-50 Y-50 Z0
```

G730	Define multipass milling cycle
G79	Execute multipass milling cycle

10.4 G740 Thread Milling Inside

Internal thread milling with a thread mill

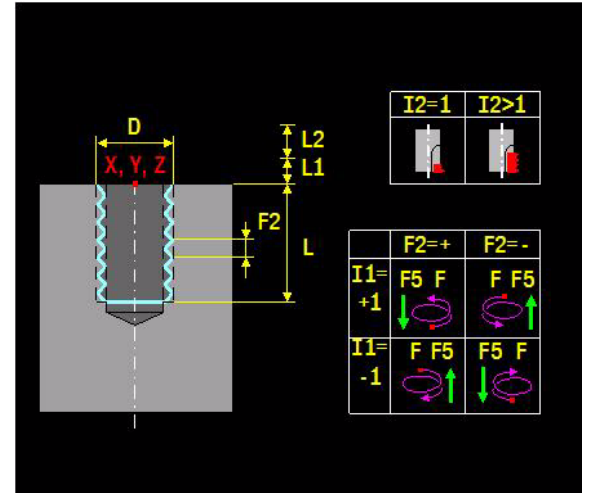
Address description

- ▶ **D diameter** Nominal thread diameter.
- ▶ **F2= pitch, +/-=thread direction** The sign determines the thread pitch: right thread (+) and left thread (-). Range: +/- 99.9999 mm.
- ▶ **L= depth** Distance between tool surface and thread base.
- ▶ **I2= number of threads per step** Number of thread ridges per tool. In between, the tool is shifted by I2 times the pitch.
 - I2=1 one ridge. Continuous helical path over the entire length of the thread
 - I2>1 multiple ridges. Multiple helical paths with approach and departure.
- ▶ **L1= 1st setup clearance** 1 distance between the tool tip and tool surface.
- ▶ **L2= 2nd setup clearance** 2 distance in tool direction whereby no collision can occur between the tool and clamp.
- ▶ **I1= milling 1=climb -1=conventional** Type of milling:
 - +1 = climb milling
 - -1 = up-cut milling
- ▶ **F5= plunge/retract rapid** Maximum speed during infeed or retraction. Can be influenced by rapid traverse override.

For a description of the additional addresses, see "Explanation of addresses" on page 420.

Default setting

I1=1, L1=F2, L2=0, F5=F



Application

Tool for thread-milling

The thread-milling tool requires a specific compensation value, which is specified in the catalog of the tool manufacturer. This value must be entered in the allowance radius (R4=) in the tool table.

Note that the tool moves beyond the programmed depth during tangential approach or departure, and a **collision** can occur in the case of insufficient clearance.

Tangential infeed and retraction with G740 and G741 is calculated as follows. Tangential infeed and retraction is executed with a semicircle where radius = pitch. Lead cut/overflow = $F2 * F2 / 2 * \text{helix diameter (helix diameter thread diameter / 2 - tool diameter)}$. The helix radius is usually smaller than the pitch, in which case the overflow is less than half of the pitch.

Mill machining starts in the tool axis at the starting point or at the thread base. This direction is determined by the pitch direction (F2=+/-) and mill direction (I1=).

For tools rotating right, the relationship between the entry parameters is as follows::

Internal thread	Pitch (F2=)	Mill direction (I1) +1 climb, -1 up-cut	Working direction of tool axis
	+ right thread	I1=+1	Z+
	+ right thread	I1=-1	Z-
	- left thread	I1=+1	Z-
	- left thread	I1=-1	Z+
External thread	Pitch (F2=)	Mill direction (I1) +1 climb, -1 up-cut	Working direction of tool axis
	+ right thread	I1=+1	Z-
	+ right thread	I1=-1	Z+
	- left thread	I1=+1	Z+
	- left thread	I1=-1	Z-

Feed

Normally, the feed is based on the tool center. In this case, the feed is based on the tool radius (see: F1=, constant cut feed with radius compensation of circles).

Procedure

- 1 The thread mill is positioned at the setup clearance above the tool surface in rapid traverse.
- 2 The thread mill moves to the starting position in rapid traverse. This position is determined by the thread pitch (F2=), the running direction (I1=), and the number of thread cuts per step (I2=).
- 3 The mill executes a compensation movement to assume the correct starting position. The mill then moves tangentially to the thread radius in a helix.
- 4 Depending on the entry parameter "Number of thread cuts per step" (I2=), the tool mills the thread in one or more cuts or in a continuous helix movement.
- 5 At the end, the mill moves tangentially away from the tool in a helix. The mill then returns to the starting position at an increased feed rate.
- 6 At the end of the cycle, the tool returns to the 1st and, if programmed, 2nd safety clearance in rapid traverse.

Note

By default, the mill direction is from bottom to top (see example). Depending on the parameters I1=/F2, the mill direction can also be from top to bottom.

Example

Internal thread milling

T2 M6

S800 F120 M3

G740 D=60 F2=5,5 L16 I2=1 F5=1500 I1=1 L1=5 F=200

G79 X0 Y0 Z0

10.5 G741 Thread Milling Outside

External thread milling with a thread mill

Address description

- ▶ **D diameter** Nominal thread diameter.
- ▶ **F2= pitch, +/-=thread direction** The sign determines the thread pitch: right thread (+) and left thread (-). Range: +/- 99.9999 mm.
- ▶ **L depth** Distance between tool surface and thread base.
- ▶ **I2= number of threads per step** Number of thread ridges per tool
 - I2=1 one ridge. Continuous helical path over the entire thread length
 - I2>1 multiple ridges. Multiple helical paths with approach and departure. In between, the tool is shifted by I2 times the pitch.
- ▶ **L1= 1st setup clearance** 1 distance between the tool tip and tool surface.
- ▶ **L2= 2nd setup clearance** 2 distance in tool direction whereby no collision can occur between the tool and clamp.
- ▶ **I1= milling 1=climb -1=conventional** Type of milling:
 - +1 = climb milling
 - -1 = up-cut milling
- ▶ **F5= plunge/retract rapid** Maximum speed during infeed or retraction. Can be influenced by rapid traverse override.

For a description of the additional addresses, see "Explanation of addresses" on page 420.

Default setting

I1=1, L1=F2, L2=0, F5=F

Example

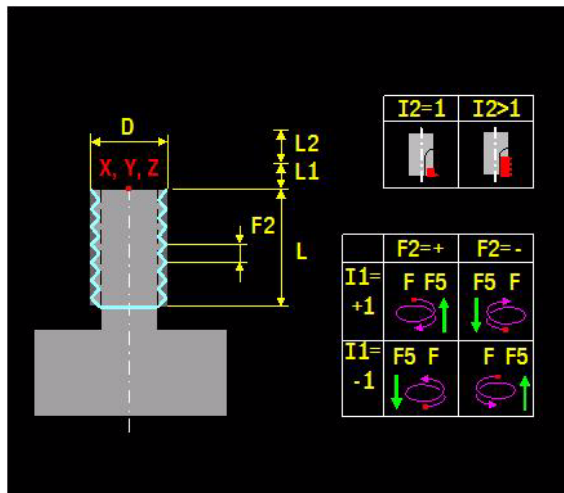
External thread milling

T2 M6

S800 F120 M3

G740 D=60 F2=5,5 L16 I2=1 F5=1500 I1=1 L1=5 F=200

G79 X0 Y0 Z0



10.6 G771 Operation on Line

Execution of a machining cycle at points located at a fixed equal distance on a line.

Address description

- ▶ B1= spacing
- ▶ K1= number of operations
- ▶ X, Y, Z position
- ▶ P1= point definition number
- ▶ A1= angle
- ▶ A5= angle of rotation

For a description of the additional addresses, see "Explanation of addresses" on page 420.

Default setting

A1=0, A2=90, A5=0.

Application

Machining position

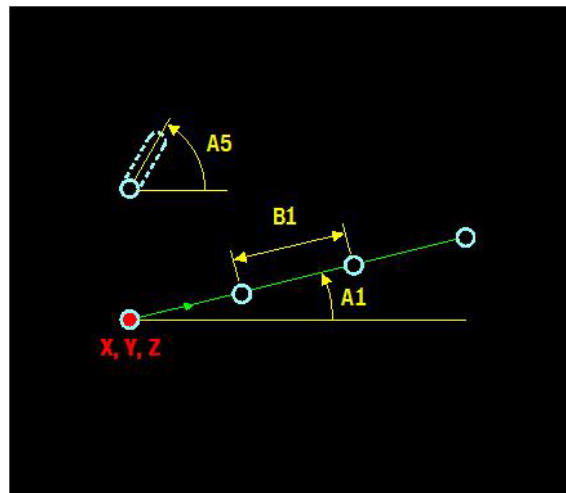
The machining position is defined via X,Y,Z or point definition number P1=.

Pocket angle

The pocket angle is defined by A5.

Procedure

- 1 Rapid traverse to position.
- 2 The machine cycle previously defined is executed at this position.
- 3 The next position is approached after execution.
- 4 Repeat procedure (2-3) until all positions (K1=) have been machined.



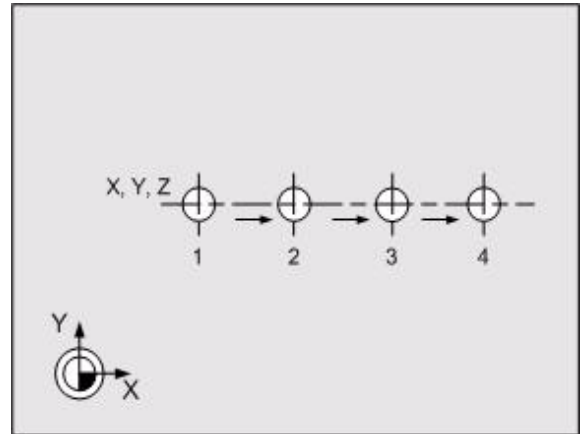
Example

G781 L30 F100 F5=6000

G771 X50 Y20 Z0 B1=40 K1=4

G781 Define bore cycle

G771 Execute bore cycle at 4 positions



10.7 G772 Operation on Quadrangle

Execution of a machining cycle at points located at fixed equal distances on a quadrangle.

Address description

- ▶ B1= longitudinal spacing
- ▶ K1= number of longitudinal operations
- ▶ B2= transverse spacing
- ▶ K2= number of transverse operations
- ▶ X, Y, Z position
- ▶ P1= point definition number
- ▶ A1= starting angle
- ▶ A2= ending angle
- ▶ A5= angle of rotation
- ▶ F feed

For a description of the additional addresses, see "Explanation of addresses" on page 420.

Default setting

A1=0, A2=90, A5=0.

Application

Machining position

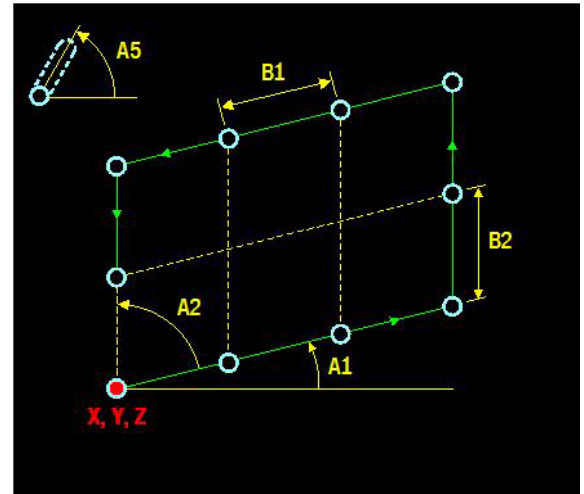
The machining position is defined via X,Y,Z or point definition number P1=.

Pocket angle

The pocket angle is defined by A5.

Procedure

- 1 Rapid traverse to position.
- 2 The machine cycle previously defined is executed at this position.
- 3 The next position is approached after execution. The direction of the rectangle is determined by angle A1=.
- 4 Repeat procedure (2-3) until all positions (K1=, K2=) have been machined.



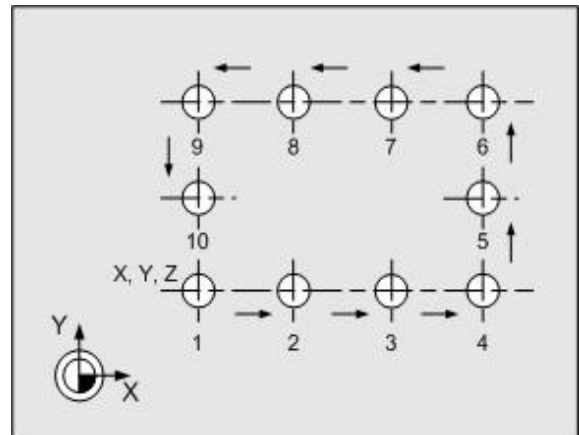
Example

G781 L30 F100 F5=6000

G772 X50 Y20 Z0 B1=40 K1=4 B2=30 K2=3

G781 Define bore cycle

G772 Execute bore cycle on rectangle with 10 positions



10.8 G773 Operation on Grid

Execution of a machining cycle at points located at fixed equal distances on a grid.

Address description

- ▶ B1= longitudinal spacing
- ▶ K1= number of longitudinal operations
- ▶ B2= transverse spacing
- ▶ K2= number of transverse operations
- ▶ X, Y, Z position
- ▶ P1= point definition number
- ▶ A1= starting angle
- ▶ A2= ending angle
- ▶ A5= angle of rotation
- ▶ F feed

For a description of the additional addresses, see "Explanation of addresses" on page 420.

Default setting

A1=0, A2=90, A5=0.

Application

Machining position

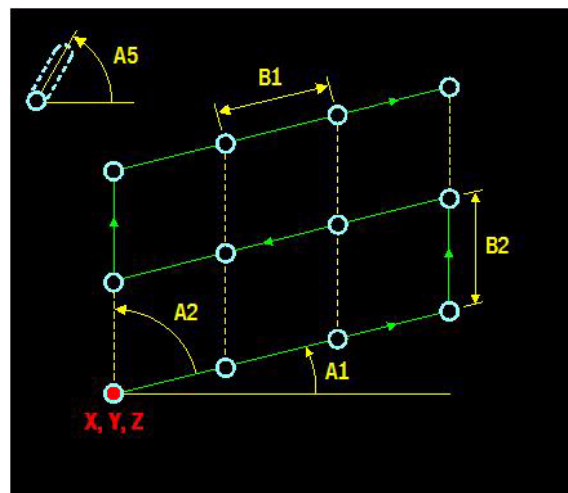
The machining position is defined via X,Y,Z or point definition number P1=.

Pocket angle

The pocket angle is defined by A5.

Procedure

- 1 Rapid traverse to position.
- 2 The machine cycle previously defined is executed at this position.
- 3 The next position is approached after execution. The positions are approached in zigzag movements in the start direction, as determined by angle A1=.
- 4 Repeat procedure (2-3) until all positions (K1=, K2=) have been machined.

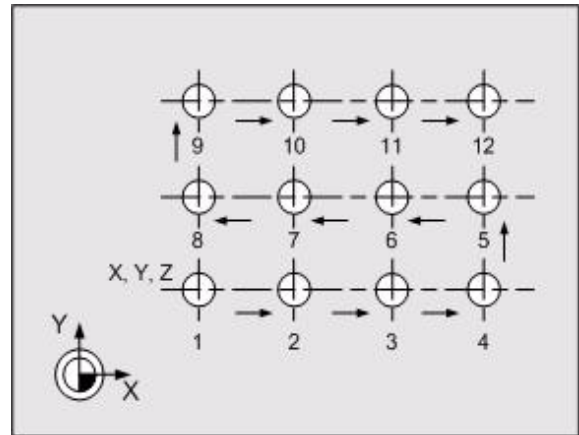


Example

G781 L30 F100 F5=6000

G773 X50 Y20 Z0 B1=40 K1=4 B2=30 K2=3

G781 Define bore cycle
G773 Execute bore cycle on grid with 10 positions



10.9 G777 Operation on Circle

Execution of a machining cycle at points located at fixed equal distances on a circular arc or full circle.

Address description

- ▶ R radius
- ▶ K1= number of operations
- ▶ X, Y, Z center position
- ▶ B2= polar angle
- ▶ L2= polar length
- ▶ P1= point definition number
- ▶ A1= starting angle
- ▶ A2= ending angle
- ▶ A5= angle of rotation
- ▶ F feed

For a description of the additional addresses, see "Explanation of addresses" on page 420.

Default setting

A1=0, A2=360.

Application

Machining position

The machining position is defined via X,Y,Z,B2,L2 or point definition number P1=.

Machining direction

If A2= negative, the holes are clockwise.

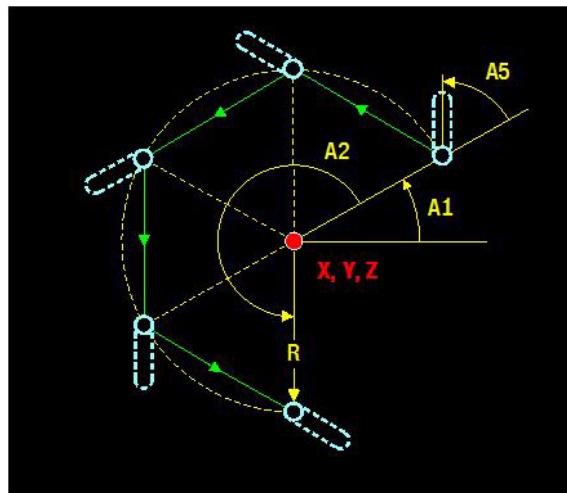
If A2= positive, the holes are counter-clockwise.

Pocket angle

If A5 is not programmed, the pocket angles opposite the principal axis are the same.

If A5=0, then the pocket angle turns with the circle.

If A5 is not equal to 0, an extra rotation is added.



Procedure

- 1 Rapid traverse to position.
- 2 The machine cycle previously defined is executed at this position.
- 3 The next position is approached after execution. The direction of the positions is determined by A1= and A2=.
- 4 Repeat procedure (2-3) until all positions (K1=) have been machined.

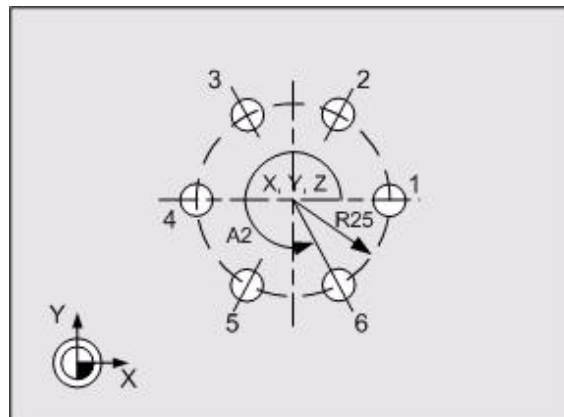
Example

Cycle on a full circle

G781 L30 F100 F5=6000

G777 X50 Y20 Z0 R=25 K1=6 A1=0 A2=300

- G781 Define bore cycle.
G777 Execute bore cycle on circle with 6 points.
- K1=6 (number of holes)
 - A1=0 (starting angle)
 - A2=300 (end angle)



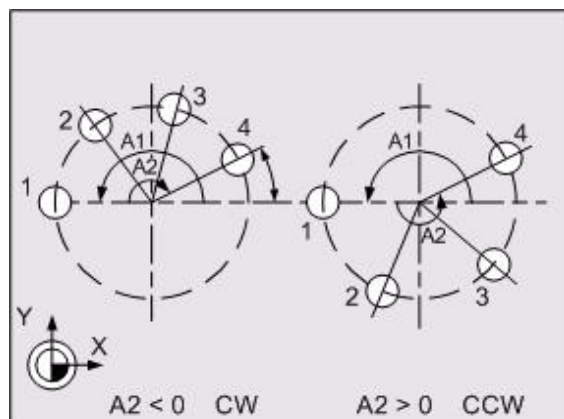
Direction of bore holes on a circular arc

G781 L30 F100 F5=6000

G777 X0 Y0 Z0 R25 A1=180 A2=-150 K1=4

G777 X0 Y0 Z0 R25 A1=-180 A2=210 K1=4

- G781 Define bore cycle.
G777 Repeat the cycle four times on the arc; move from 180 degrees to 30 degrees in a clockwise (CW) direction.
G777 Repeat the cycle four times on the arc; move from 180 degrees to 30 degrees in a counter-clockwise (CCW) direction.



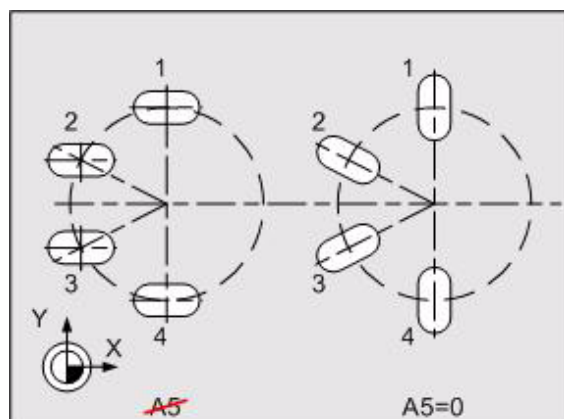
Angle of the slots on a circular arc

G788 B1=16 B2=8 L5 F5=6000

G777 X0 Y0 Z0 R25 A1=90 A2=180 K1=4

G777 X0 Y0 Z0 R25 A1=90 A2=180 K1=4 A5=0

- G788 Define slot cycle.
G777 The slots all have the same direction.
G777 The slot angle is dependent on the position on the circular arc.



10.10 G781 Drilling/Centring

Definition of a simple drilling or centring cycle with possible chip break in a single program block.

Address description

- ▶ L depth
- ▶ L1=, L2= setup clearance
- ▶ C1= cutting depth
- ▶ D3= dwell [revolutions]
- ▶ S spindle speed
- ▶ F5= retract rapid
- ▶ F feed

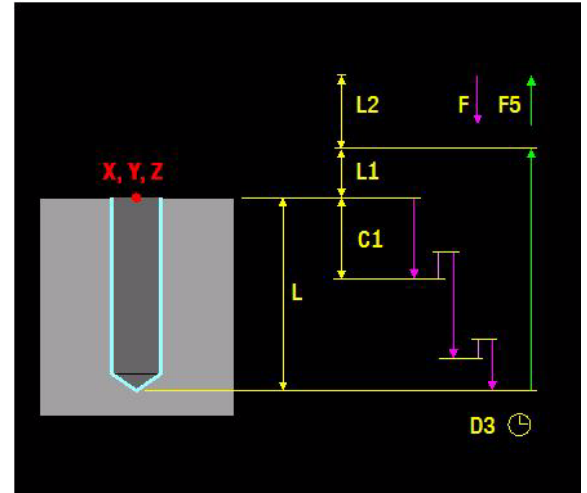
For a description of the additional addresses, see "Explanation of addresses" on page 420.

Default setting

L1=1, L2=0, C1=L, D3=0.

Procedure

- 1 Rapid traverse to 1st setup clearance (L1=).
- 2 Drill with drilling advance by the cutting depth (C1=) or depth (L).
- 3 Rapid retraction (F5=) by 0.2 mm.
- 4 Repeat procedure (2-3) until the depth (L) has been reached.
- 5 At the bottom of the hole, dwell (D3=) for free cutting.
- 6 Rapid retraction (F5=) to the 1st setup clearance (L1=) and rapid traverse back to the 2nd setup clearance (L2=).



Example

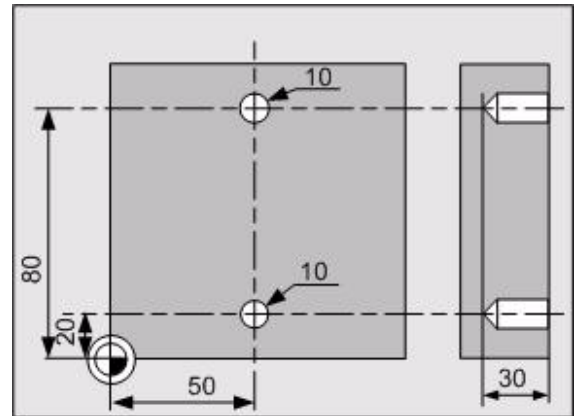
Drilling two holes

G781 L30 F100 F5=6000

G779 X50 Y20 Z0

G779 X50 Y80 Z0

G781 Define bore cycle.
G779 Execute drilling cycle at point 1.
G779 Execute drilling cycle at point 2.



10.11 G782 Deep-Hole Drilling

Definition of a deep-hole drilling cycle with reducing plunging depth for chip break and regular chip removal in a single program block.

Address description

- ▶ **L depth**
- ▶ **L1=, L2= setup clearance**
- ▶ **C1= cutting depth** If the cutting depth (C1=) is not programmed or C1= is greater than or equal to the depth (L), the addresses C2=, C3=, C5=, C6=, C7=, and K1= are meaningless.
- ▶ **C3= minimum cutting depth**
- ▶ **D3= dwell [revolutions]**
- ▶ **S spindle speed**
- ▶ **F2= in depth rapid**
- ▶ **F5= retract rapid**
- ▶ **F feed**

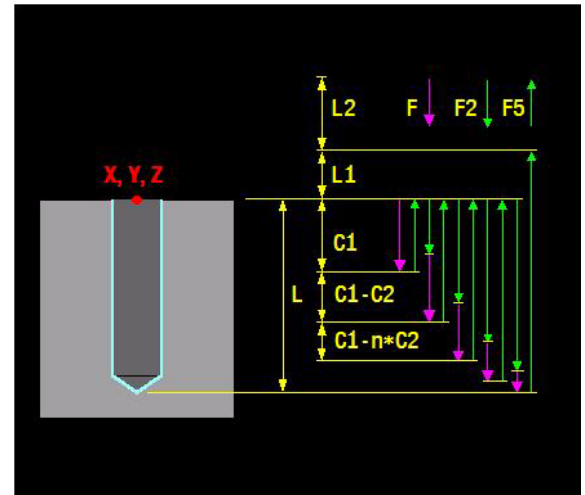
With distributed cuts for chip break and/or chip removal

- ▶ **C2= cutting depth reduction** Value by which the feed depth reduces after every infeed. ($C1 = C1 - n * C2$). The cutting depth (C1=) is always greater than or equal to the minimum feed depth (C3=).
- ▶ **C5= retract distance for chip break** (incremental): Distance by which the tool retracts during chip break.

Chip removal after multiple cuts

- ▶ **K1= number of steps before retract** Number of advance movements (C1=) before the tool retracts from the hole for chip removal. For chip breaking without removal, the tool retracts by the retraction distance (C5=) in each case. No chip removal takes place if K1=0.
- ▶ **C6= safety distance after retract** Safety distance for rapid positioning when the tool returns to the current feed depth after being retracted from the hole. This value applies to the first infeed.
- ▶ **C7= safety dist. after last retract** Safety distance for rapid positioning when the tool returns to the current feed depth after being retracted from the hole. This value applies to the last infeed.
 - If C6= is not equal to C7=, the setup clearance between the first and last cuts is gradually reduced.

For a description of the additional addresses, see "Explanation of addresses" on page 420.



Default setting

L1=1, L2=0, C1=L, C2=0, C3=C2, C5=0.1, C6=0.5, C7=0.5, K1=1, D3=0

Application

Rules for distributed cuts

- 1 The cutting depth is always limited by the drill depth (L).
- 2 If C3 is programmed and there are 2 cuts, the first drilling cut can be reduced.
- 3 Every cut is smaller than or equal to the preceding one.
- 4 If there are more than 2 cuts and a final cut, the final cut and the one preceding it are executed in 2 equal steps. This avoids a very small final cut.

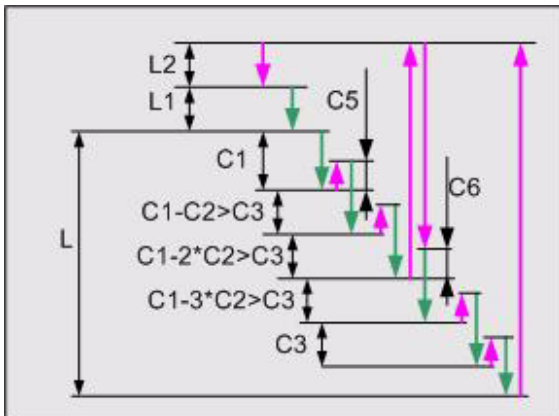
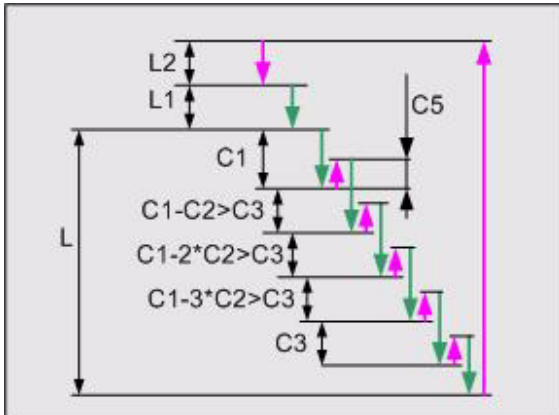
Examples of distributed cuts

Programming	Drilling cuts	Rule
One or two drilling cuts:		
G782 L10 C1=15	10	Rule 1
G782 L10 C1=9	9 1	
G782 L10 C1=9 C3=2	8 2	Rule 2
G782 L10 C1=7 C3=6	8 2	Rules 2 and 3
More than 2 drilling cuts		
G782 L25 C1=7	7 7 5.5 5.5	Rule 4
G782 L25 C1=7 C2=2	7 5 3 2 2 2 2 2	
G782 L24 C1=7 C2=2	7 5 3 2 2 2 1.5 1.5	Rule 4
G782 L29 C1=7 C2=2 C3=3	7 5 3 3 3 3 2.5 2.5	Rule 4

Machining sequence

Input: C1=..., K1=large (see figure)

Input: C1=..., K1=3 (see figure)



Procedure

- 1 Rapid traverse to 1st setup clearance (L1).
- 2 Drill with drilling advance by the cutting depth (C1=).
- 3 With chip break: reverse movement by the retract distance (C5=).
With chip removal: rapid retraction (F5=) upwards followed by rapid plunging (F2=) as far as the safety distance (C6= up to C7= down).
- 4 The feed depth (C1=) is then reduced by the cutting depth decrement (C2=). The minimum feed depth is equal to C3=.
- 5 Repeat procedure (2-4) until the drill depth (L) has been reached.
- 6 At the bottom of the hole, dwell (D3=) for free cutting.
- 7 Rapid retraction (F5=) to the 1st setup clearance (L1=) and rapid traverse back to the 2nd setup clearance (L2=).

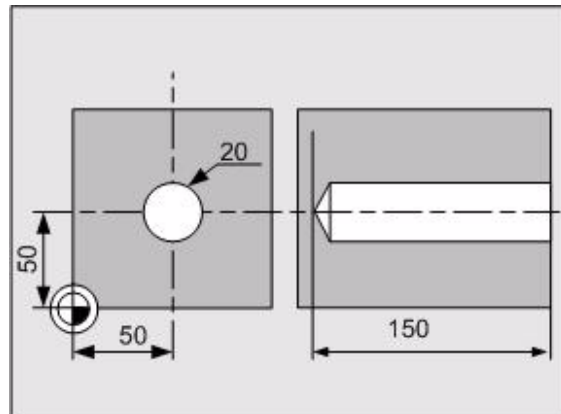
Example

Deep-hole drilling

G782 L150 L1=4 C1=20 C2=2 C3=6

G79X X50 Y50 Z0

G782 Define deep-hole drilling cycle
G79 Execute deep-hole drilling cycle



10.12 G783 Deep-Hole Drill. Add Chip Break

Definition of a deep-hole drilling cycle with reducing feed depth for chip removal and a fixed chip break distance in a single program block.

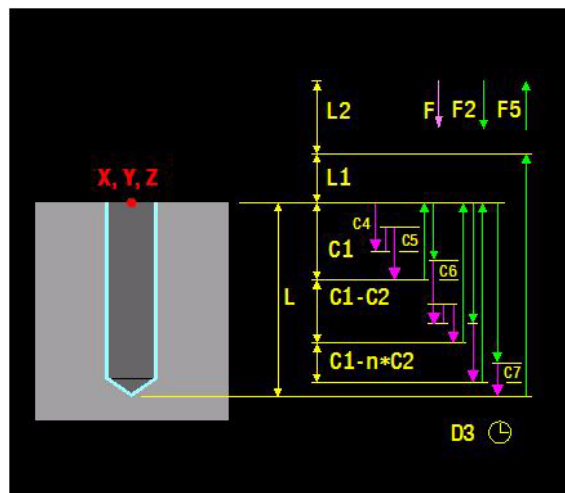
Address description

- ▶ **L** depth
- ▶ **L1=**, **L2=** setup clearance
- ▶ **C1= cutting depth** If the cutting depth (C1=) is not programmed or C1= is greater than or equal to the depth (L), the addresses C2=, C3=, C4=, C5=, C6=, and C7= are meaningless.
- ▶ **C2= cutting depth reduction**
- ▶ **C3= minimum cutting depth**
- ▶ **C4= drilling depth before chip break** Advance after which a chip break is executed. No chip break if $C4 > C1$ or is not programmed (addresses C6= and C7= are meaningless).
- ▶ **C5= retract distance for chip break**
- ▶ **C6= safety distance after retract** Safety distance for rapid positioning when the tool returns to the current feed depth after being retracted from the hole. This value applies to the first infeed.
- ▶ **C7= safety dist. after last retract** Safety distance for rapid positioning when the tool returns to the current feed depth after being retracted from the hole. This value applies to the last infeed.
 - If C6= is not equal to C7=, the setup clearance between the first and last cuts is gradually reduced.
- ▶ **D3= dwell [revolutions]**
- ▶ **S** spindle speed
- ▶ **F2= in depth rapid**
- ▶ **F5= retract rapid**
- ▶ **F** feed

For a description of the additional addresses, see "Explanation of addresses" on page 420.

Default setting

L1=1, L2=0, C1=L, C2=0, C3=C1, C4=C1, C5=0.1, C6=0.5, C7=C6, D3=0



Application

Cutting depth

If more than 2 cuts are required, the final cut and the one preceding it are executed in 2 equal steps. This avoids a very small final cut.

Machining sequence

Input: $C1=...$, $C4=C1$ (see figure)

Input: $C1=...$, $C4<C1$ (see figure)

Procedure

- 1 Rapid traverse to the 1st setup clearance.
- 2 No chip break ($C4 \geq C1$ or $C4$ not programmed): Drill with drilling advance by the cutting depth ($C1=$).
With chip break ($0 < C4 < C1$): Drill by depth ($C4=$). Then retract by the retraction distance ($C5=$). Repeat until the cutting depth ($C1=$) is reached.
- 3 Rapid retraction ($F5=$) upwards followed by rapid plunging ($F2=$) as far as the safety distance ($C5=$ up to $C7=$ down).
- 4 The feed depth ($C1=$) is then reduced by the cutting depth decrement ($C2=$). The minimum feed depth is equal to $C3=$.
- 5 Repeat procedure (2-4) until the drill depth (L) has been reached.
- 6 At the bottom of the hole, dwell ($D3=$) for free cutting.
- 7 Rapid retraction ($F5=$) to the 1st setup clearance ($L1=$) and rapid traverse back to the 2nd setup clearance ($L2=$).

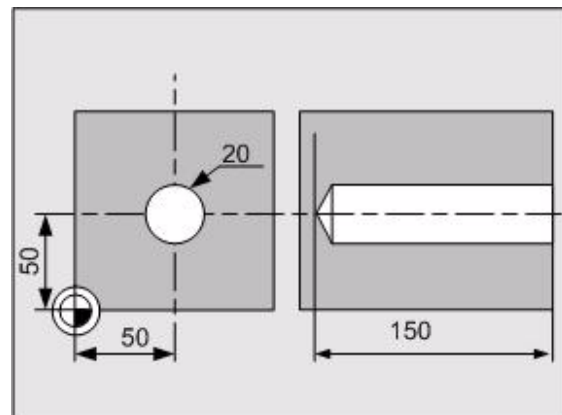
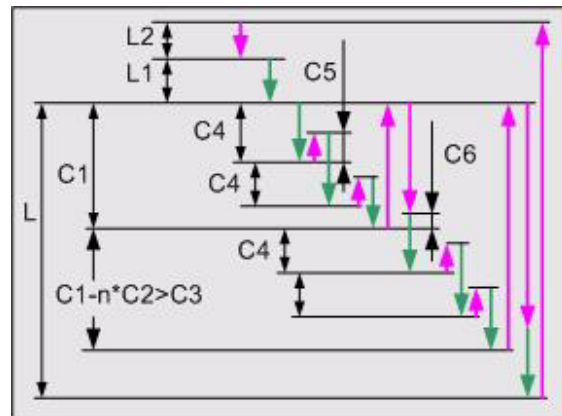
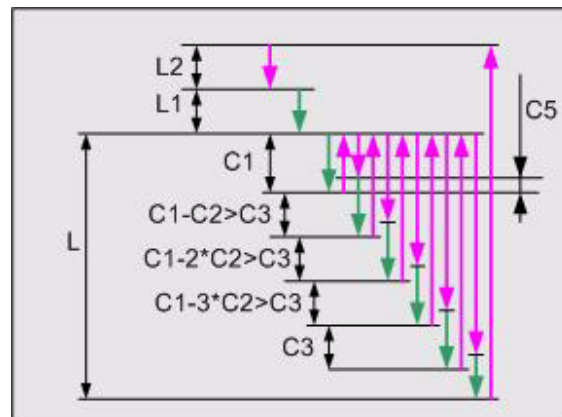
Example

Deep-hole drilling with chip break

G783 L150 L1=4 C1=20 C=5 C2=2 C3=6 C5=0.5 F200

G79 X50 Y50 Z0

G783 Define deep-hole drilling cycle.
G79 Execute deep-hole drilling cycle.



10.13 G784 Tapping

Definition of a tapping cycle in a single program block.

Address description

- ▶ **L depth** (> 0)
- ▶ **F2= pitch**
- ▶ **L1= 1st setup clearance** Reference value: 4x pitch.
- ▶ **L2= 2nd setup clearance**
- ▶ **D3= dwell time [s]** Time in seconds that the tool remains at the hole bottom.
- ▶ **C1= cutting depth** Advance after which a chip break is executed. No algebraic sign.
- ▶ **C5= retract distance for chip break** The tool is retracted by the specified distance during chip breaking. Entering 0 means that it is fully retracted from the hole (to the safety clearance) for chip removal. No algebraic sign.
- ▶ **F feed**

For a description of the additional addresses, see "Explanation of addresses" on page 420.

Default setting

L1=1, L2=0, D3=0.

Application

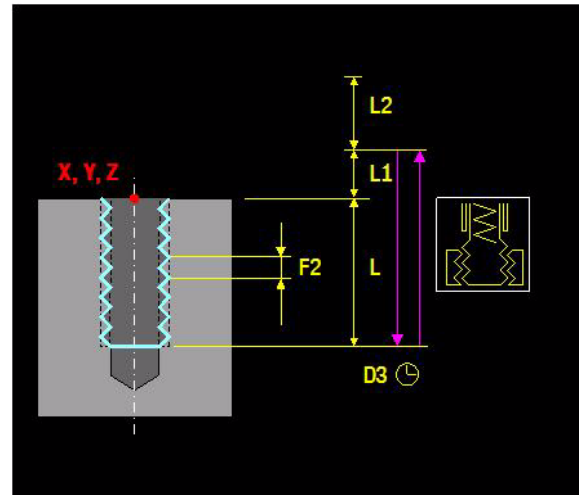
The tool must be clamped in a floating tap holder. A floating tap holder compensates for the advance and speed tolerances during machining.

At the end of the cycle, the coolant and spindle are restored to their pre-cycle status.

The advance is determined by the speed. Speed override is active during tapping. Feed override is not active.

When a G784-cycle is called using G79, the CNC must be set to G94-mode (feed in mm/min), not G95-(feed in mm/rev).

The machine and CNC must be prepared for the G784 cycle by the machine manufacturer.



Procedure

- 1 Rapid traverse in the spindle axis to the 1st setup clearance (L1=).
- 2 Tapping with pitch (L3=) to depth (L).
- 3 After the dwell time (D3=), the direction of spindle rotation is reversed.
- 4 The tool is retracted with pitch (L3=) to the 1st setup clearance (L1=) and then rapidly retracted to the 2nd setup clearance (L2=).
- 5 At the end, the direction of spindle rotation is reversed once more.

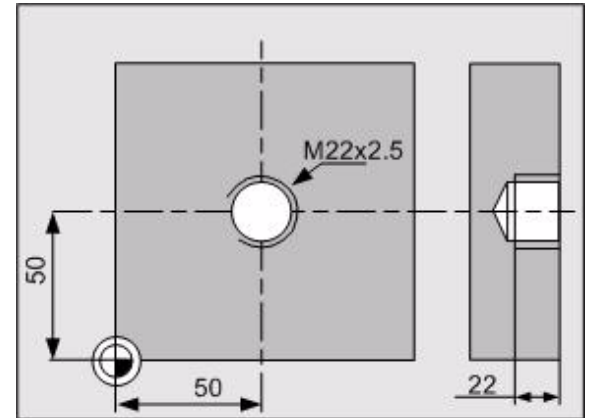
Example

Tapping

G784 L22 L1=9 L3=2.5

G79 X50 Y50 Z0

- G784 Define tapping cycle.
A floating tap holder must be used.
- G79 Execute the cycle at the programmed position.



10.14 G785 Reaming

Definition of a reaming cycle in a single program block.

Address description

- ▶ **L** depth
- ▶ **L1=**, **L2=** setup clearance
- ▶ **I1=** spindle stop 0=yes 1=no
 - I1=0 rapid retraction and stationary spindle.
 - I1=1 retraction with feed and rotating spindle.
- ▶ **D3=** dwell [revolutions]
- ▶ **S** spindle speed
- ▶ **F5=** retract rapid Rapid traverse (I1=0) or feed (I1=1) retraction: Traversing speed of the tool when retracting from the hole in mm/min.
- ▶ **F** feed

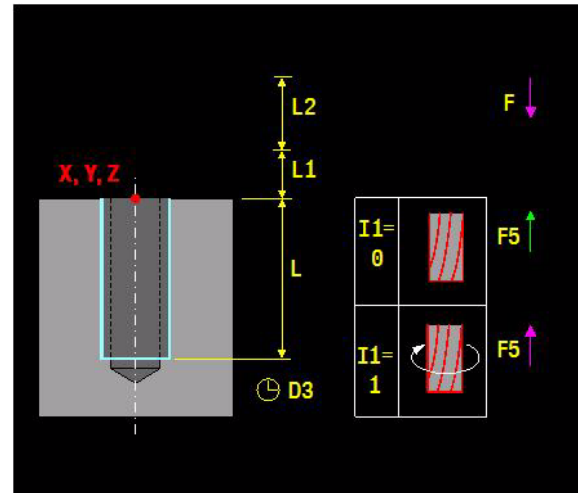
For a description of the additional addresses, see "Explanation of addresses" on page 420.

Default setting

L1=1, L2=0, I1=0, D3=0

Procedure

- 1 Rapid traverse to 1st setup clearance (L1=).
- 2 Reaming with feed F down to depth (L).
- 3 Dwell at bottom of hole (D3=).
- 4 Rapid retraction (F5=).
 - To the setup clearance (L1=)
 - To the 2nd setup clearance (L2=) in rapid traverse



Example

Reaming

G785 L29 D3=2 F100 F5=2000

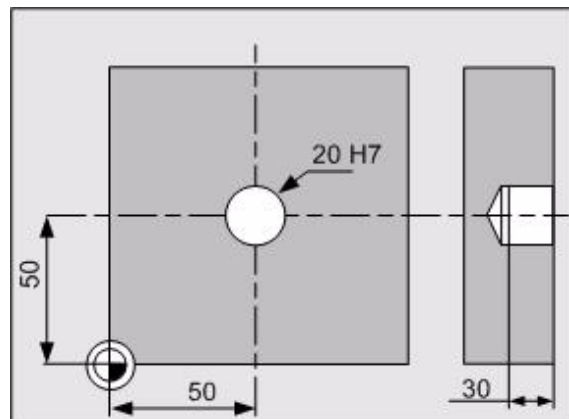
G79 X50 Y50 Z0

G785

Define reaming cycle.

G79

Execute the reaming cycle at the programmed position.



10.15 G786 Boring

Definition of a reverse boring cycle with the option of disengaging an oriented spindle in a single program block.

Address description

- ▶ **L depth**
- ▶ **L1=, L2= setup clearance**
- ▶ **C1= retract distance from side** Distance by which the tool is retracted from the wall when disengaging.
- ▶ **D orientation angle tool tip** Angle (absolute) at which the tool positions itself before disengaging (I1=2 only). The disengage direction is -X in G17/G18 and -Y in G19.
- ▶ **D3= dwell [revolutions]**
- ▶ **I1= retract 0=M5 1=M3/M4 2=M19**
 - I1=0 Retract with with rapid traverse and stationary spindle without disengaging
 - I1=0 Retract with with feed and rotating spindle without disengaging
 - I1=2 Retract with oriented spindle (M19) and in rapid traverse.
- ▶ **S spindle speed**
- ▶ **F5= retract rapid** Rapid traverse (I1=0 or I1=2) or feed (I1=1) retraction: Traversing speed of the tool when retracting from the hole in mm/min.
- ▶ **F feed**

For a description of the additional addresses, see "Explanation of addresses" on page 420.

Default setting

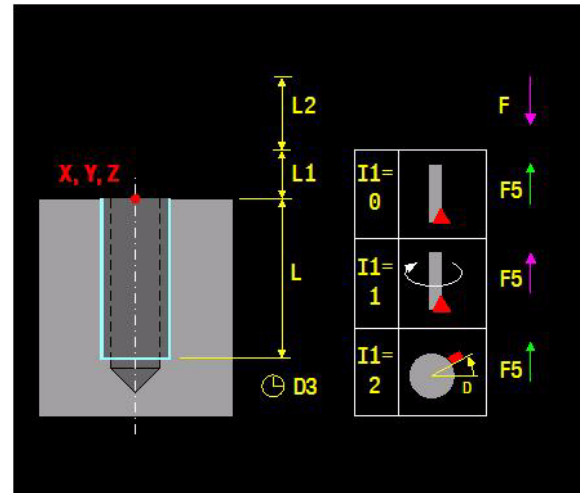
L1=1, L2=0, C1=0.2, D=0, D3=0, I1=0, F5=rapid traverse (I1=0 or I1=2) or F5=F (I1=1)

Application

At the end of the cycle the spindle status that was active before the cycle is reactivated.



Note: The tool tip must be aligned (MDI) such that it points to the positive principal axis. The angle displayed must be entered as the orientation angle (D) so that the tool moves away from the edge of the hole in the direction of the negative principal axis. The disengage direction is -X in G17/G18 and -Y in G19.



Procedure

- 1 Rapid traverse to 1st setup clearance (L1=).
- 2 Reverse boring with feed (F) down to depth (L).
- 3 Dwell (D3=) at bottom of hole with running spindle for free cutting.
- 4 With I1=2, the tool performs a spindle orientation (D=) and a reverse movement along the principal axis by the retraction distance (C1=).
- 5 Rapid retraction (F5=) to the 1st setup clearance (L1=) and rapid traverse back to the 2nd setup clearance (L2=).

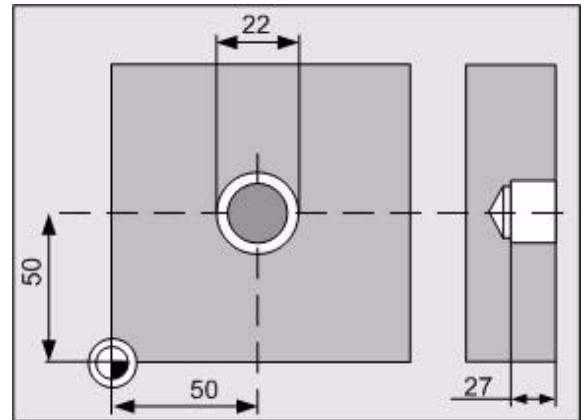
Example

Boring

G786 L27 L1=4 L2=10 D3=1 F100

G79 X50 Y50 Z0

G786 Define boring cycle.
G79 Execute the cycle at the programmed position.



10.16 G787 Pocket Milling

Definition of a pocket milling cycle for rough machining of rectangular pockets in a single program block. This cycle allows oblique plunging and mills in a continuous spiral path.

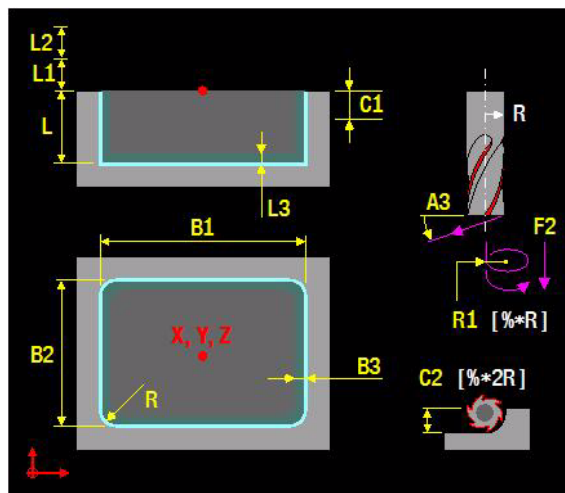
Address description

- ▶ **B1= 1st side length** Length of the pockets in the principal axis
- ▶ **B2= 2nd side length** Width of the pockets in the minor axis
- ▶ **L depth**
- ▶ **L1=, L2= setup clearance**
- ▶ **L3= finishing allowance bottom**
- ▶ **B3= finishing allowance sides**
- ▶ **C1= plunging depth**
- ▶ **C2= proportional cutting width** Percentage of the tool diameter to be used as the cutting width on each pass. The total width is divided into equal sections.
- ▶ **R rounding radius** Radius for the pocket corners. If radius $R=0$, the rounding radius is the same as the tool radius.
- ▶ **R1= proportional helix radius** Percentage of the tool radius to be used as the cutting width (>0) for oblique plunging.
- ▶ **A3= plunging angle** Angle ($0..90^\circ$) at which the tool can plunge into the workpiece. The plunging angle is adjusted so that the tool always plunges with a whole number of rectangular movements. It only plunges vertically at 90° .
- ▶ **I1= milling 1=climb -1=conventional**
- ▶ **S spindle speed**
- ▶ **F2= feed for plunging**
- ▶ **F feed**

For a description of the additional addresses, see "Explanation of addresses" on page 420.

Default setting

$L1=1$, $L2=0$, $L3=0$, $B3=0$, $C1=L$, $C2=67\%$, R = tool radius, $R1=80\%$, $A3=90$, $I1=1$, $F2=0.5 \cdot F$ for vertical plunging, and $F2=F$ for oblique plunging.



Application

B1= and B2= must be greater than 2*(tool radius + finishing allowance for sides B3).

For finishing, the dimensions L3 and B3 must be entered.

Procedure

- 1 Rapid traverse to 1st setup clearance (L1=) over the pocket center.
- 2 If the plunging angle $A3=90^\circ$, the tool advances with feed (F2=) to the first feed depth (C1=). If the plunging angle $A3<90^\circ$, the tool advances obliquely to the first feed depth (C1=), with plunging feed and a whole number of rectangular movements.
- 3 Machining with feed (F) in the positive direction of the long side, in a flowing movement from inside to outside.
- 4 At the end of this process, the tool is retracted from the wall and the floor in a tangent to the helix and moved rapidly to the center.
- 5 Repeat procedure (2-4) until the depth (L) has been reached.
- 6 At the end, a rapid traverse movement to the 1st plus 2nd setup clearance (L1= plus L2=) is performed.

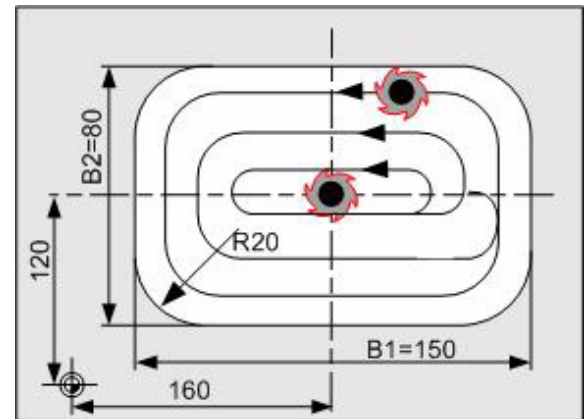
Example

Pocket milling

G787 B1=150 B2=60 L6 L1=1 A3=5 C1=3 C2=60 R20 I1=1 F200

G79 X160 Y120 Z0

G787 Define pocket milling cycle.
G79 Execute the cycle at the programmed position.



10.17 G788 Key-Way Milling

Definition of a pocket milling cycle for rough machining and/or finishing of a slot in a single program block. This cycle allows oblique plunging.

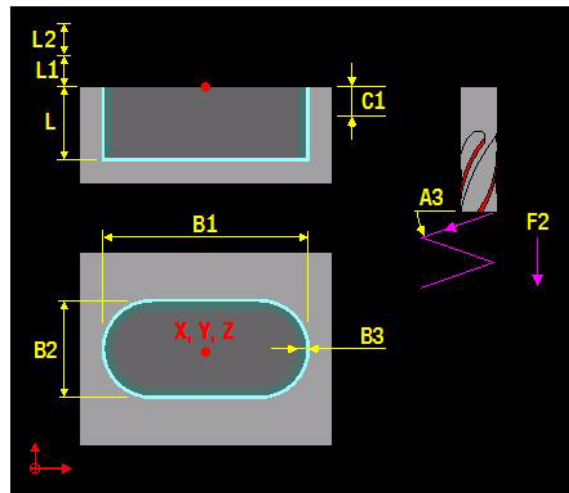
Address description

- ▶ **B1= 1st side length** Length of the slot in the principal axis
- ▶ **B2= 2nd side length** Width of the slot in the minor axis. If the slot width is the same as the tool diameter, only roughing is performed.
- ▶ **L depth**
- ▶ **L1=, L2= setup clearance**
- ▶ **B3= finishing allowance sides**
- ▶ **C1= plunging depth roughing**
- ▶ **A3= plunging angle** Maximum angle (0..90°) at which the tool can plunge into the workpiece. It only plunges vertically at 90°.
- ▶ **I1= milling 1=climb -1=conventional**
- ▶ **0=roughing 1=roughing + finishing** Roughing or finishing:
 - 0: only roughing
 - 1: roughing and finishing.
- ▶ **S spindle speed**
- ▶ **F2= feed for plunging**
- ▶ **F feed**

For a description of the additional addresses, see "Explanation of addresses" on page 420.

Default setting

L1=1, L2=0, B3=0, C1=L, A3=90, I1=1, I2=0, F2=0.5*F for vertical plunging and F2=F for oblique plunging.



Application

When roughing with oblique plunging, the tool performs reciprocating plunges to cut the material from one end of the slot to the other. Pilot drilling is therefore not necessary.

Vertical plunging is always performed into the end of the slot on the negative side. Pilot drilling is required at this point.

The diameter of the milling cutter must be no greater than the width of the slot and no smaller than a third of the slot width.

The cutter diameter must be smaller than half the slot length; otherwise the CNC cannot perform a reciprocating plunge.

For finishing, the dimension (B3=) must be entered.

Procedure

Roughing:

- 1 Rapid traverse to the 1st setup clearance (L1=) and into the center of the left circle.
- 2 If the plunging angle $A3=90^\circ$, the tool advances with feed (F2=) to the first feed depth (C1=) and then with feed F into the center of the right circle. If the plunging angle $A3<90^\circ$, the tool advances obliquely and with plunging feed (F2=) into the center of the right circle. The tool then moves back to the center of the left circle, again plunging obliquely. These steps are repeated until the cutting depth (C1=) is reached.
- 3 At the milling depth, the tool moves to the other end of the slot and then machines the slot shape until the finishing dimension is reached.
- 4 Repeat procedure (2–3) until the programmed depth (L) has been reached.

Finishing:

- 5 The tool moves tangentially to the contour in the left or right circle of the slot and finishes it using climb milling (I1=1).
- 6 At the end of the contour, the tool retracts tangentially from the contour and floor to the center of the slot.
- 7 At the end, a rapid traverse movement to the 1st plus 2nd setup clearance (L1= plus L2=) is performed.

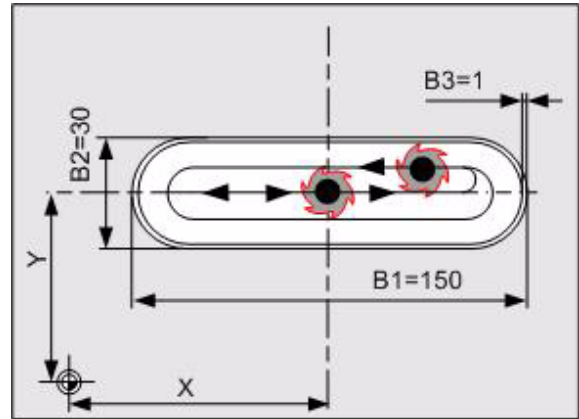
Example

Key-way milling

G788 B1=150 B2=30 L6 L1=1 A3=5 C1=3 I1=1 I2=0 F200

G79 X20 Y20 Z0

- G788 Define the key-way milling cycle, parallel to the X-axis.
- G79 Execute the cycle at the programmed position.



10.18 G789 Circular Pocket Milling

Definition of a pocket milling cycle for rough machining of circular pockets in a single program block. This cycle allows oblique plunging and mills a continuous spiral path.

Address description

- ▶ **R** radius
- ▶ **L** depth
- ▶ **L1=**, **L2=** setup clearance
- ▶ **L3=** finishing allowance bottom
- ▶ **B3=** finishing allowance sides
- ▶ **C1=** plunging depth
- ▶ **C2=** **proportional cutting width** Percentage of the tool diameter to be used as the cutting width on each pass. The total width is divided into equal sections.
- ▶ **R1=** **proportional helix radius** Percentage of the tool radius to be used as the cutting width (>0) for oblique plunging.
- ▶ **A3=** **plunging angle** Angle (0..90°) at which the tool can plunge into the workpiece. It only plunges vertically at 90°
- ▶ **I1=** **milling 1=climb -1=conventional**
- ▶ **S** spindle speed
- ▶ **F2=** **feed for plunging**
- ▶ **F** feed

For a description of the additional addresses, see "Explanation of addresses" on page 420.

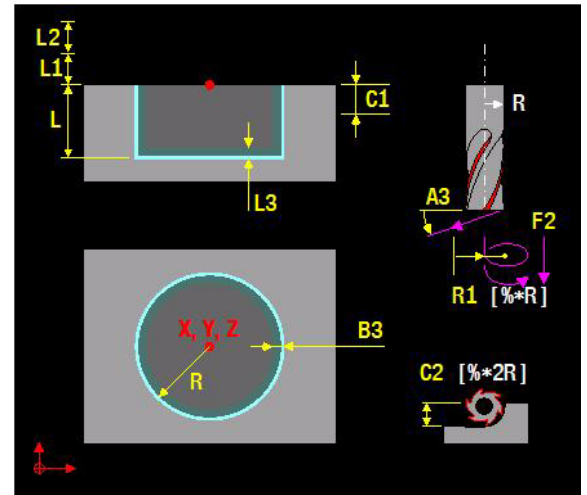
Default setting

L1=1, L2=0, L3=0, B3=0, C1=L, C2=67%, R1=80%, A3=90, I1=1, F2=0.5*F for vertical plunging and F2=F for oblique plunging.

Application

R must be greater than 2*(tool radius + finishing allowance for sides B3=).

For finishing, the dimensions L3 and B3 must be entered.



Procedure

- 1 Rapid traverse to 1st setup clearance ($L1=$) over the pocket center.
- 2 If the plunging angle $A3=90^\circ$, the tool advances with feed ($F2=$) to the first feed depth ($C1=$).
If the plunging angle $A3<90^\circ$, the tool advances obliquely to the first feed depth ($C1=$), with plunging feed and a number of circular movements.
- 3 Machining with feed (F) in an outward-moving spiral.
- 4 At the end of this process, the tool is retracted from the wall and the floor in a tangent to the helix and moved rapidly to the center.
- 5 Repeat procedure (2-4) until the depth (L) has been reached.
- 6 At the end, a rapid traverse movement to the 1st plus 2nd setup clearance ($L1=$ plus $L2=$) is performed.

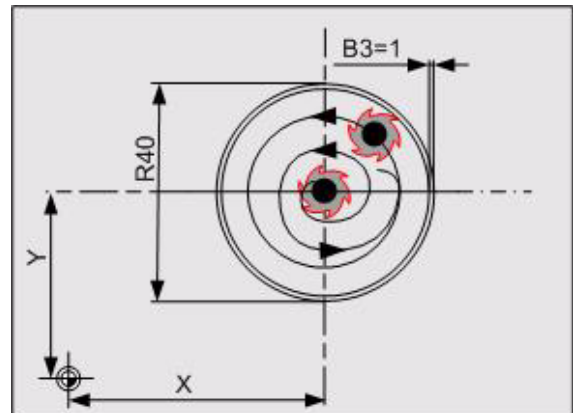
Example

Circular pocket milling

G789 R0 L6 L=1 A3=5 C1=3 C2=65 I1=1 F200

G79 X160 Y20 Z0

G789 Define pocket milling cycle.
G79 Execute the cycle at the programmed position.



10.19 G790 Back-Boring

Definition of a back-boring cycle in a single program block. The cycle operates only with reverse boring bars to create counterbores on the underside of the workpiece.

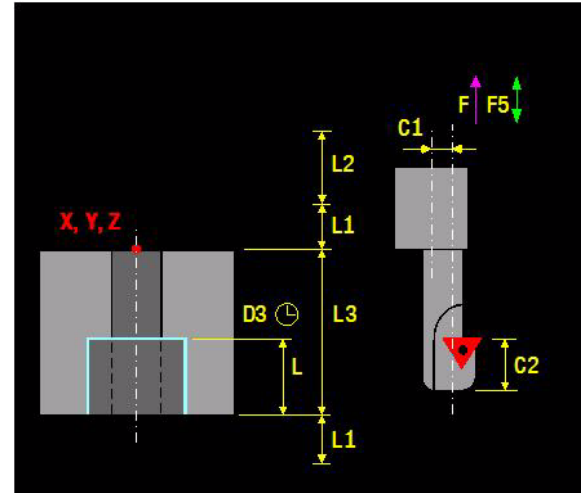
Address description

- ▶ **L counterbore depth**
- ▶ **L3= material thickness**
- ▶ **C1= eccentricity** Eccentricity of the boring bar (to be taken from the tool data sheet).
- ▶ **L1=, L2= setup clearance**
- ▶ **C2= cutting edge height** Distance from bottom edge of boring bar to main cutter (to be taken from the tool data sheet).
- ▶ **D orientation angle tool tip** Angle (absolute) at which the tool is positioned before plunging into and retracting out of the hole. The disengage direction is -X in G17/G18 and -Y in G19.
- ▶ **D3= dwell [revolutions]**
- ▶ **S spindle speed**
- ▶ **F5= retract rapid**
- ▶ **F feed**

For a description of the additional addresses, see "Explanation of addresses" on page 420.

Default setting

L1=1, L2=0, C2=0, D=0, D3=0.2, F5=rapid traverse.



Application

Enter the tool length so that the cutting plate of the boring bar is measured.

The CNC takes the height of the cutting edge (C2=) into account when calculating the starting point.

At the end of the cycle, the spindle status that was active before the cycle is reactivated.



Danger of collision!

The tool tip must be aligned (MDI) such that it points to the positive principal axis. The angle displayed must be entered as the orientation angle (D) so that the tool moves away from the edge of the hole in the direction of the negative principal axis. The disengage direction is -X in G17/G18 and -Y in G19.

Procedure

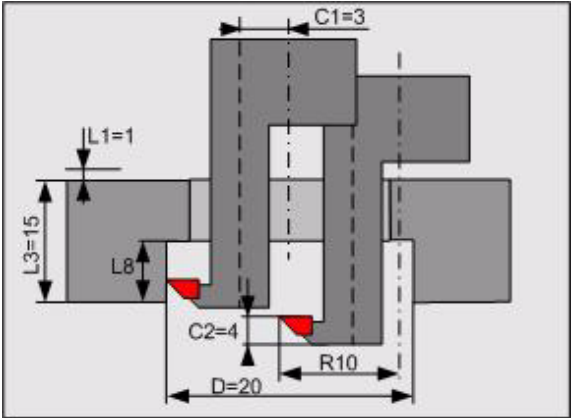
- 1 Rapid traverse to 1st setup clearance (L1=).
- 2 Spindle orientation to the D position and tool offset by the eccentricity (C1=).
- 3 Rapid retract (F5=) plunging into the pre-drilled hole until the cutting edge is at the 1st setup clearance (L1=) below the bottom of the workpiece.
- 4 Movement to the center of the hole, switch on spindle and coolant, and machine at countersinking feed to the specified depth.
- 5 At the bottom of the hole, the tool dwells with a running spindle for free cutting.
- 6 The tool then moves out of the hole, performs spindle orientation, and is once again displaced by the eccentricity (C1=).
- 7 At the end, rapid retraction (F5=) to the 1st setup clearance (L1=) and rapid traverse to the 2nd setup clearance (L2=)

Example

Back boring

```
T1 M6
G790 L3=15 L8 L1=1 C1=3 C2=4 F100
G79 X30 Y40 Z0
```

- T1 Insert tool.
Tool radius R10
Eccentricity C1=3
Cutting edge height C2=4
Angle for spindle orientation D0
- G790 Define back boring cycle.
- G79 Execute defined cycle at point.



10.20 G794 Tapping, Interpolated

Definition of a tapping cycle with interpolation in a single program block.

Address description

- ▶ **L depth**
- ▶ **F2= pitch**
- ▶ **L1=, L2= setup clearance**
- ▶ **C1= cutting depth** Advance after which a chip break is executed. No algebraic sign.
- ▶ **C5= retract distance for chip break** The tool is retracted by the specified distance during chip breaking. Entering 0 means that it is fully retracted from the hole (to the safety clearance) for chip removal. No algebraic sign.
- ▶ **D orientation angle spindle** Angle at which the tool is positioned before the thread is cut. This allows you to regroove the thread, if required.
- ▶ **F feed**

For a description of the additional addresses, see "Explanation of addresses" on page 420.

Default setting

L1=1, L2=0.

Application

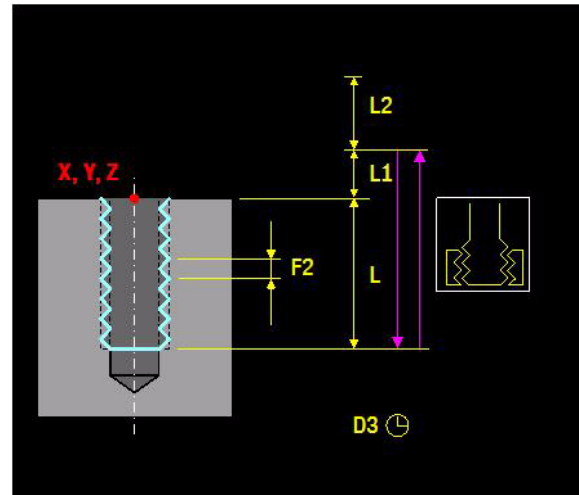
At the end of the cycle, the coolant status and spindle status that were active before the cycle are reactivated.

The advance is determined by the speed. Speed override is active during tapping. Feed override is not active.

When a G794-cycle is called using G79, the CNC must be set to G94-mode (feed in mm/min).

In the case of spindle orientation, the machine parameters must be correctly set during tapping. The spindle acceleration is calculated for each gear using maxFeed and maxAccSpeedCtrl in CFGFeedLimits.

The machine and CNC must be prepared for the G794 cycle by the machine manufacturer.



Procedure

- 1 Rapid traverse in the spindle axis to the 1st setup clearance (L1=) followed by spindle orientation.
- 2 Tapping with pitch (L3=) to depth (L).
- 3 The direction of spindle rotation is then reversed once more.
- 4 The tool is retracted with pitch (L3=) to the 1st setup clearance (L1=) and then rapidly retracted to the 2nd setup clearance (L2=).
- 5 The spindle is stopped here.

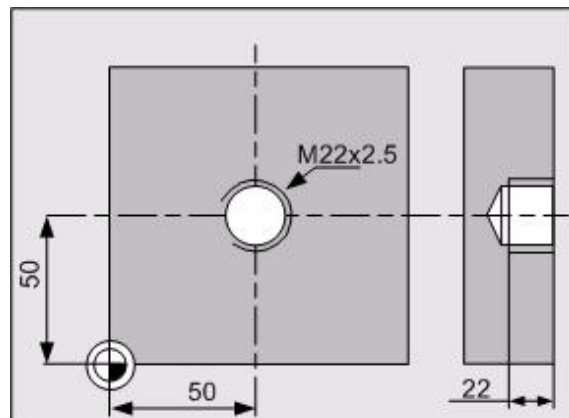
Example

Tapping, interpolated

G794 L22 L1=9 L3=2.5

G79 X50 Y50 Z0

G794 Define the tapping cycle.
G79 Execute the cycle at the programmed position.



10.21 G797 Pocket Finishing

Definition of a rectangular pocket milling cycle for finishing the wall and floor of rectangular pockets in a single program block. The sides can be machined in a number of advances. This cycle allows oblique plunging into the floor and mills in a continuous spiral path.

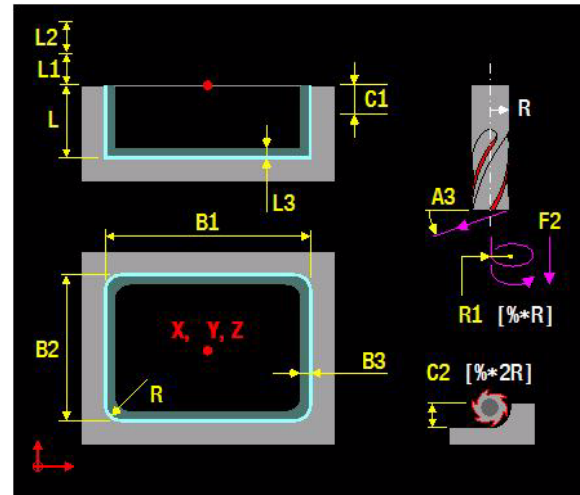
Address description

- ▶ **B1= 1st side length** Length of the slot in the principal axis
- ▶ **B2= 2nd side length** Width of the slot in the minor axis
- ▶ **L depth**
- ▶ **L1=, L2= setup clearance**
- ▶ **L3= allowance bottom** Milled away during finishing.
- ▶ **B3= allowance sides**
- ▶ **C1= plunging depth**
- ▶ **C2= proportional cutting width** Percentage of the tool diameter to be used as the cutting width on each pass. The total width is divided into equal sections.
- ▶ **R rounding radius** Radius for the pocket corners. If radius $R=0$, the rounding radius is the same as the tool radius.
- ▶ **R1= proportional helix radius** Percentage of the tool radius to be used as the helix radius (>0) for plunging.
- ▶ **A3= plunging angle** Angle ($0..90^\circ$) at which the tool can plunge into the workpiece. The plunging angle is adjusted so that the tool always plunges with a whole number of rectangular movements. It only plunges vertically at 90° .
- ▶ **I1= milling 1=climb -1=conventional**
- ▶ **I2= finishing 0=complete 1=sides**
 - 0: finishing of side and bottom
 - 1: finishing of side only
- ▶ **S speed**
- ▶ **F2= feed for plunging**
- ▶ **F feed**

For a description of the additional addresses, see "Explanation of addresses" on page 420.

Default setting

$L1=1$, $L2=0$, $L3=0$, $B3=1$, $C1=L$, $C2=67\%$, R = tool radius, 0, $R1=80\%$, $A3=90$, $I1=1$, $F2=0.5 \cdot F$ for vertical plunging, and $F2=F$ for oblique plunging.



Application

B1= or B2= must be greater than 2*(tool radius + finishing allowance for sides B3=).

Procedure

- 1 Rapid traverse to 1st setup clearance (L1=) over the pocket center.

Finishing the floor:

- 2 If the plunging angle $A3=90^\circ$, the tool advances with drilling feed (F2=) to the depth (L).
If the plunging angle $A3<90^\circ$, the tool advances obliquely, using a whole number of rectangular movements, to the depth (L).
- 3 Machining with feed (F) in the positive direction of the longer side, in a flowing movement from inside to outside.
- 4 At the end of this process, the tool is retracted from the wall and the floor in a tangent to the helix and moved rapidly to the center.

Finishing the side:

- 5 Rapid traverse to the plunging depth (C1=).
- 6 The starting position is the first plunging depth and at least the finishing allowance (B3=) from the side. The tool moves inward tangentially, mills the contour, and retracts tangentially.
- 7 Repeat procedure (5-6) until the depth (L) has been reached.
- 8 At the end of the cycle, the tool moves rapidly to the 1st plus 2nd setup clearances (L1= plus L2=) and then into the center of the pocket.

Example

Pocket finishing

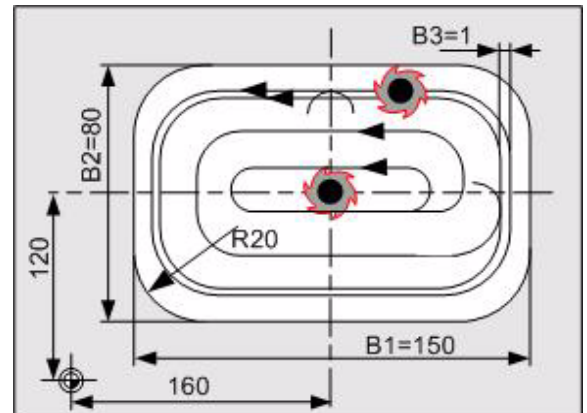
G787 B1=150 B2=80 B3=1 L6 I1=1 L3=1 R20 A3=5 C2=65 C1=3.

G79 X160 Y120 Z0

G797 B1=150 B2=80 B3=1 L6 L3=1 R20 A3=5 C2=60 C1=3

G79 X160 Y120 X0

- | | |
|------|---|
| G787 | Define pocket milling roughing cycle. |
| G79 | Execute the roughing cycle at the programmed position. |
| G797 | Define pocket finishing cycle. |
| G79 | Execute the finishing cycle at the programmed position. |



10.22 G798 Key-Way Finishing

Definition of a key-way milling cycle for finishing in a single program block.

Address description

- ▶ **B1= 1st side length** Length of the slot in the principal axis
- ▶ **B2= 2nd side length** Width of the slot in the minor axis
- ▶ **L depth**
- ▶ **L1=, L2= setup clearance**
- ▶ **C1= plunging depth**
- ▶ **I1= milling 1=climb -1=conventional**
- ▶ **S speed**
- ▶ **F feed**

For a description of the additional addresses, see "Explanation of addresses" on page 420.

Default setting

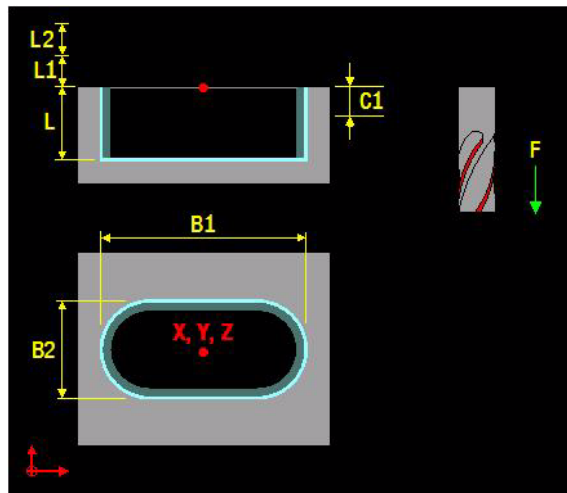
L1=1, L2=0, C1=L, I1=1.

Application

The diameter of the milling cutter must be no greater than the width of the slot and no less than a third of the slot width.

Procedure

- 1 Rapid traverse to 1st setup clearance (L1=) over the slot center.
- 2 The tool moves tangentially to the contour from the center of the slot and finishes it using climb milling (I1=1).
- 3 At the end of the contour, the tool retracts tangentially from the contour and floor to the center of the slot
- 4 The tool then moves rapidly to the 1st plus 2nd setup clearances (L1= plus L2=).



Example

Key-way finishing

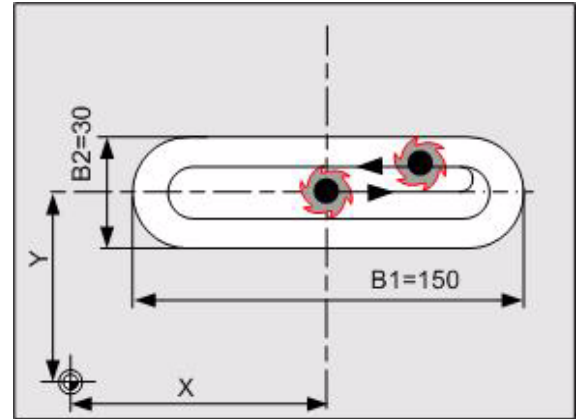
**G788 B1=150 B2=20 B3=1 L6 L1=1 A3=10 C1=3 I1=1 I2=0 F100
F2=200**

G79 X20 Y20 Z0

G798 B1=150 B2=30 L6 L1=1 I1=1 F100

G79 X20 Y20 Z0

- G788 Define the key-way roughing cycle, parallel to the X-axis.
- G79 Execute the roughing cycle at the programmed position.
- G798 Define the key-way finishing cycle, parallel to the X-axis.
- G79 Execute the finishing cycle at the programmed position.



10.23 G799 Circular Pocket Finishing

Definition of a circular pocket milling cycle for finishing the wall and floor of rectangular pockets in a single program block. The sides can be machined in a number of advances. This cycle allows oblique plunging into the floor and mills in a continuous spiral path.

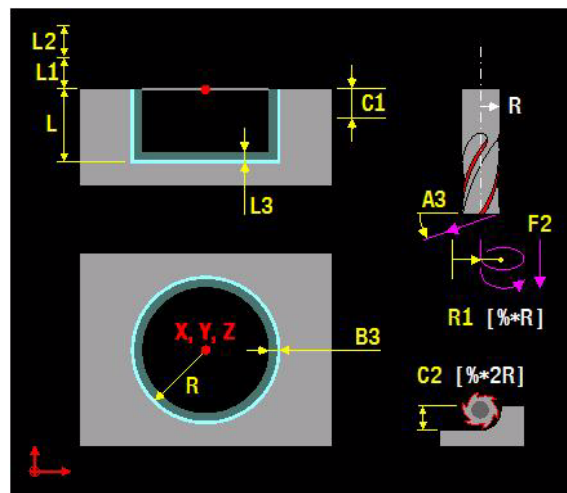
Address description

- ▶ **R radius**
- ▶ **L depth**
- ▶ **L1=, L2= setup clearance**
- ▶ **L3= finishing allowance bottom** Milled away during finishing.
- ▶ **B3= finishing allowance sides** Milled away during finishing.
- ▶ **C1= plunging depth**
- ▶ **C2= proportional cutting width** Percentage of the tool diameter to be used as the cutting width on each pass. The total width is divided into equal sections.
- ▶ **R1= proportional helix radius**
- ▶ **A3= plunging angle**
 - Angle (0 to 90°) at which the tool can plunge into the workpiece
 - It only plunges vertically at 90°
- ▶ **I1= milling 1=climb -1=conventional**
- ▶ **I2= finishing 0=complete 1=sides**
 - 0: finishing of side and bottom
 - 1: finishing of side only
- ▶ **S speed**
- ▶ **F2= feed for plunging**
- ▶ **F feed**

For a description of the additional addresses, see "Explanation of addresses" on page 420.

Default setting

L1=1, L2=0, L3=1, B3=1, C1=L, C2=67%, R1=80%, A3=90, I1=1, I2=0, F2=0.5*F for vertical plunging and F2=F for oblique plunging.



Application

The minimum size of the pocket (R) is $2 \times (\text{tool radius} + \text{finishing allowance for sides } B3=)$.

Procedure

Finishing the floor

- 1 Rapid traverse to the center of the pocket and remain at the 1st setup clearance ($L1=$) above the center of the pocket.
- 2 If the plunging angle $A3=90^\circ$, the tool advances with feed ($F2=$) to the depth (L).
If the plunging angle $A3<90^\circ$, the tool advances obliquely, using a whole number of circular movements, to the depth (L).
- 3 The tool then moves in a spiral path (direction depends on forward rotation ($I1=1$) with $M3$) and then roughs the floor of the pocket from inside to outside.

Finishing the side

- 4 Rapid traverse to the plunging depth ($C1=$).
- 5 The side is then machined in a number of sections. The starting position is the first plunging depth and at least the finishing allowance ($B3=$) from the side. The tool then moves inward tangentially, mills the contour, and retracts tangentially.
- 6 Repeat procedure (4-5) until the depth (L) has been reached.
- 7 At the end of the cycle, the tool moves rapidly to the 1st plus 2nd setup clearances ($L1=$ plus $L2=$) and then into the center of the pocket.

Example

Circular pocket finishing

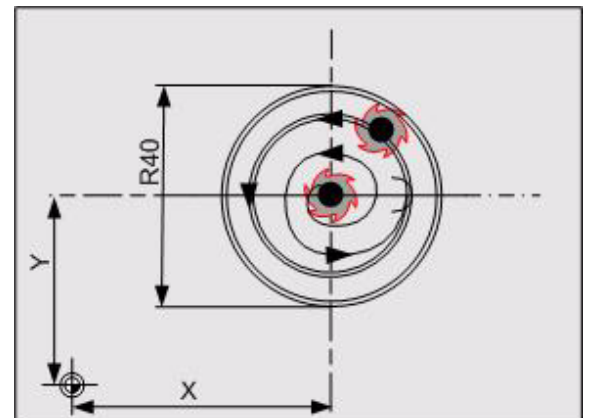
G789 R40 L6 B3=1 I1=1 L1=1 L3=1 A3=5 C2=65 C1=3 F200

G79 X160 Y120 Z0

G799 R40 B3=1 L6 L1=1 L3=1 A3=5 C1=3 C2=65 I1=1 F200

G79 X160 Y120 Z0

- | | |
|------|---|
| G787 | Define circular pocket roughing cycle. |
| G79 | Execute the roughing cycle at the programmed position. |
| G797 | Define circular pocket finishing cycle. |
| G79 | Execute the finishing cycle at the programmed position. |



11

**G800-G899 Turning
Cycles**

11.1 Turning Cycles

Reserved for turning cycle extensions

Availability

These cycles will appear in a future version.

12

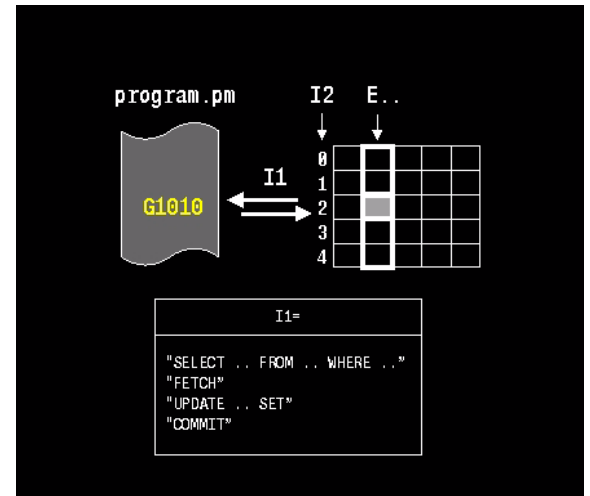
**G1000-G1099 G-Codes
for Macros**

12.1 G1010 Edit Function for SQL tables

The function allows you to read from and write to an SQL table. The action is executed using an SQL statement, with which a part of the SQL table is selected. You can read and write data in this selection.

Address description

- ▶ **E parameter with number of SQL selection** This number is automatically assigned with the SQL statement "SELECT ..". This number has to be specified with all other SQL statements that use the selection.
- ▶ **I1= SQL statement** Defines the actual SQL statement in a string enclosed in double quotation marks.
 - "SELECT .. FROM .. WHERE .."
 - "FETCH"
 - "UPDATE .. SET"
 - "COMMIT"
- ▶ **I2= record index** Address I2= can be used to select a specific data record. I2= is only permitted with I1="FETCH". If address I2= is not programmed, and was never programmed in the program before, FETCH retrieves the first data record from the table. If the address I2= is not programmed in the subsequent G1010 block, FETCH retrieves the next data record from the table.
- ▶ **O1= parameter number for result** Parameter number in which the result of the SQL statement is written.
 - 0 = SQL statement successful
 - 1 = SQL statement not successful (e.g. searched column not found)



Application

Statement

The statement to be executed must be enclosed in double quotation marks. Within a statement, a reference to an E parameter can be used for the condition (WHERE). To do so, enclose the E parameter in single quotation marks after a colon. MillPlus replaces this sequence with the value of the E parameter.

Example:

```
G1010 E5 I1="SELECT L,R FROM TOOL WHERE T_NR=17"
```

Handle

The SQL HANDLE describes the result of a previous SQL query and is stored in the E parameter (e.g. E5). Only values assigned by the SQL server are valid handles. The value 0 identifies an invalid handle. With the SELECT command, the handle is assigned a value. In the case of the UPDATE, COMMIT, and FETCH commands, the handle must have a value.

Example:

```
G1010 E5 I1="SELECT L,R FROM TOOL WHERE T_NR==1"
```

FETCH

FETCH uses the SQL result (e.g. E5) of the previous SQL query, after which the data can be read from the columns using the SQLRead() function. If the values in the table are expressed in inches, lengths and feed rates are converted into millimeters during the reading process. The values in the bound parameters are always assumed to be metric. As with G1018, this also applies if the current program is entered in inches. If no I2= is specified, the first row of the result set is transferred. The specified E parameter (e.g. E80) is assigned a return code. If the statement is completed successfully, the E parameter is assigned the value "0". If not, it is assigned the value "1".

Example:

```
G1010 E5 I1="FETCH" I2=4 O1=80
```

UPDATE

UPDATE assigns the data written with the SQLwrite() function to the relevant table rows or table columns. If the values in the table are expressed in inches, lengths and feed rates are converted into millimeters before the assignment process. The values in the bound parameters are always assumed to be metric. The specified E parameter (e.g. E80) is assigned a return code. If the statement is completed successfully, the E parameter is assigned the value "0". If not, it is assigned the value "1".

Example:

```
G1010 E5 I1="UPDATE" O1=80
```

COMMIT

COMMIT cancels locks on table rows or table columns. Edited table data is permanently transferred using COMMIT. The specified E parameter (e.g. E80) is assigned a return code. If the statement is completed successfully, the E parameter is assigned the value "0". If not, it is assigned the value "1".

Example:

```
G1010 E5 I1="COMMIT" O1=80
```

SELECT

To select data, use the SQL statement SELECT. In the SELECT command, you enter the data source (table name) and the relevant column names. Enter the data source after the keyword FROM. The SELECT command provides various command options for defining conditions, sorting sequences, and locks, which modify the effect of the command.

WHERE

The WHERE option limits the effect of a command to the rows of the selected columns that satisfy the specified condition. The condition can be defined by directly entering a numeric value or using the contents of an ES parameter.

Example:

The row to be assigned to tool T 1 is selected from the columns L and R of table TOOL.T (WHERE T=1):

```
G1010 E5 I1="SELECT L,R FROM '%USR%\TABLE\TOOL.T'
WHERE T_NR=1"
```

The contents of parameter ES21 can also be used for defining the WHERE condition. For example, ES21 contains the value "1":

```
G1010 E5 I1="SELECT L,R FROM '%USR%\TABLE\TOOL.T'
WHERE T_NR='&ES21&'"
```

ORDER BY

The ORDER BY option defines the sequence of rows. Select a column by which the rows are to be sorted.

Example:

The rows to be assigned to tool T 0 and to the tool numbers from the E parameter E31 are selected from the columns L and R of the TOOL.T table. The result set is sorted by tool number T_NR:


```
G1010 E5 I1="SELECT L,R FROM '%USR%\TABLE\TOOL.T'
WHERE T_NR=0 OR T_NR="&ES31&" ORDER BY T_NR"
```

FOR UPDATE

The FOR UPDATE option locks the rows during selection in order to prevent unauthorized access. Without the FOR UPDATE option, the rows are not locked until immediately before they are changed (COMMIT command).

Example:

```
G1010 E5 I1="SELECT L,R FROM '%USR%\TABLE\TOOL.T'
WHERE T_NR=0 OR T_NR="&ES31&" ORDER BY T_NR FOR
UPDATE"
```



The keywords must be written in upper case

Example

Reading the length of tool 17

G1010 E0 I1="SELECT L FROM TOOL WHERE T_NR=17"	E0=SQL HANDLE
G1010 E0 I1="FETCH" O1=1	FETCH DATA RECORD
IF (E1 = 0) THEN	
E2 = CDBL(SQLREAD(E0, "L"))	E2=TOOL LENGTH
ELSE	
G300 D602	TOOL NOT FOUND
END IF	
G1010 E0 I1="COMMIT"	END TRANSACTION

12.2 G1016 Export Formatted Text and E Parameter

With the G1016 function, you can output formatted parameter values or texts to a file or display them on screen. To output the formatted texts and parameter values, use a text editor to create a text file, in which you then specify the formats and parameters to be output. When writing text to a file, you can either overwrite the file or append text to the file.

Address description

- **N= output definition**
- **N5= name of format file**

Application

Output definition

Address N defines an output to a file or on-screen display. To output to a file, you must enter a string in double quotation marks along with the relevant path and file name. The path is relative to <%USR%\>.

Example: N="MeasuringResult\BladeWheel.txt" writes a file <BladeWheel.txt> in the directory<MeasuringResult>

Display on screen

If N="screen:", the output is displayed on screen. A pop-up window is opened during the first write operation. This window is closed again after either of the following:

- N="sclr:"
- End (M30) of the NC program
- <ESC> key when the window is selected

Format file name

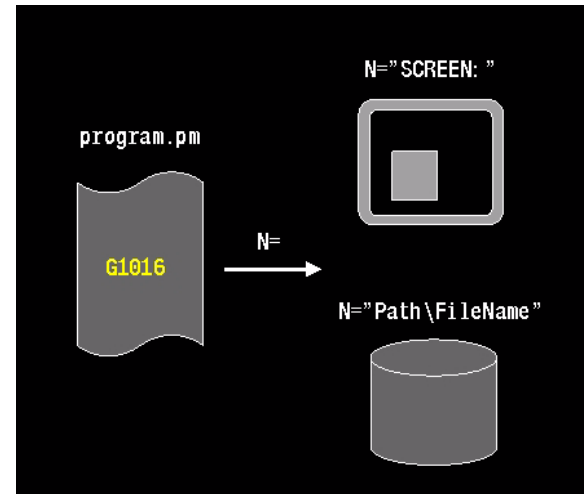
Address N5= defines the format file. To do this you must enter a string in double quotation marks along with the relevant path and file name. The path must be absolute.

Example: N5="%USR%\Format\Messprotokol.cfg"

Format variables

The format file can have the following format variables:

- Q.. E parameter is defined with Q, e.g. Q10
- QS.. ES parameter is defined with QS, e.g. QS12



Keywords

The format file can have the following keywords:

Key	Description
CALL_PATH	Outputs the path name for the NC program in which the G1016 function is located.
M_APPEND	If the log file already exists, the new output data is added.
M_CLOSE	Closes the file to which you are writing with G1016. The file can be read. If M_CLOSE is not programmed, the file will be closed at the end of the program
L_ENGLISH	Display text only in English conversational
L_GERMAN	Display text only in German conversational
L_DUTCH	Display text only in Dutch conversational
L_FRENCH	Display text only in French conversational
L_ITALIAN	Display text only in Italian conversational
L_SPANISH	Display text only in Spanish conversational
L_PORTUGUESE	Display text only in Portuguese conversational
L_DANISH	Display text only in Danish conversational
L_SWEDISH	Display text only in Swedish conversational
L_FINNISH	Display text only in Finnish conversational
L_CZECH	Display text only in Czech conversational
L_POLISH	Display text only in Polish conversational
L_ALL	Display text independently of the conversational language
HOUR	Number of hours from the real-time clock
MIN	Number of minutes from the real-time clock
SEC	Number of seconds from the real-time clock
DAY	Day from the real-time clock
MONTH	Month as a number from the real-time clock
STR_MONTH	Month as a string abbreviation from the real-time clock
YEAR2	Two-digit year from the real-time clock
YEAR4	Four-digit year from the real-time clock

Format functions

The format file can have the following format functions:

- "....." Define output for text and variables between double quotation marks
- %5.3f Define format (e.g. for E parameter): 5 places before and 3 places after the decimal point, floating. If can be written instead of f
- %s Format for text variable
- %d Format for date and time, examples:

Format	Example
%1d-%1d-%4d	2-2-2008
%02d-%02d-%4d	02-12-2007
%2d-%2d-%4d	14- 3-2008

- , Separator between output format, variables, and keywords
- ; End of record character, finishes a row and starts comment on this row
- ""; Empty row

Example

G1016 N5="%USR%\Format\Messprotokol.cfg" N="screen:"

The format file Messprotokol.cfg (MeasuringLog.cfg):

"Measuring result of blade wheel mass center";
"Date: %02d-%02d-%4d",DAY,MONTH,YEAR4;
"Time: %02d:%02d:%02d", HOUR,MIN,SEC;
"_____",
";"
"X = %5.3lf", Q10;
"Y = %5.3lf", Q11;
"Z = %5.3lf", Q12;

Delivers the following screen output:

Measuring result of blade wheel mass center
Date: 21-09-2007
Time: 12:22:45

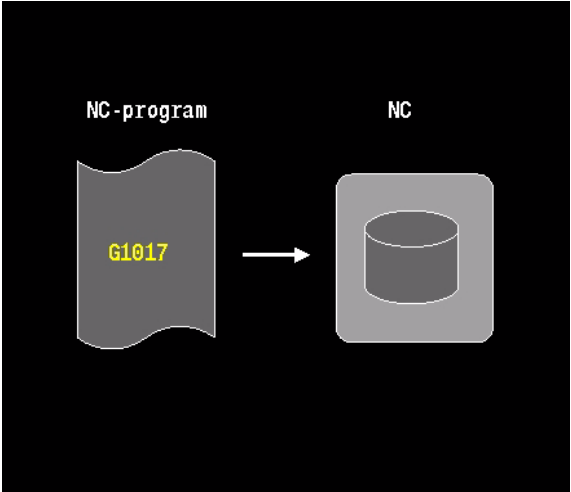
X = 134.998
Y = 24.989
Z = 0.008

12.3 G1017 Write NC System Data

This function enables the NC program to write NC system data. Examples of NC system data are limit switches, axes parameters, and touch probe settings.

Address description

- ▶ **E** parameter number with system data
- ▶ **I1=** group number
 - 61 = Write tool definition
 - 230 = Write software limit switch
 - 350 = Touch probe
 - 610 = Write LookAhead parameter
 - 990 = Write start-up behavior
- ▶ **I2=** system data number
- ▶ **I3=** index of system data (default is 0)
- ▶ **I4=** value of system data



Address	Description
I1=61 I2=1	TOOL DEF Tool number (T column in tool magazine); the value I4= is stored in the interpreter
I1=61 I2=4	TOOL DEF Tool index (IDX column in tool table); the value I4= is stored in the interpreter
I1=230 I2=2	Negative software limit switch Write negative software limit switch of axis I3= with the value I4= I3=1 X axis I3=2 Y axis .. I3=4 A axis I3=5 B axis .. I3=9 W axis I4= New value for limit switch [mm deg].

Address	Description
I1=230 I2=3	Positive software limit switch Write positive software limit switch of axis I3= with the value I4= I3=1 X axis I3=2 Y axis .. I3=4 A axis I3=5 B axis .. I3=9 W axis I4= New value for limit switch [mm deg].
I1=230 I2=4	Software limit switches for multiple axes The positive and negative software limit switches of multiple axes are changed I3= number of axes starting with the X axis E= initial value of E parameter range The first two E parameters contain the new values of the positive and negative limit switches of the X axis, the next two E parameters contain the new values of the Y axis and so on.
I1=230 I2=5	Switch software limit switches on and off Software limit switch monitoring can be switched off or on: I4=0 switch off I4=1 switch on
I1=350 I2=70 I3=1	Touch probe data The touch probe type of the tool measuring system (tool touch probe) is written
I1=350 I2=75 I3=	Touch probe data The feed rate of the tool measuring system (tool touch probe) is written I3=1 rapid traverse I3=2 measuring feed rate when spindle is stationary
I1=610 I2=	LookAhead parameter of path The LookAhead parameter of the path is written with the value from the E parameter: I2=1 minimum path feed rate [mm/min] I2=2 minimum corner feed rate [mm/min] I2=3 feed rate limit for high speed [mm/min] I2=4 maximum jerk (normal) [m/s3] I2=5 maximum jerk (at high speed) [m/s3] I2=6 tolerance (normal) [mm] I2=7 tolerance (at high speed) [mm] I2=8 maximum yank [m/s4] I2=9 contour tolerance factor [-] I2=10 contour jerk factor [-] I2=11 filter frequency [Hz] I2=12 angle tolerance (normal) [mm] I2=13 angle tolerance (at high speed) [mm] I2=99 reset all LookAhead parameters (to be programmed with E=0)

Address	Description
I1=610 I2= I3=	LookAhead parameter of axes The LookAhead parameter of axis I3= is written with the value from the E parameter: I2=20 maximum feed rate [mm/min] I2=21 maximum acceleration [m/s2] I2=22 maximum brake acceleration [m/s2] I2=23 maximum jerk [m/s3] I2=24 feed rate acceleration compensation [As2/rev] I3=1 X axis I3=2 Y axis .. I3=4 A axis I3=5 B axis .. I3=9 W axis
I1=990 I2=2	Switch touch probe monitoring on and off Touch probe monitoring can be switched off or on: I4=0 switch off I4=1 switch on
I1=990 I2=6	Touch probe active or inactive Tool touch probe monitoring can be switched off or on: I4=0 switch off I4=1 switch on

Application

Procedure

The value of the new system data is transferred by the NC program and stored in the NC.

Configuration

IpoCfgSchema.doc specifies the LookAhead parameter (I1=610).

Example

The negative software limit switch of the X axis is written.

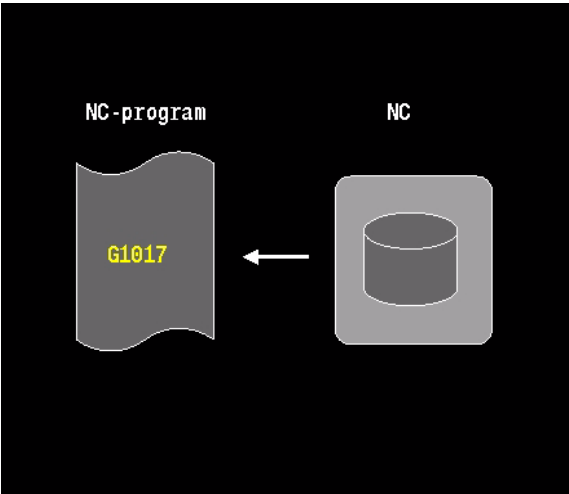
```
G1017 I1=230 I2=2 I3=1 I4=E861 E1
```

12.4 G1018 Read NC System Data

This function enables the NC program to read NC system data. Examples of NC system data are limit switches, axes parameters, and touch probe settings.

Address description

- ▶ **E** parameter receiving parameter
- ▶ **I1=** group ID
 - 20 = Read machine status
 - 60 = Read M67 tool definition
 - 61 = Write tool definition
 - 230 = Read software limit switch
 - 610 = Read LookAhead parameter
- ▶ **I2=** system parameter NR
- ▶ **I3=** index of system parameter (default is 0)



Address	Description
I1=20 I2=	Machine status The machine status is read and stored in the E parameter I2=1 current tool number I2=2 number of the prepared tool I2=3 tool axis (X=1,Y=2,Z=3,U=7,V=8,W=9) I2=4 programmed spindle speed I2=5 spindle status: undefined = -1, turns CW=0, turns CCW=1, stops after turning CW=2, stops after turning CCW=3 I2=6 no function I2=7 no function I2=8 coolant status (off=0, on=1) I2=9 last programmed feed rate (rapid traverse=-1) I2=10 step index of the prepared tool I2=11 step index of the active tool
I1=60 I2=1	M67 TOOL Tool number (T column in tool magazine); the value is stored in the E-Parameter (no SQL)
I1=60 I2=8	M67 TOOL Tool index (IDX column in tool table); the value is stored in the E parameter (no SQL)
I1=61 I2=1	TOOL DEF Tool number (T column in tool magazine); the value is stored in the E parameter (no SQL)
I1=61 I2=4	TOOL DEF Tool index (IDX column in tool table); the value is stored in the E parameter (no SQL)

Address	Description
I1=230 I2=2	Negative software limit switch The negative software limit switch of axis I3= is read and stored in the E parameter: I3=1 X axis I3=2 Y axis .. I3=4 A axis I3=5 B axis .. I3=9 W axis
I1=230 I2=3	Positive software limit switch The positive software limit switch of axis I3= is read and stored in the E parameter: I3=1 X axis I3=2 Y axis .. I3=4 A axis I3=5 B axis .. I3=9 W axis
I1=230 I2=4	Software limit switches for multiple axes Not permitted
I1=230 I2=5	Software limit switch monitoring Software limit switch monitoring can be switched off or on: 0: switched off 1: switched on
I1=610 I2=	LookAhead parameter of path The LookAhead parameter of the path is written with the value from the E parameter: I2=1 minimum path feed rate [mm/min] I2=2 minimum corner feed rate [mm/min] I2=3 feed rate limit for high speed [mm/min] I2=4 maximum jerk (normal) [m/s ³] I2=5 maximum jerk (at high speed) [m/s ³] I2=6 tolerance (normal) [mm] I2=7 tolerance (at high speed) [mm] I2=8 maximum yank [m/s ⁴] I2=9 contour tolerance factor [-] I2=10 contour jerk factor [-] I2=11 filter frequency [Hz] I2=12 angle tolerance (normal) [mm] I2=13 angle tolerance (at high speed) [mm] I2=99 reset all LookAhead parameters (to be programmed with E=0)

Address	Description
I1=610 I2= I3=	LookAhead parameters of axes The LookAhead parameter of axis I3= is read and stored in the E parameter. The following parameters are defined for each axis and programmed with I3=: I2=20 maximum feed rate [mm/min] I2=21 maximum acceleration [m/s2] I2=22 maximum brake acceleration [m/s2] I2=23 maximum jerk [m/s3] I2=24 feed rate acceleration compensation [As2/rev] I3=1 X axis I3=2 Y axis .. I3=4 A axis I3=5 B axis .. I3=9 W axis

Application

Procedure

The value of the system data is stored as an E parameter.

Configuration

IpoCfgSchema.doc specifies the LookAhead parameter (I1=610).

Example

The current tool number is read.

```
G1018 I1=20 I2=1 E2
```


12.5 G1019 Define up to Two PLC values

With the G1019 function, you can transfer up to two numerical values to the PLC in sync.

Address description

- ▶ I1= PLC value
- ▶ I2= PLC value

Application

Output to PLC

The values are decoded in the PLC. Used, for example, in the tool change macro: G1019 I1=101 I2=E861 (tool number transfer)

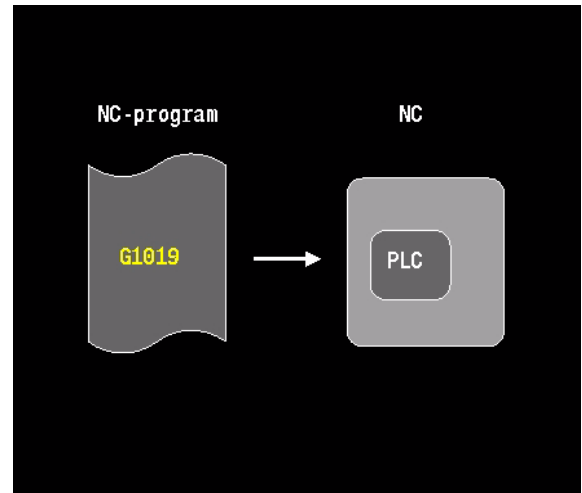
Configuration

With machine parameter CfgPlcMStrobe.

Example

Tool number transfer, e.g. in tool change macro.

```
G1019 I1=101 I2=E861
```



12.6 G1022 Activate Tool Exchange in PLC

General function for writing data to the NC. Note: Texts are to be used for activating tool changes in the PLC (I1=950 and I1=955)

Address description

- ▶ **I1= group ID**
 - 850 = Adapt kinematic model intermittently
 - 950 = Prepare tool data
 - 955 = Tool change to PLC
- ▶ **I2= I1=95x: tool exchange mode (0=DEF,1=CALL)**
- ▶ **I3= I1=95x: tool number**
- ▶ **I4= I1=95x: tool offset index**
- ▶ **I5= I1=95x: tool position in magazine**
- ▶ **I6= I1=95x: SQL handle tool data table**
- ▶ **I7= I1=95x: SQL handle tool magazine table**
- ▶ **N5= I1=850: place in kinematic model**

Application

Procedure

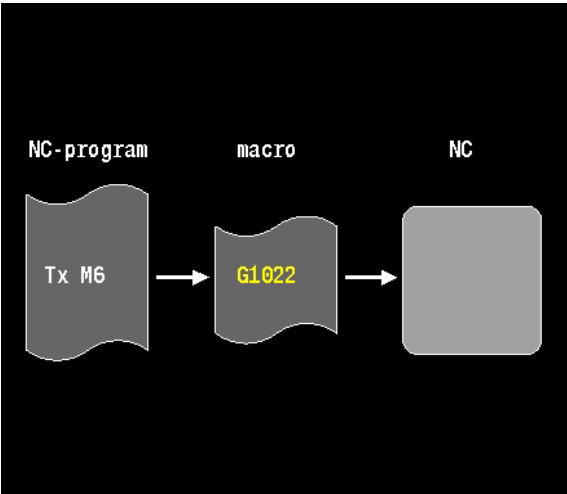
I1=955: activates the T strobe and thus sets the correct PLC marker.

Configuration

CfgSimPosition contains axis coordinates that define the end position of the tool change during mid-program startup.

PLC

PlcTStrobe, TOOL DEF, TOOL CALL, TOOL WAIT



Address	Description
I1=850 I2=	Adapt kinematic model 0 = activate other model 1 = modify data in active model (old description only)
I1=850 I2=1	Write to kinematic model intermittently This allows data to be written to the active kinematic model. The data remains active until the control is switched off. Note: There are two types of description for kinematic models in the MillPlus V600. In the case of the "modern" description with "CfgKinSimpleModel", G1022 I1=850 I2=1 cannot be used, while G339 must be used.

Address	Description
I1=850 I2=0 N5=	<p>Activate other kinematic model. N5= defines the key of the other kinematic model. This model remains active after the control has been switched off. For example, G1022 I1=850 I2=0 N5="KinTestCB" activates the model "KinTestCB".</p> <p>Note: There are two types of description for kinematic models in the MillPlus V600. Models using either description can be activated with G1022 I1=850 I2=0.</p>
I1=850 I2=1 N5=	<p>Location at which the written data is activated</p> <p>There are three basic location types:</p> <ol style="list-style-type: none"> 1) At the start of the kinematic chain (tool side) 2) At the end of the kinematic chain (table side) 3) A specific location in the kinematic chain <p>These three locations are defined in the N5=address as:</p> <p>N5="toolSide" (tool side); at the start of the chain</p> <p>N5="tableSide" (table side); at the end of the chain</p> <p>N5="nnnn" "nnnn"; defines the same string as the key in the Cfg element <CfgTrafoByDir> that is to be modified. The vector in the <location> attribute of this <CfgTrafoByDir> is modified</p>
I1=850 I2=1 I3=	<p>X position</p> <p>Position that is written to the vector element <location.0> of the kinematic element</p>
I1=850 I2=1 I4=	<p>Y position</p> <p>Position that is written to the vector element <location.1> of the kinematic element</p>
I1=850 I2=1 I5=	<p>Z position</p> <p>Position that is written to the vector element <location.2> of the kinematic element</p>
I1=850 I2=1 I6=	<p>Write mode (0=incremental, 1=absolute)</p> <p>In write mode = 0 "incremental", data is added to the existing data.</p> <p>In write mode = 1 "absolute", the existing data is overwritten with new data.</p> <p>Note: The X, Y and Z positions (I3=, I4=, I5=) must always be written at the same time.</p>
I1=950 I2=	<p>Write tool data in the CNC and interpreter</p> <p>Data type</p> <p>0 = Tool number TOOL CALL</p> <p>1 = Calculate tool and tool life</p> <p>2 = Tool TOOL DEF</p> <p>3 = Tool dimensions</p>
I1=950 I2=0-3 I3=	<p>Tool number</p> <p>I3= defines the number (8.2 Format) of the tool</p>
I1=950 I2=0-3 I4=	<p>Tool index</p> <p>I4= defines the tool index (programmed with T2=) of the tool</p>
I1=950 I2=1 I5=	<p>TOOL CALL tool life to NC</p> <p>I5= SQL handle for tool life monitoring</p>
I1=950 I2=0 I6=	<p>All TOOL CALL tool data apart from tool life to NC</p> <p>I6= SQL handle with all tool data</p>
I1=950 I2=3 I6=	<p>M67 tool data to NC</p> <p>I6= SQL handle with M67 tool data</p>
I1=955	<p>Tool change to PLC</p> <p>Prompt to the PLC to execute the requested tool change. Sends PlcTStrobe and sets system data</p>

Address	Description
I1=955 I2=	Tool change mode I2= defines the tool change mode 0 = Unload (TOOL CALL) 1 = Load (TOOL CALL) 2 = Prepare (TOOL DEF) 3 = Reserve (TOOL WAIT) 11 = Calculate (M67)
I1=955 I3=	Tool number in the tool magazine I3= defines the number (5.0 Format) of the tool to be loaded or unloaded
I1=955 I4=	Tool index I4= defines the tool index (programmed with T2=) of the tool to be prepared
I1=955 I2=0-2 I5=	Transfer T strobe to PLC I5=0: do not transfer T strobe I5<>0: T strobe being transferred Special cases: I5=6 46 66: CfgSimPosition of M6 and/or M46 and M66 being activated Tool magazine position I5= defines the number of the magazine position at which the tool to be loaded is located
I1=955 I6=	SQL handle of the tool data table
I1=955 I7=	SQL handle of the tool magazine table (no longer used)
I1=955 I2=0-1 I7 <>0	Interpreter generates key values for CfgSimPosition T0Mxx: end position for tool unload T1Mxx: end position for tool load Simulated positions are activated during mid-program startup
I1=955 N5=	Text displayed in the dialog during mid-program startup

12.7 G1029 Define up to eight PLC values

With the G1029 function, you can transfer up to eight numerical values to the PLC out of sync.

Address description

- ▶ I PLC value
- ▶ I1= PLC value
- ▶ I2= PLC value
- ▶ I3= PLC value
- ▶ I4= PLC value
- ▶ I5= PLC value
- ▶ I6= PLC value
- ▶ I7= PLC value

Application



The parameter In= can only be used if the parameter I(n-1)= is already in use

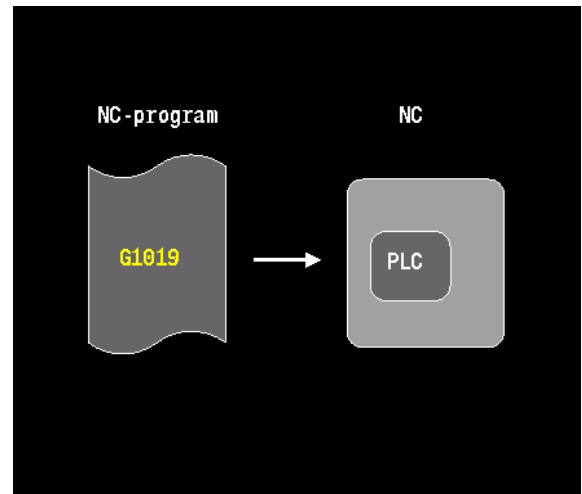
Output to PLC

The values are decoded in the PLC. Used, for example, in the tool change macro: G1029 I1=101 I2=E861 (tool number transfer)

Example

Tool number transfer, e.g. in tool change macro.

```
G1029 I=101 I1=E861
```



13

Changed G-functions

13.1 Description of changed G-functions with respect to version V500-V530

During the development of the new MillPlus version V600, it is endeavored to keep the MillPlus language compatible to former versions. With the new developments although, some new functions have become available and some G-functions and its executions have been changed. In this chapter you will find an overview of the most important changes.

G0..G3_G91

Danger: Other position after incremental linear and rotary axes movements

Cause:

A combination of incremental movements with linear axes and at least one rotary axis is (with active G108 kinematics calculation) executed differently than in former versions.

During calculation of the end position of the linear axes, now the begin position of the linear axes is recalculated to the, by the rotary axis changed, kinematics position. After that, the incremental movement is added.

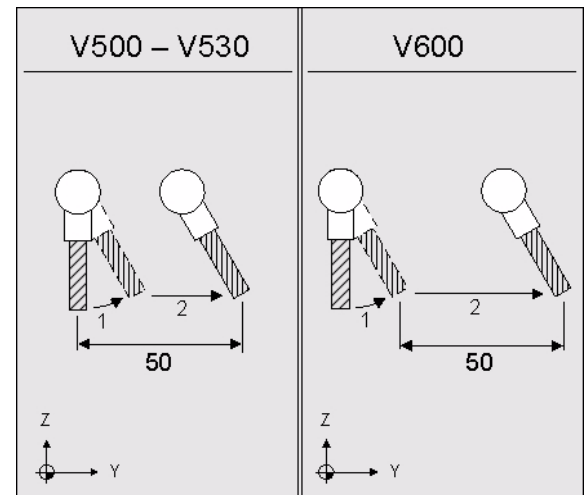
Example V500-V530:

```
..
X0 Y50 Z50 A0 C0
G91
G1 Y50 A30
..
```

Action:

Depending on the application, there are more solutions possible:

- Program the end position with absolute coordinates
- If it is not necessary to execute the linear and the rotary axes in one movement, split the NC-block in two NC-blocks. First a NC-block with the incremental linear axes movements and then a NC-block with the incremental rotary axes movements
- Change the incremental coordinates



G1, G41 und G64

G1_G41_G42_A

Rotary axis programming during G41 or G42 is no longer permitted

Cause:

Rotary axis movements are not possible anymore when tool radius correction is active.

Action:

Change the NC program:

- Switch tool radius correction off temporarily

G1_G64_B1

Angle B1 must have exactly the same direction as the movement

Cause:

Angle B1 must have exactly the same direction as the movement. In former versions, the direction had to be only correct within 180 degrees.

Action:

B1 gives the direction of a line. The definition of angle B1 is based on:

- The + X-axis in the XY- or XZ-plane
- The - Z-axis in the YZ-plane

G1_G64_B1_X

A combination of angle B1 and a coordinate is not permitted anymore

Cause:

A combination of angle B1 with a linear axis X, Y, or Z is not permitted anymore. In former versions this was possible, to indicate the end point of a movement.

Action:

Program the end point with a combination of an angle and a length, or with two main plane coordinates.

G1_P_PN

Combination of P and Pn= is not permitted

Cause:

A combination of points programmed with P and Pn= is no longer permitted. In former versions, this combination was possible.

Action:

Program points with either e.g. P7 P8 P9 P10 or P1=7 P2=8 P3=9 P4=10.

G1_G64_X und Y

The two main plane axes must be programmed, but the tool axis may not be programmed

Cause:

In former versions, in certain cases only one main plane axis needs to be programmed.

Action:

Program both main plane axes, but not the tool axis.

G1_G64_X1 und Y1

The two main plane axes must be programmed, but the tool axis may not be programmed

Cause:

In former versions, in certain cases only one main plane axis needs to be programmed.

Action:

Program both main plane axes, but not the tool axis.

G1_G64_R1=n

A tangential movement must be continuous

Cause:

It is only permitted to program continuous movements ($R1=0$).

In former versions, also discontinuous movements could be programmed.

Example V500-V530:

..
G64
G1 R1=2
G2 I20 J40 X30 Y40
G63
..

Action:

Program a continuous contour ($R1=0$).

G1_G64_J1

The intersection point indicator J1 is replaced by I2

Cause:

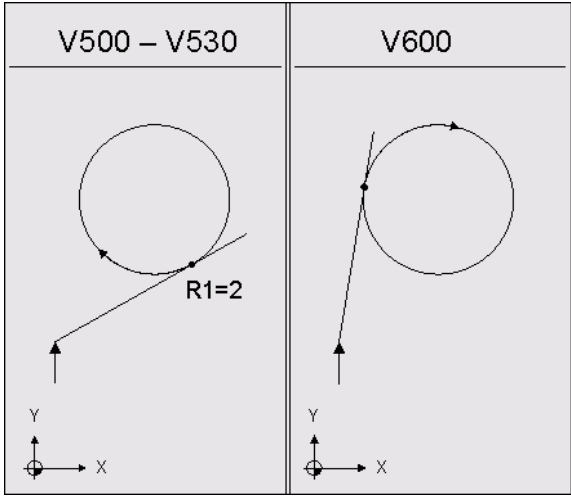
The intersection point indicator J1 is replaced by an intersection point indicator with another function.

I2 must be programmed at the end of the free contour, whereas J1 was programmed at the start of the free contour.

The value of I2 is in most cases identical to the value of J1.

Example V500-V530:

..
G0 X0 Y0 Z0
G64
G1 B1=45 J1=2
G2 I20 J12 X30 Y12
G63
..



Action:

Replace address J1 by address I2 in the block at the end of the free contour part. The value can remain unchanged in most cases.

Check the contour graphically.

Try another value for I2 if necessary and check the contour graphically again.

Note: Unfortunately there is no unambiguous algorithm to obtain the correct value for I2.

Example V600:

```
..
G0 X0 Y0 Z0
G64
G1 B1=45
G2 I20 J12 X30 Y12 I2=2
G63
..
```

G2

G2_G64_K1

The rounding or connecting circle indicator K1 is not available

Cause:

Only continuous movements, which do no intersect with themselves, can be programmed.

In former versions, also discontinuous or itself intersecting movements could be programmed.

Action:

Program a continuous contour that does not intersect itself.

G2_G64_R

Danger: rounding R can be executed differently

Cause:

For a rounding between two linear movements, programmed with endpoints, address R can no longer be used. Use address R2 instead.

Action:

- Replace address R by address R2
- Check the contour graphically

G5

Function G5 is replaced by G305

Cause:

To synchronize CNC and PLC, function G305 must be programmed.

Action:

Replace function G5 by function G305. Adapt the PLC program if necessary.

G6

Function G6 is not available

Cause:

Spline interpolation is not available.

Action:

NC Program cannot be executed with this version.

G7

G7_A6

The incremental definition of G7 is changed

Cause:

The incremental angles for a tilted plane are now defined by adding the incremental value A51=, B51=, C51= to the absolute values A5=, B5=, C5=.

In former versions, the incremental values A6=, B6=, C6= were based on the already active plane.

Action:

The tilted plane must be defined incrementally with A51, B51 and/or C51.

The programmed value is added to the already active value. E.g. if G7 A5=10 B5=10 is active and G7 A51=5 is programmed, the result is G7 A5=15 B5=10.

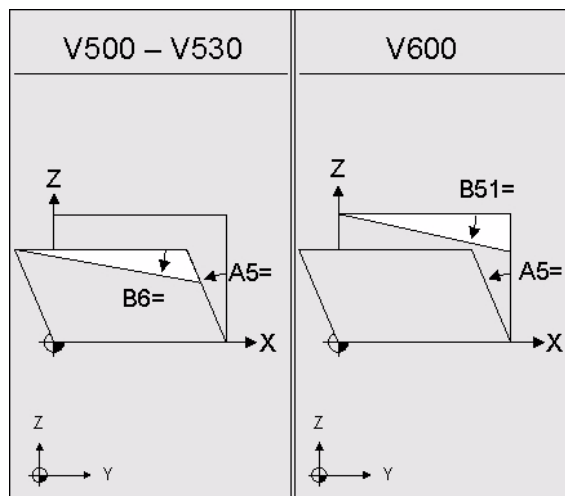
G7_L1=0

Linear axes positioning after G7 L1=0 is changed

Cause:

After G7 without rotary axes positioning (L1=0 or without L1=), the display of the linear axes positions in this version differs from former versions. Programmed movements are executed to other machine positions now.

Only when the rotary axes are positioned corresponding to the G7 tilted plane, the linear axes positions and programming are identical.



Action:

Check all programmed movements between the activation of the G7 tilted plane and the positioning of the rotary axes.

G7_L1=1, L1=2

The rotary axes positions after G7 L1=1 or L1=2 can be different

Cause:

In certain cases, the rotary axes positioning of G7 L1=1 or L1=2 can choose between two possibilities to position the rotary axes. Both possibilities are valid.

This version can select a different combination of rotary axes positions than former versions.

Action:

Try to position the rotary axes towards the wanted positions in the block before the G7-block. In this way it is possible to influence the G7-positioning.

G7_L2

Address L2 is not available

Cause:

The move direction of the rotary axes cannot be programmed. In former versions, it was possible to position rotary axes in two different ways with G7 L2=1 or L2=2. This version always chooses the shortest way when positioning rotary axes.

Example V500-V530:

```
..
G0 A0 C0
G7 C5=30 L2=1
..
```

Action:

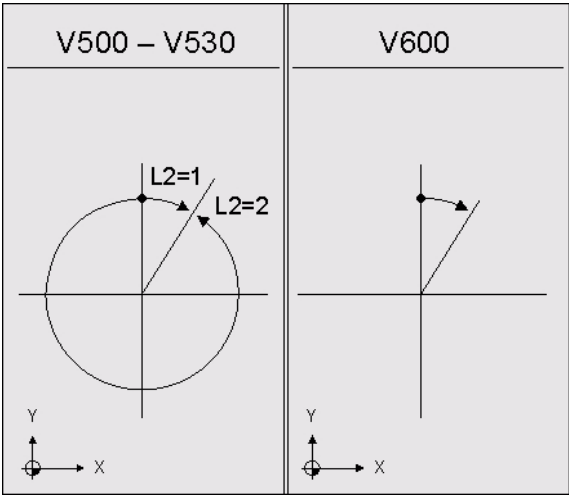
Remove L2 from this block. Try to position the rotary axes towards the wanted positions in the block before the G7-block. In this way it is possible to influence the G7-positioning.

G7_B47

Function G7 B47= is removed

Cause:

The resulting main plane rotation, which is calculated during the tilting of the working plane, cannot be read anymore.



G8

Function G8 is not available

Cause:

Tilting tool orientation is not available.

Action:

NC Program cannot be executed with this version.

G9_B2

Function G9 is not available with polar coordinates

Cause:

The G9 pole can only be programmed with cartesian coordinates.

In former versions, the G9 pole itself could be programmed with addresses B2= and L2= or with angle B1=.

Action:

Change the NC program:

- Replace the polar or angle programming of the pole by programming with cartesian coordinates

G11

G11_B_X

A combination of angle B or B1 and a coordinate is not permitted anymore

Cause:

A combination of angle B1 with a linear axis X, Y, or Z is not permitted anymore. In former versions this was possible, to indicate the end point of a movement.

Action:

Program the auxiliary or end point with a combination of an angle and a length, or with two main plane coordinates.

G11_G91

In this version, the endpoint after an incremental movement after a G11 block with rounding/chamfer, differs from former versions.

Cause:

In former versions, the endpoint after an incremental movement after a G11 block with rounding/chamfer was calculated from the endpoint of the added chamfer or rounding. In this version, the endpoint is calculated from the programmed end point of the G11 block.

Example V500-V530:

```
..
G1 X20 Y20
G11 X80 Y20 R30
G91 Y50
..
```

Action:

Program a new endpoint for the incremental movement.

G14_E

Function G14 E is not available

Cause:

Programming the number of repeats with address E is not possible.

Action:

Change the NC program:

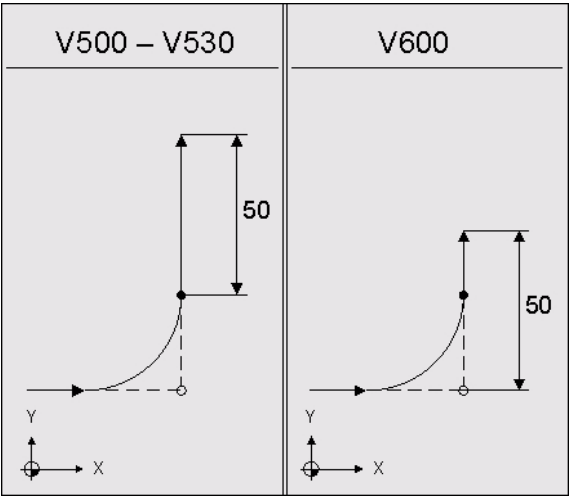
- Replace address E with address J

G26

Function G26 is not available

Cause:

Deactivation of the feed and speed override is not available.



G28

G28_I2

Function G28 I2 is not available

Cause:

A reduction of the path jerk cannot be programmed.

G28_I3

Functions G28 I3 and G28 I4 are not available

Cause:

A stop between two movements (inpod) can no longer be programmed with G28.

Action:

Instead of a stop, a corner accuracy can be programmed with the contour tolerance function G28 I7.

G36

Turning mode is not available

Cause:

The G-functions for turning (33, 36, 96, 228, 302, 356, and 368) and turning cycles (615, 690, 691, 692 and 8xx) are not available.

Action:

NC Program cannot be executed with this version.

G39_G41_L

Function G39 L is not possible during G41 or G42

Cause:

Programming a length offset on the active tool during active tool radius correction, is not possible.

Action:

Change the NC program:

- Switch tool radius correction off temporarily

G40_G91

Danger: an incremental movement after G40 goes to another position

Cause:

The end position of an incremental movement after switching off 2D tool radius correction is now based on the actual position. The actual position is the position that is corrected with the tool radius.

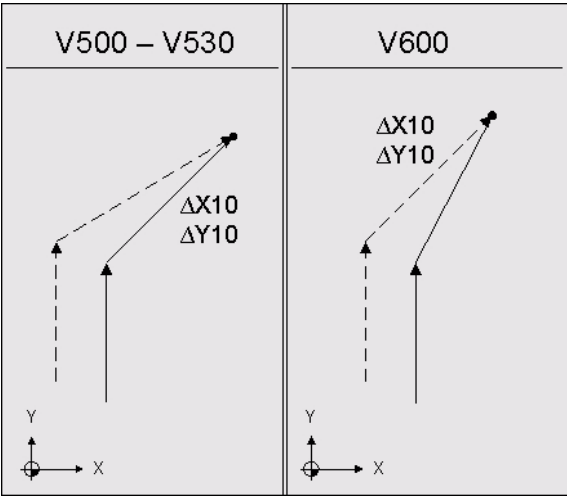
In former versions, an incremental movement was based on the programmed position.

Example V500-V530:

```
..
G41
G91
G1 X10 Y10
G1 X0 Y5
G40
G1 X10 Y10
..
```

Action:

- Program the end position with absolute coordinates
- Change the incremental coordinates



G41-G42

G41-G42_G40

After G40, not programmed axes do not move

Cause:

In former versions, not programmed axes moved as well after G40

Example V500-V530:

G41
..
G1 X11 Y12
G40
G0 Y22

Action:

Also program the axis that must move and that wasn't programmed yet.

G49_E

Function G49 E is not available

Cause:

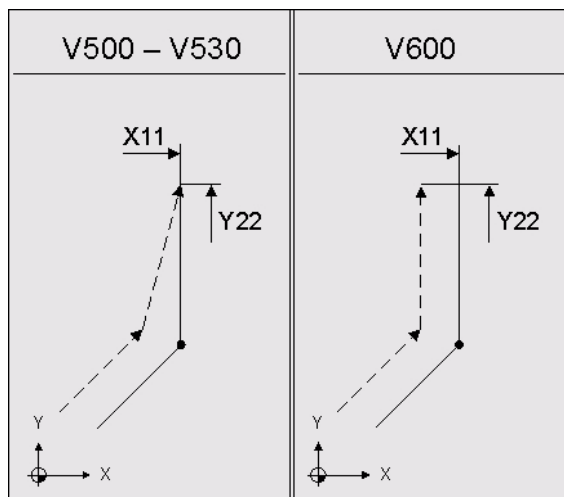
Function G49 jumps or repeats only once and only depending on the measuring tolerance.

In former versions, extra jump conditions and more repeats could be programmed.

Action:

Change the NC program:

- Remove address E
- Program extra jump conditions (G29) or more repeats (G14) in the G29 or G14 block



G54_G41

Transformations during G41 - G44 are not permitted anymore

Cause:

Transformations, such as zero point shifts, mirroring and scaling, are not possible anymore when tool radius correction is active.

Action:

Change the NC program:

- Switch tool radius correction off temporarily
- Accomplish the transformation directly in the programmed coordinates

G61 und G62

G61-G62_G41-G42

Radius correction must be activated directly before the G61 block

Cause:

In former versions, radius correction could be active before the G61 block and both linear and approach movements were executed with radius correction.

Action:

Change the NC program:

- program the approach movement and radius correction in this order in seperate blocks before the G61 block.

G61_B2_Z

Polar programming with tool axis is not available

Cause:

For tangential approach and departure, the tool axis cannot be programmed, in case the main axes are programmed with polar coordinates.

In former versions, it was permitted to program the tool axis with cartesian coordinates.

Action:

Change the NC program:

- Program the main axes with cartesian coordinates
- Program the tool axis movement in a separate NC block

V500 – V530	V600
<pre>.. G41 G1 Z10 G1 Y10 G1 X0 G1 Y0 G61 X0 Y0 I2=1 R10 G3 I25 J0 X30 Y0 G40 ..</pre>	<pre>.. G41 G1 Z10 G1 Y10 G1 X0 G40 Y0 G41 G61 X0 Y0 I2=1 R10 G3 I25 J0 X30 Y0 G40 ..</pre>

G61_I1=0

Approach movements are always executed with feed

Cause:

The first movement of G61 tangential approach and the second movement of G62 tangential departure are always executed with feed. A rapid movement (I1=0, default) is not permitted anymore.

In former versions, these movements could be programmed with rapid (I1=0).

Action:

- Remove I1=0. All movements will be executed with feed
- When I1 is not programmed, note that all movements are with feed now and not with rapid

G61_I2=0

In certain cases, the approach movement with I2=0 doesn't fit

Cause:

An approach movement with line and tangent circle (I2=0) generates an error message, when:

- The distance between the actual position and the approach circle is smaller than the cutter radius
- The start point lies within the approach circle

In former versions, the approach method was automatically transformed into a movement with a quarter circle (I2=1).

Action:

Change the approach method into a quarter circle movement (I2=1).

G63 und G64

G64_G1_G2_G3_X und Y

The two main plane axes must be programmed, but the tool axis may not be programmed

Cause:

In former versions, in certain cases only one main plane axis needs to be programmed.

Action:

Only program both main plane axes.

G64-G63_G250-G269

New "free contour" G-functions G250-G269.

Cause:

In former versions, a free contour could be programmed with G1, G2 and G3 movements. In this version, the functions G250-G269 must be used.

Action:

Change the NC program:

- Program the functions G250-G269 instead of the functions G1, G2 and G3

G67

Function G67 is not available

Cause:

The tool length correction is always performed in the direction of the real position of the tool.

In former versions, it was possible to define with G67 that the tool length was corrected in the positive direction of the by the main plane defined tool axis. With G67 the rotation direction of circle segments was also inverted and the tool radius correction was changed from left into right or vice versa.

Action:

This NC program probably cannot be executed with this version.

Danger: after you removed G67 from this NC program, collisions may occur. To prevent collisions:

- Check the tool length correction
- Invert the rotation direction of circle segments from G2 into G3, or vice versa
- Change the tool radius correction from G41 into G42, or vice versa
- Invert the milling direction of the cycles
- Check the NC program graphically

G73_G92

The order of G73 mirroring and G92 rotation is changed

Cause:

In this version, the order is 1) rotating 2) mirroring.

Example V500-V530:

```
..
G1 X0 Y0
G73 X-1
G1 X100 Y0
G92 B4=45
G1 X100 Y0
..
```

Action:

Change the NC program:

- Change the rotation angle for G92.

Example V600:

```
..
G1 X0 Y0
G73 X-1
G1 X100 Y0
G92 B4=-45
G1 X100 Y0
..
```


G74

G74_K2

Function G74 K2 is not available

Cause:

Absolute positioning G74 with connection circle to the next movement is removed.

Action:

Remove K2 and, if present, K2= from the G74 block. This changes the executed path and makes it slower. Check the changed path on collisions.

G74_X_X1

Incremental programming relative to a machine position is not available

Cause:

In a G74 block, a machine position (e.g. X1=) cannot be programmed combined with an axis position (e.g. X).

In former versions, it is possible to move to an incremental position relative to a machine position.

Action:

Change the NC program:

- Program the position absolute with G74

G77_G91

Danger: Other position for incremental contour instruction after G77

Cause:

The end position of an incremental contour instruction (G0, G1, G2, G3) after a G77 bolt hole circle is now based on the actual position.

In former versions, an incremental movement was based on the programmed bolt hole circle center point.

Remark: The execution of an incremental cycle call (G77 or G79) is still based on the previous programmed bolt hole circle center point.

Action:

- Program the end position with absolute coordinates
- Change the incremental coordinates

G79

G79_G41

Cycles cannot be executed anymore, when G41 - G44 is active

Cause:

Cycles cannot be executed anymore, when tool radius correction is active.

In former versions, tool radius correction was switched off in the cycle.

Action:

Change the NC program:

- Before the cycle call, switch tool radius correction off by G40

G79_B1 und L1

Address B1 for rotation of the cycle is changed into address A5

Cause:

The rotation angle for pockets or slots is now programmed with the A5 address.

In former versions, this angle was programmed with the B1 address. This was however confusing with polar programming.

Action:

Change the B1 address into the A5 address.

G79_B2_L2

Polar programming is not available

Cause:

Polar programming with addresses B1 and L1 or with B2 and L2 is not available.

Action:

Change the NC program:

- Change the polar coordinates into cartesian coordinates

G84

G84_I

Function G84 I is not available

Cause:

The deceleration after tapping cannot be programmed.

In former versions, the deceleration could be programmed as a number of revolutions.

Action:

Remove the I address from the G84 block. The configured deceleration is taken now.

Check the deceleration as defined in the configuration.

G84_I1=0

With G84 I1=0, M19 is always done

Cause:

At the start of tapping without interpolation G84 I1=0, a spindle positioning with M19 is done.

In former versions, the spindle was not positioned first.

G98

G98

Graphic view is defined differently

Cause:

G98 and G195 define the graphic blank form now. The graphic view is derived automatically as an offset to the blank form.

In former versions, G98 or G195 only defined the graphic view. The blank form (G99) was defined in a separate NC block.

Action:

Change the NC program:

- If a separate block with the G99 blank form is defined, remove the block with G98 or G195
- Adapt the programmed values, so that blank form and graphic view are correct

G98_B

Graphic view cannot be rotated

Cause:

The graphic view cannot be set up rotated.

In former versions, G98 or G195 could set up a rotated graphic view with the addresses B, B1= and/or B2=.

Action:

Change the NC program:

- Remove the addresses B, B1= and/or B2=

Set the graphic view as wished by means of the soft key operation.

G106

Function G106 is not available

Cause:

The function G108 "Kinematic calculation" cannot be switched off anymore by G106. Therefore, the tool length correction is always calculated in the real direction of the tool.

When G106 is active, the tool length correction was according to the defined plane.

This difference can cause a collision, when:

- The tool is in a different direction as defined by the plane (G17, G18, G19)
- An angular head mill is active

Action:

Change the NC program:

- Remove G106

Check if the tool is in the direction as defined by the plane.

G108_I1=2

Function G108 I1=2 is not available

Cause:

Function G108 "Kinematic calculation" always calculates the head and the tool.

In former versions, I1=2 defined that the tool was not calculated.

Action:

Danger: if I1=2 is removed from G108, there can be collision danger.

Check the program on possible collisions.

G126

G126

Tool lifting is defined in the tool table

Cause:

A tool will only lift off, if this is defined in the tool table.

In former versions, lift off was only possible for certain G functions (e.g. tapping).

Action:

Check the NC program and the tool table:

- For all used tools, it must be defined in the column "LIFTOFF" if lift off is permitted or not

G126_I1

G126 I1= or I2= or I3= are not available

Cause:

Tool lift off is only activated with intervention.

In former versions, lift off could be activated by:

- The PLC (I1=1)
- Intervention (I2=1)
- Errors (I3=1)

Action:

Change the NC program:

- A programmed I2=1 can be removed. The behavior is then identical to former versions
- With the other address combinations, this NC program cannot be executed in the same way as in former versions

G136 und G137

Functions G136 and G137 are not available

Cause:

The axes configuration cannot be switched.

Action:

NC Program cannot be executed with this version.

G141

G141_G93

If G92/G93 with rotary axes is active, G141 is erroneous

Cause:

Check if G92 or G93 with rotary axes is active.

An active programmable zero offset in one of the rotary axes, causes 3D tool correction to position the rotary axes wrong.

Action:

If G92 or G93 is active with rotary axes, the NC program must be changed:

- Replace G92 or G93 by G54. The programmed axes values of G92 or G93 must be added to the already active axes values of G54

G141_I7

Function G141 I7 is not available

Cause:

The accuracy during 3D tool correction cannot be programmed in the NC program.

Action:

Remove I7 from the G141 block. The accuracy during G141 is now defined by machine parameter: NCchannel - ChannelSettings - CH_NC - CfgTCPM - Tolerance.

G141_L2

Danger: address L2 is only effective for 'RollOver' axes

Cause:

The shortest distance criterion L2= is only compatible to former versions, if a rotary axis is configured as a 'RollOver' axis.

Action:

The rotary axis must be configured as 'RollOver' axis. The corresponding attribute 'shortestDistance' can, if desired, be configured as well. Within G141 however, the configuration of 'shortestDistance' is overruled by L2.

G145

G145_E

Function G145 E is not available

Cause:

The measuring device status cannot be read with the E address.

Action:

Change the NC-Program:

- Replace address E by one or more of the addresses O1=, O2=, O3= or O4=

G145_I3

Function G145 I3 is not available

Cause:

Guarding of the measuring probe status (e.g. for Laser) cannot be switched off anymore.

Action:

Change the NC program:

- Replace address I3= with address O3=. With G145 O3= the measuring device status is read and no error message is given.

G148_I1=3

Function G148 I1=3 is not available

Cause:

PLC information on whether the table probe or laser measuring device is activated, can no longer be read with G148 I1=3.

Action:

Change the NC program:

- Replace G148 I1=3 to G328 to read the concerning PLC markers

G149

G149_T_E

E for tool status is changed into I1= and I2=

Cause:

Read or write of the tool status is done with a combination of Tnn, I1= "Tool lock" and I2= "Tool status". The function of the tool status in the tool table has been changed.

In former versions, the tool status was read or written with the E address.

Action:

Replace the E address by a combination of I1= und I2=. Note the new function of the tool status in the tool table.

G149_T_M1

With G149 T.. M1=, the actual tool life CUR_TIME is read.

Cause:

The function of the M1 address in G149 has changed: actual tool life CUR_TIME

In former version, with M1 the rest tool life was meant.

Action:

Change the NC program:

- Read the maximum tool life TIME1 with G321 T.. I1=13 E..
- Read the actual tool life CUR_TIME with G149 T.. M1=..
- Calculate the new actual rest tool life: actual rest tool life = maximum tool life TIME1 - actual tool life CUR_TIME

G150_T_M1

With G150 T.. M1=, the actual tool life CUR_TIME is written.

Cause:

The function of the M1 address in G150 has changed: actual tool life CUR_TIME.

In former versions, with M1 the rest tool life was meant.

Action:

Change the NC program:

- Read the maximum tool life TIME1 with G321 T.. l1=13 E..
- Calculate the new actual tool life: $\text{actual tool life CUR_TIME} = \text{maximum tool life TIME1} - \text{rest tool life}$
- Write the new actual tool life CUR_TIME with G150 T.. M1=..

G151 und G152

Functions G151 - G152 are replaced by G270 - G273

Cause:

Limit of travelling distance must be activated with G270 - G273 Limit plane.

Action:

Replace G151 by G270 and G152 by a combination of G271, G272 and/or G273.

G182

Function G182 is not available

Cause:

Cylinder interpolation is not available.

Action:

NC Program cannot be executed with this version.

G199

Graphic contour is removed

Cause:

A graphic contour can no longer be programmed. A graphical blank form can only be programmed cubic (G99).

In former versions, any graphic contour or blank form could be programmed with G199, G198, G197 and G196.

Action:

This graphic contour or blank form cannot be drawn with this version.

Change the NC program:

- Remove the G functions G199, G198, G197, G196 and all G1, G2, G3 etc. contour description blocks in between

G200 - G208

Functions G200-G208 are replaced by G280-G286

Cause:

The pocket cycles G200-G208 have been replaced by the contour milling cycles G280-G286.

Action:

Change the NC program:

- Replace functions G200-G208 by functions G280-G286

G217 und G218

Functions G217 and G218 are not available

Cause:

Angular head tools cannot be activated.

Action:

NC Program cannot be executed with this version.

G231

Function G231 is not available

Cause:

Interpolation between a spindle and an axis is not available.

Action:

NC Program cannot be executed with this version.

G241

Function G241 is replaced by G242

Cause:

Contour check must be activated with G242.

Action:

Replace G241 by G242. G242 adapts the executed contour for undercuts. G241 only gave an error.

G318

Function G318 is not available

Cause:

Reading pallet or job data is not available.

Action:

NC Program cannot be executed with this version.

G319_I2=1

Function G319 I2=1 is not available

Cause:

The actual data cannot be read.

Action:

NC Program cannot be executed with this version.

G320_I1

Die folgende Programmierung von G320 I1= ist nicht verfügbar:

- I1=4 Angle of rotation A-axis
- I1=5 Angle of rotation B-axis
- I1=6 Angle of rotation C-axis
- I1=7 Pole coordinate X-axis
- I1=8 Pole coordinate Y-axis
- I1=9 Pole coordinate Z-axis
- I1=14 Feed movement
- I1=15 Rapid movement
- I1=16 Positioning logic
- I1=17 Acceleration reduction
- I1=18 Contour tolerance
- I1=21 Palette zero point shift in X-axis
- I1=22 Palette zero point shift in Y-axis
- I1=23 Palette zero point shift in Z-axis
- I1=24 Palette zero point shift in A-axis
- I1=25 Palette zero point shift in B-axis
- I1=26 Palette zero point shift in C-axis
- I1=41 Zero point shift in X-axis
- I1=42 Zero point shift in Y-axis
- I1=43 Zero point shift in Z-axis
- I1=44 Zero point shift in A-axis
- I1=45 Zero point shift in B-axis
- I1=46 Zero point shift in C-axis
- I1=47 Angle of rotation
- I1=62 Actual tool length
- I1=63 Actual tool radius
- I1=64 Actual tool corner radius
- I1=65 Actual tool orientation
- I1=66 Projected actual spindle position angle
- I1=67 Total shift in X
- I1=68 Total shift in Y
- I1=69 Total shift in Z
- I1=71 Programmed status
- I1=72 Programmed status

- I1=73 Programmed distance
- I1=74 Kinematic position of A-rotary axis
- I1=75 Kinematic position of B-rotary axis
- I1=76 Kinematic position of C-rotary axis
- I1=77 Distance to positive software end switch in X
- I1=78 Distance to positive software end switch in Y
- I1=79 Distance to positive software end switch in Z
- I1=80 Distance to negative software end switch in X
- I1=81 Distance to negative software end switch in Y
- I1=82 Distance to negative software end switch in Z
- I1=83 G108 offset in X-axis
- I1=84 G108 offset in Y-axis
- I1=85 G108 offset in Z-axis
- I1=86 G154 offset in X-axis
- I1=87 G154 offset in Y-axis
- I1=88 G154 offset in Z-axis
- I1=89 G218 offset in X-axis
- I1=90 G218 offset in Y-axis
- I1=91 G218 offset in Z-axis
- I1=92 G218 rotation (space angle) in A-direction
- I1=93 G218 rotation (space angle) in B-direction
- I1=94 G218 rotation (space angle) in C-direction

Action:

See function G1010.

G321

G321_I1

The following programming of G321 is not available:

- I1=6 G Graphics
- I1=7 Q3 Type
- I1=15 M2= Tool life monitoring
- I1=17 B1= Breakage monitoring
- I1=18 L1= First extra length
- I1=19 R1= First extra radius
- I1=20 C1= First extra corner radius
- I1=21 L2= Second extra length
- I1=22 R2= Second extra radius
- I1=23 C2= Second extra corner radius
- I1=28 Q5= Breakage monitoring cycle (0-9999)
- I1=29 O Tool orientation
- I1=30 C6= Cutting width

Action:

See function G1010.

G321_I1=14

With G321 I1=14 E.. the actual tool life CUR_TIME is read.

Cause:

The function of I1=14 has changed: actual tool life CUR_TIME.

Action:

Change the NC program:

- Read the maximum tool life TIME1 with G321 T.. I1=13 E..
- Read the actual tool life CUR_TIME with G321 T.. I1=14 E..
- Calculate the new actual tool rest tool life: actual rest tool life = maximum tool life TIME1 - actual tool life CUR_TIME

G321_I2

Function G321 I2= is removed

Cause:

Data of the active spare tool cannot be requested directly anymore.

Action:

Data of an active spare tool can be read in two stages:

- 1 Program G319 I1=3 I2=1 Enn to read the active spare tool number
- 2 Program G321 T=Enn I1=... to read the requested data of this spare tool

G322

G322_N1

Address N1= is removed

Cause:

Address N1= was replaced by N5=.

Action:

Program G322 N5=.

G322_E

Address E is removed

Cause:

Address E was replaced by O1=.

Action:

Program G322 O1=

.
.

G323

G323_O3

The function of address O3= has changed

Cause:

In former versions, with address O3= the number of the E-parameter was programmed in which the safety distance was written. In this version, with address O3= the number of the first E-parameter of the cycle definition is written.

Action:

The safety distance is written in the second E-parameter from the number of the O3 address. Change the E-parameter number in the NC program that means the safety distance.

G323_O4

The function of address O3= has changed

Cause:

In former versions, with address O4= the number of the E-parameter was programmed in which the retract distance was written. In this version, with address O4= the number of the last E-parameter of the cycle definition is written.

Action:

The retract distance is written in the third E-parameter from the number of the O3 address. Change the E-parameter number in the NC program that means the retract distance.

G324_I1

The following functionality of G324 is removed:

► I1= G-group (1,2,usw.)

- I1=6 G81, G83, G84, G85, G86, G87, G88, G89, G98.
- I1=18 G61, G62
- I1=21 G9
- I1=29 G106 G108

G326

The following functionality of G326 is removed:

- I1=2 Position to reference point
- Reading out during graphical simulation

G327

The following programming of G327 is removed:

- I1=4 Test run

G328

The programming and the interface to the PLC of G328 and G338 have changed

Cause:

The functions G328 and G338 have changed:

- PLC interface is changed
- Addresses I1 and N1 are replaced by N5
- Address E of G328 is changed into O1 or O2
- In this version, function G338 does no longer check whether the IPLC-signal, defined by N5=, is enabled by the IPLC

Action:

Change the NC program:

- Replace addresses I1 and N1 by N5
- For G328 replace address E by O1 or O2

G329

Read/write kinematical correction is changed

Action:

See the changed description of function G329.

G330

Function G330 is replaced by SQL functions

Cause:

Read point memory must be programmed with SQL functions.

Action:

See description of function G1010.

G331**G331_T_I1=14**

With G331 T.. I1=14 E.. the actual tool life CUR_TIME is written.

Cause:

The function of I1=14 has changed: actual tool life CUR_TIME.

In former versions, the rest tool life was meant with I1=14.

Action:

Change the NC program:

- Read the maximum tool life TIME1 with G321 T.. I1=13 E..
- Calculate the new actual tool life: actual tool life CUR_TIME = maximum tool life TIME1 - rest tool life
- Write the new actual tool life CUR_TIME with G331 T.. I1=14 E..

G331_I1

The following programming of G331 is not available:

- I1=6 G Graphics
- I1=7 Q3 Type
- I1=15 M2= Tool life monitoring
- I1=17 B1= Breakage monitoring
- I1=18 L1= First extra length
- I1=19 R1= First extra radius
- I1=20 C1= First extra tool corner radius
- I1=21 L2= Second extra length
- I1=22 R2= Second extra radius
- I1=23 C2= Second extra corner radius
- I1=28 Q5= Breakage monitoring cycle (0-9999)
- I1=29 O Tool orientation
- I1=30 C6= Cutting width

Action:

See function G1010.

G350 und G351

Functions G350 and G351 are replaced by G1016

Cause:

The functions write to window or hard disk must be activated with G1016.

Action:

Replace G350 and G351 by G1016.

G364

Function G364 is not available

Cause:

Calculating an intersection point between two elements is not available.

G606

Function G606 is not available

Cause:

TT Calibration is not available.

G607

Function G607 is not available

Cause:

TT Measuring tool length is not available.

G608

Function G608 is not available

Cause:

TT Measuring tool radius is not available.

G609

Function G609 is not available

Cause:

TT Measuring length and radius is not available.

G610

Function G610 is not available

Cause:

TT Tool breakage control is not available.

G611

Function G611 is not available

Cause:

TT Measuring turning tools is not available.

G615

Function G615 is not available

Cause:

Laser: Measuring turning tools is not available.

G631

Function G631 is not available

Cause:

Measure position of inclined plane is not available.

G640

Function G640 is not available

Cause:

Locate table rotation center is not available.

G642

Function G642 is not available

Cause:

Laser: Temperature compensation is not available.

G690

Function G690 is not available

Cause:

Unbalance calibration is not available.

G691

Function G691 is not available

Cause:

Measure unbalance is not available.

G692

Function G692 is not available

Cause:

Unbalance checking is not available.

Symbols

& ... 68

Numerics

3D Tool Correction ... 261

A

About these instructions ... 19
 Absolute Position Approach ... 213
 Absolute Programming ... 240
 Activate Cylinder Interpolation ... 294
 Activate Geometric Calculations ... 194
 Activate Pallet Zero Point Shift ... 181
 Activate Tool Exchange in PLC ... 487
 Activate Zero Point Shift ... 184
 Added functions ... 4, 346
 And ... 69
 AndAlso ... 70
 Angle Measurement ... 381
 Angle Measurement 2 Holes ... 402
 Arithmetic operators ... 54
 Axis configurations ... 46
 Axis configurations on machine tools ... 46

B

Back-Boring ... 458
 Basic tapping with chip breaking ... 150
 Begin Contour Milling ... 335
 Begin contour pocket description ... 335
 Block number N ... 42
 Bolt Hole Circle ... 216
 Boring ... 232, 449

C

Call ... 78
 Cancel Cylinder Interpolation ... 292
 Cancel G152 ... 283
 Cancel G54-G59 Zero Point Shift ... 183
 Cancel Geometric Calculations ... 193
 Cancel Mirror Image and Scaling ... 210
 Cancel Pallet Zero Point Shift ... 180
 Cancel Tool Radius
 Compensation ... 157
 Cancellation of zero point shift ... 180
 Cartesian coordinates ... 47
 Change Tool or Zero Offset
 Values ... 280
 Changes compared with V5xx ... 4
 Checking on Tolerances ... 173
 Circle Measurement Inside ... 397
 Circle Measurement Inside (CP) ... 407
 Circle Measurement Outside ... 394

Circular Counter-Clockwise ... 106
 Circular CW ... 101
 Circular Pocket Finishing ... 467
 Circular Pocket Milling ... 238, 456
 Contour Cycle Call ... 291
 Contour Data Definition ... 337
 Contour Definition Program ... 336
 Contour Finishing ... 342
 Contour Milling Cycles ... 327
 Contour Pilot Drilling ... 338
 Contour Pre-Calculation
 OFF ... 300
 On ... 301
 Contour Programming ... 302
 Contour Roughing ... 340
 Coordinate system ... 46
 Corner Inside Measurement ... 388
 Corner Outside Measurement ... 386
 Correct Workpiece Zero Point
 OFF ... 286
 ON ... 287
 Creating a part program ... 43
 Cycle Call ... 221

D

Datum Inside Rectangle ... 392
 Datum Outside Rectangle ... 390
 Datums ... 44
 Deep-Hole Drilling ... 225, 440
 Deep-hole drilling with additional chip break ... 443
 Define Pole Position ... 122
 Define up to Eight PLC Values ... 490
 Define up to Two PLC values ... 486
 Definition of Lower Limit Plane ... 319
 Definition of Upper Limit Plane ... 321
 Disable Feed/Speed Override ... 143
 Disables Limit
 Planes ... 317
 Drilling/Centering ... 223
 Drilling/Centring ... 438
 Dwell Time ... 107

E

E parameters ... 51
 Edit Function for SQL tables ... 472
 Enable Feed/Speed Override ... 142
 Enables Defined
 Limit Planes ... 318
 End Contour Milling ... 334
 End Graphic Model Description ... 298
 ES parameters ... 53
 Export Formatted Text and E
 Parameter ... 476

F

F ... 22
 F function ... 22
 F1= ... 23, 24
 F2= ... 24
 F3= ... 22, 24
 F5= ... 24
 F6= ... 24
 Face Turning ... 421
 Feed in mm/min (inch/min) ... 248
 Feed in mm/rev (inch/rev) ... 250
 Fixture datum ... 45
 Format of words with path or angle information ... 41
 Free Chamfer ... 314
 Free Circular Movement CCW ... 310
 Free Circular Movement, CCW,
 Tangential ... 313
 Free Circular Movement, CW ... 308
 Free Circular Movement, CW,
 Tangential ... 312
 Free Contour Selection ... 316
 Free Linear Movement ... 307
 Free Linear Movement,
 Tangential ... 311
 Free Rounding ... 315
 Functions, not available anymore ... 4

G

G0 ... 94
 G1 ... 97
 G1010 ... 472
 G1016 ... 476
 G1017 ... 479
 G1018 ... 483
 G1019 ... 486
 G1022 ... 487
 G1029 ... 490
 G11 ... 125
 G125 ... 256
 G126 ... 257

G14 ... 131	G29 ... 148	G64 ... 194
G141 ... 261	G3 ... 106	G7 ... 108
G145 ... 268	G300 ... 347	G70 ... 208
G148 ... 272	G303 ... 348	G700 ... 421
G149 ... 274	G305 ... 349	G71 ... 209
G150 ... 280	G31 ... 150	G72 ... 210
G151 ... 283	G319 ... 350	G73 ... 211
G152 ... 284	G320 ... 351	G730 ... 424
G153 ... 286	G321 ... 354	G74 ... 213
G154 ... 287	G322 ... 356	G740 ... 426
G17 ... 133	G323 ... 357	G741 ... 429
G174 ... 289	G324 ... 358	G77 ... 216
G179 ... 291	G326 ... 360	G771 ... 430
G18 ... 135	G327 ... 362	G772 ... 432
G180 ... 292	G328 ... 363	G773 ... 434
G182 ... 294	G329 ... 365	G777 ... 436
G19 ... 137	G331 ... 368	G78 ... 219
G195 ... 297	G338 ... 370	G781 ... 438
G196 ... 298	G339 ... 371	G782 ... 440
G2 ... 101	G37 ... 153	G783 ... 443
G22 ... 138	G380 ... 373	G784 ... 445
G23 ... 140	G39 ... 154	G785 ... 447
G240 ... 300	G4 ... 107	G786 ... 449
G242 ... 301	G40 ... 157	G787 ... 451
G25 ... 142	G41 ... 158	G788 ... 453
G251 ... 307	G42 ... 162	G789 ... 456
G251-G269 ... 302	G43 ... 164	G79 ... 221
G252 ... 308	G44 ... 166	G790 ... 458
G253 ... 310	G45 ... 167	G794 ... 461
G26 ... 143	G46 ... 170	G797 ... 463
G261 ... 311	G49 ... 173	G798 ... 465
G262 ... 312	G50 ... 175	G799 ... 467
G263 ... 313	G51 ... 180	G8 ... 117
G265 ... 314	G52 ... 181	G81 ... 223
G266 ... 315	G53 ... 183	G83 ... 225
G269 ... 316	G54..G59 ... 184	G84 ... 228
G27 ... 145	G61 ... 188	G85 ... 230
G270 ... 317	G62 ... 191	G86 ... 232
G271 ... 318	G620 ... 381	G87 ... 234
G272 ... 319	G621 ... 384	G88 ... 236
G273 ... 321	G622 ... 386	G89 ... 238
G275 ... 323	G623 ... 388	G9 ... 122
G276 ... 324	G626 ... 390	G90 ... 240
G277 ... 325	G627 ... 392	G91 ... 242
G28 ... 146	G628 ... 394	G92 ... 244
G280 ... 334	G629 ... 397	G93 ... 246
G280-G286 ... 327	G63 ... 193	G94 ... 248
G281 ... 335	G631 ... 400	G95 ... 250
G282 ... 336	G633 ... 402	G97 ... 251
G283 ... 337	G634 ... 404	G98 ... 252
G284 ... 338	G636 ... 407	G99 ... 253
G285 ... 340	G638 ... 410	General programming information ... 40
G286 ... 342	G639 ... 413	GoTo ... 79

Graphic Material Definition ... 253
 Graphic Window Definition ... 252, 297

H

High-level language ... 67

I

If ... 80
 Inch
 Programming ... 208
 Incremental coordinates ... 48
 Incremental Programming ... 242
 Instructions ... 77
 Introduction ... 18
 Is ... 71
 IsNot ... 72

J

Jump Function ... 148

K

Key-Way Finishing ... 465
 Key-Way Milling ... 236, 453

L

Lifting Tool on Intervention
 OFF ... 256
 ON ... 257
 Like ... 73
 Limiting the Traverse Ranges ... 284
 Linear Chamfer Rounding Cycle ... 125
 Linear Measuring Movement ... 268
 Logical operators ... 62

M

M function ... 26
 M0/M1 program stop ... 26
 M19 Oriented spindle stop ... 32
 M19 with Programmable
 Direction ... 348
 M3/M4/M5 spindle ON clockwise/counterclockwise or spindle stop ... 27
 M30 End of part program ... 33
 M41/M42/M43/M44 Selecting the
 spindle speed range ... 34
 M6 Automatic tool change ... 27
 M66 Executing an automatic tool
 change ... 29
 M67 Changing the tool data ... 30
 M7/M8/M9/M13/M14 Coolant no. 2 /
 no. 1 on/off ... 31
 Machine datum ... 44
 Machining and Positioning
 Cycles ... 418

Main Plane XY, Tool Z ... 133
 Main Plane XZ, Tool Y ... 135
 Main Plane YZ, Tool X ... 137
 Mathematical functions ... 56
 Measure Inclined Plane ... 400
 Measurement Center 4 Holes ... 404
 Measuring a Circle ... 170
 Measuring a Point ... 167
 Measuring Cycles ... 378
 Metric
 Programming ... 209
 Milling Operation ... 153
 Mirror Image and Scaling ... 211
 Mm or inches ... 41
 Modified functions (incompatible) ... 4
 Multipass Milling ... 424

N

Not ... 74

O

One-point geometry ... 126
 Operation on Circle ... 436
 Operation on Grid ... 434
 Operation on Line ... 430
 Operation on Quadrangle ... 432
 Operators ... 54, 67
 Or ... 75
 OrElse ... 76

P

Pallet datum ... 45
 Part programs ... 40
 Pocket Finishing ... 463
 Pocket Milling ... 234, 451
 Point Definition ... 219
 Polar coordinates ... 48
 Position Measurement ... 384
 Positioning Functions ... 146
 Processing Measuring Results ... 175
 Program blocks ... 42
 Program Call ... 140
 Program datum (W) ... 45
 Program Error Call ... 347
 Program identifier ... 43
 Program words ... 40
 Protection Zones ... 373

R

Rapid Traverse ... 94
 Read Actual G Data ... 351
 Read Actual Position ... 360
 Read Actual Technology Data ... 350

Read Cycle Data ... 357
 Read G Group ... 358
 Read IPLC Marker or I/O ... 363
 Read Machine Constant
 Memory ... 356
 Read Measure Probe Status ... 272
 Read NC System Data ... 483
 Read Offset from Kinematic
 Model ... 365
 Read Operation Mode ... 362
 Read Tool Data ... 354
 Read Tool or Zero Offset Values ... 274
 Reaming ... 230, 447
 Reference point ... 44
 Relational operators ... 61
 Repeat Function ... 131
 Reset Positioning Functions ... 145

S

S function ... 25
 Select Case ... 81
 Sequence of operators in the
 evaluation ... 63
 Specific G Codes for Macros ... 346
 Spindle Speed ... 251
 Storage of programs ... 43
 Structure of a part program ... 43
 Subprogram Call ... 138
 Synchronize CNC and PLC ... 349

T

T Function Tool Table ... 35
 Tangential Approach ... 188
 Tangential Exit ... 191
 Tapping ... 228, 445
 Tapping with Chip Breaking ... 150
 Tapping, Interpolated ... 461
 Thread Milling Inside ... 426
 Thread Milling Outside ... 429
 Tilting Tool Orientation ... 117
 Tilting Working Plane ... 108
 Tool change ... 36
 Tool life monitoring ... 37
 Tool Measuring Cycles for Laser
 Measurements ... 376
 Tool Measuring Cycles for Tool Touch
 Probe Measuring Systems ... 377
 Tool Offset Change ... 154
 Tool Radius Compensation Past End
 Point ... 166
 Tool Radius Compensation to End Point
 G43 ... 164
 Tool Radius Compensation, Left ... 158

- Tool Radius Compensation,
Right ... 162
- Tool Retract Movement ... 289
- Touch Probe Calibration ... 413
- Touch Probe Calibration on Ball ... 410
- Trigonometric functions ... 59
- Turning Cycles ... 470
- Two-line geometry ... 126

U

- Using tools in the program ... 35

W

- While ... 82
- Write IPLC Marker or I/O ... 370
- Write NC System Data ... 479
- Write Offset in Kinematic Model ... 371
- Write Tool Data ... 368

Z

- Zero Point Shift Abs./Rotation ... 246
- Zero Point Shift Incr./Rotation ... 244
- Zoning Planes:
 - Disable ... 323
 - Enable ... 324
- Zoning Planes: Define ... 325

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH

Dr.-Johannes-Heidenhain-Straße 5

83301 Traunreut, Germany

☎ +49 (8669) 31-0

FAX +49 (8669) 5061

E-mail: info@heidenhain.de

Technical support FAX +49 (8669) 32-1000

Measuring systems ☎ +49 (8669) 31-3104

E-mail: service.ms-support@heidenhain.de

TNC support ☎ +49 (8669) 31-3101

E-mail: service.nc-support@heidenhain.de

NC programming ☎ +49 (8669) 31-3103

E-mail: service.nc-pgm@heidenhain.de

PLC programming ☎ +49 (8669) 31-3102

E-mail: service.plc@heidenhain.de

Lathe controls ☎ +49 (8669) 31-3105

E-mail: service.lathe-support@heidenhain.de

www.heidenhain.de

