

Programming Instructions

CNC Programming

XCx and Pronumeric

Version 07/07

Part No. R4.322.2090.0 (322 381 62)



schleicher



schleicher

CNC Programming for XCx and ProNumeric

Target Group

These programming instructions have been written for trained personnel with specialised knowledge. There are special demands on the selection and training of the personnel who work on the automation system. Suitable personnel are, for example, electricians and electrical engineers who have had the relevant training (see also Safety-related information, "Selection and Qualification of Personnel").

Applicability of these Programming Instructions

Version Hardware XX / Software XX

Previous versions of these programming instructions

11/00 08/02 07/05 02/06

Where can I obtain manuals?

You can download all our programming and operating manuals free of charge from our web site at <http://www.schleicher-electronic.com> or order them by writing to the following address (please quote order no.):

SCHLEICHER Electronic
GmbH & Co. KG
Pichelswerderstraße 3-5
D-13597 Berlin
Germany

Copyright by

SCHLEICHER Electronic
GmbH & Co. KG
Pichelswerderstraße 3-5

D-13597 Berlin
Germany

Phone ++49 (0) 30 33005 - 330

Fax ++49 (0) 30 33005 - 305

Hotline ++49 (0) 30 33005 - 304

Internet <http://www.schleicher-electronic.com>

Errors and omissions excepted. Subject to modifications.

Content

1	CNC Programming XCx and ProNumeric	7
1.1	Record structure	8
1.2	Program structure	11
1.2.1	Program number and program name	11
1.2.2	M17 and M30 program end	12
1.2.3	Initialization program	12
1.2.4	BN and BNR unconditional program branches	13
1.2.5	B% Unconditional subroutine call	14
1.2.6	Conditional program execution, comparisons	15
1.2.7	Conditional skipping of parts of records	16
1.2.8	Indirect programming with arithmetic parameters	17
1.2.9	Indexed programming	17
1.3	Calculations in the record	18
1.3.1	Calculations	18
1.3.2	Coordinate calculation	19
1.3.3	Constants	19
2	Feedrate, Acceleration and Spindle Speed	20
2.1	Programming Feedrate F (Path Feedrate)	20
2.2	Programming Acceleration ACC	22
2.3	Programming Spindle Speed S	23
3	G Functions	24
3.1	G0 Contour Control with Rapid Feed Velocity	26
3.2	G1 Contour Control with Linear Interpolation	27
3.3	G2, G3 and RC Circular and Helix Interpolation	28
3.4	G4 and T1 Dwell Time	32
3.5	G9, G60 Exact Positioning	33
3.6	G10 Point-to-point Positioning in Rapid Feed	34
3.7	G11 Homing	35
3.8	G12 and G13 Spiral Interpolation	36
3.9	G17, G18 and G19 Selecting the Working Planes	38
3.10	G25 and G26 Online Curve Interpolation OCI	39
3.11	G27 Freeform Interpolation	40
3.12	G32 Tapping with Controlled Spindle	41
3.13	G33 Thread Cutting Single Record	42
3.14	G39 Stop Record Preparation	44
3.15	G40 Switch Off Tool Radius Compensation	44
3.16	T Word Tool Selection for Tool Compensation	46
3.17	G41/G42 Tool Radius Compensation	47
3.18	G43 / G44 Tool Radius Compensation, Positive/Negative	50
3.19	G50 Tool Radius Compensation Without Transition Contour	51
3.20	G45/G46 Path Feedrate Compensation	52
3.21	RA, RB, RD, RF Rounding	52
3.22	G53 to G59 Zero Point Offset	56
3.23	G61, G64 Smoothing	58
3.24	G62 Record change with acceleration monitoring	60
3.25	G63 Tapping without compensating chuck	61
3.26	G66 Synchronization of IPO support points	62
3.27	G67 Special Function for Oscillating	63
3.28	G70 and G71 Inch/Metric Switching	63
3.29	G76 Thread Cutting Cycle	64
3.30	G77 Tapping without compensating chuck cycle	66
3.31	G80 to G89 Machining Cycles G80 to G89	67
3.32	G90, G91 Measurements Absolute/Incremental	68
3.33	G92 Reference Point Offset	69
3.34	G94, G95 Evaluation of F Word	70
3.35	G96, G97 Evaluation of S Word	71
3.36	G98, G99 Self-maintaining Preparatory Functions in Subroutines	72
4	\$ Functions	73
4.1	\$1 Stop Axis Motion Without Ramp	74



4.2	\$20 Handwheel Enable for Velocity Superposition.....	75
4.3	\$21 Handwheel Enable for Path Superposition.....	75
4.4	\$23 Internal Follow-up Operation On	75
4.5	\$24 Follow-up Operation On	76
4.6	\$25 Switch Off Follow-up Operation.....	76
4.7	\$26 Exclude Axes from Interpolation Context	77
4.8	\$27 Include Independent Axes in Interpolation Context	78
4.9	\$28 Include Independent Axis in Record Change	78
4.10	\$29 Do Not Include Independent Axis in Record Change	79
4.11	\$31 Switch On Synchronous Operation	80
4.12	\$32 Switch Off Synchronous Operation	81
4.13	\$33 Select Lead Axis for Thread Cutting.....	82
4.14	\$34 Select Radius Axis.....	82
4.15	\$37 Path length calculation	82
4.16	\$38 and \$39 Axis Selection for Path Feedrate Calculation	83
4.17	\$40 Switch Oscillation Off.....	84
4.18	\$41 Oscillation With Continuous Infeed.....	85
4.19	\$42 Oscillating With Infeed at Both Reversal Points	87
4.20	\$43 Oscillating With Infeed Only at Right Reversal Point.....	89
4.21	\$44 Oscillating With Infeed Only at Left Reversal Point	90
4.22	\$47 Define Machining Plane.....	90
4.23	\$48 Release Axis for Subsystem Change	91
4.24	\$53 - \$54 Abort Motion	92
4.25	\$90, \$91 Absolute/Incremental Measurements, Axis-specific	93
5	M Functions	94
5.1	M0.....	94
5.2	M1.....	94
5.3	M3 and M4.....	95
5.4	M5.....	96
5.5	M90 to M98 Synchronization of NC Subsystems	97
6	CNC-PLC Interface	98
6.1	E	98
6.2	SE.....	98
6.3	RS.....	99
6.4	WA and WN.....	99
7	Arithmetic Parameters for ProNumeric	100
7.1	General Arithmetic Parameters R2000 to R5999 (Integer Values)	100
7.2	General Arithmetic Parameters R6000 to R9999 (Real Values)	100
7.3	System-specific Arithmetic Parameters R000 to R999 (Integer Values)	100
7.4	System-specific Arithmetic Parameters R1000 to R1999 (Real Values)	101
7.5	Zero Point Offsets R10001 to R10564	101
7.6	Zero Overlays R10601 to R10664.....	102
7.7	R10701 to R10764 Reference Point Offset	102
8	Arithmetic Parameters for XCx (in Preparation).....	103
8.1	General Arithmetic Parameters R2000 to R5999 (Integer Values)	103
8.2	General Arithmetic Parameters R6000 to R9999 (Real Values)	103
8.3	System-specific Arithmetic Parameters R000 to R999 (Integer Values)	103
8.4	System-specific Arithmetic Parameters R1000 to R1999 (Real Values)	104
8.5	Zero Point Offsets R10001 to R10564	104
8.6	Zero Overlays R10601 to R10664.....	105
8.7	R10701 to R10764 Reference Point Offset	105
9	Overview Tables	106
9.1	Overview of G Words	106
9.2	Overview of \$ Words	108
9.3	M functions	109
9.4	CNC-PLC Interface.....	109
10	Appendix	110
10.1	Tool Compensations.....	110
10.1.1	Measuring tools	110
10.1.2	Quadrant assignment for tool nose radius compensation	111
10.2	Tool Data Memory	112

10.2.1	Tool monitoring.....	113
10.3	Approach and Departure Strategies.....	114
10.4	Contour Transitions	116
10.5	Lending NC Axes Between NC Subsystems.....	121
11	Index	122

Document conventions

This programming manual uses the following signs to indicate safety-related and handling warnings:



Possible injury to persons or damage to the automation system or the equipment if relevant warnings are not observed.

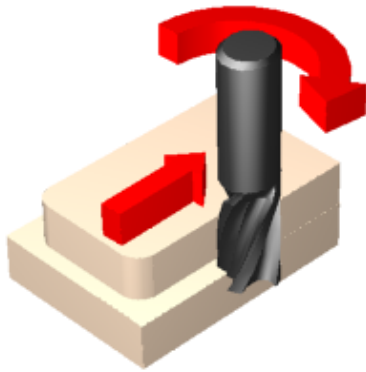


Important information on the handling of the automation system or the respective part in the operating manual.

Other objects are represented as follows.

Object	Example
File names	MANUAL.DOC
Menus / Menu Items	<i>Einfügen / Graphik / Aus Datei [Insert / Graphic / From file]</i>
Paths / directories	<i>C:\Windows\System</i>
Hyperlinks	http://www.schleicher-electronic.com
Program listings	MaxTsdr_9.6 = 60 MaxTsdr_93.75 = 60
Keys	<Esc> <Enter> (press first key, let go and press next key) <Ctrl+Alt+Del> (press all keys at the same time)
Configuration data identifier	Q34 and Q.054
Names of shared RAM variables	<i>cncMem.sysSect[n].flgN2P.bM345Act</i>

1 CNC Programming XCx and ProNumeric



The NC program for the XCx and ProNumeric is created to DIN 66025

An NC program comprises records, which are made up of words.

This can be called NC language.

A word in NC language is an address character and a sequence of digits.

Additional preparatory functions which are not defined in DIN 66025 are indicated by address identifier \$, followed by a one- or two-digit number.

The NC program processes one record after another. For the program to run on XCx or ProNumeric a PLC user program must be running, because the PLC program and the NC program work together.

You can create NC programs with any text editor that can save in ASCII format.

This programming manual uses the following signs to indicate safety-related and handling warnings:



Possible injury to persons or damage to the automation system or the equipment if relevant warnings are not observed.



Important information on the handling of the automation system or the respective part in the operating manual.



1.1 Record structure

Each record consists of several words (functions) and the record delimiter (inserted automatically when you press enter).
The type and number of words in a record is not fixed. The words in a record should be in the following order:

N	Record number
G, \$	G word for preparatory function The \$ function is placed in the record according to the function
A, B, C, D, L, O, P, U, V, W, X, Y, Z @I, @J, @K	Axis designations Axes I, J, K must be programmed with a prefixed @ to distinguish them from the interpolation parameters I, J, K.
I.. J.. K..	Interpolation parameters or parameters for thread pitch. These words each relate to a particular group of words for the coordinates and must be placed directly after the group.
F	Feedrate The F word is the feedrate for all programmed axes. For axes that do not move at feedrate (e.g. \$ function) the preparatory function, then the axis coordinates and then the F word plus axis letter are written after the path assignments.
ACC	Acceleration
S	Spindle speed
T	Tool including compensation
M	Additional or switching function
(.....)	Comment
\	Record extension (see Programming subsequent records)
/	Extraction symbol



The words for record number, coordinate, interpolation parameter and thread pitch parameter must not be repeated in a record.

A record must not contain more than 120 characters, including spaces and record delimiter. A record can be extended by Programming subsequent records.

[\(see page 10\)](#)

Record number

N	Record number
Format	Nnnnnnnn (nnnnnnn = 7-digit decimal number in range 1 to 9999999)
Explanation	The number is for locating program sections.
Note	The number does not determine the order in which records are processed. You can program records with the same number as long as they are not the destination of a jump instruction.
Example	
	N10
	N9999999

Comment

(.....)	Comment
Format	(Text)
Note	Use only displayable 7-bit ASCII characters, without the () characters.
Example	
	N10 (this is a comment)



Programming subsequent records

If the maximum record length of 120 characters is insufficient to program all the required NC words in one record you can use a backslash (\) at the end of the record to declare the next record as a subsequent record. The record decoder then treats both records as one.

Subsequent records do not appear in the record display (monitor etc.).

Example		
	N100 G1 G90 G61 X200.002 + R9012 * 12.345 - R9100 Y145.901 -R9102 / 1.205 (KOMMENTAR) ZR9600 * 123.456 M77 SE11 WA22\	Record and subsequent record
	N1001 F3000 R34:= R20+12	This record belongs to N100
	N110 G0	This is a new record



If the record number of a subsequent record is programmed as a jump address the record decoder will identify it as a normal record ID. Subsequent records should always have an unambiguous record number.

Example record structure	N10 G1 X100 Y5.4 F1000 ACC150 S500 M03 (comment)
N10	Record number
G1	G word should be at the start of the record for reasons of clarity.
X100	Axis designation and target coordinate in mm for all axes which move in this record. Up to 4 places after the point can be programmed, depending on the interpolation fineness. The default resolution setting is 1µm or 0.001.
F1000	Path feedrate in mm/min.
ACC150	Acceleration in %
S500	Speed of main spindle in r.p.m.
M03	M functions Switching functions whose execution is programmed in the PLC.
(.....)	Comment in brackets.
\	Subsequent record character (see "Programming subsequent records")

1.2 Program structure

%1000 (Name)	Program start	Program number and program name
N10	Sequence of records	The number of program records is limited only by the available memory capacity.
N20		
N30		
.		
.		
Nnnnn M30	M17 and M30 program end	

1.2.1 Program number and program name

%	Program number and program name	
Format	%nnnnnnnn (Name) nnnnnnnn = 8-digit decimal number in range 1 to 99999999 (name) = program name, max. 20 characters	
Explanation		
Note	There is basically no difference between the program numbers of main programs and subroutines. But you should organize the program numbers so that the program structure is clear.	
Example		
	%1	Program No. 1
	%1000 (machine startup)	Program No. 1000 Name: Machine startup
	%99999999	Program number 99999999 has preassigned special functions. (See Initialization program, G80 to G89 and cycle programming.)



The operating system can cope with 16 to 4096 programs.
 The default setting is 256.
 You can alter the setting via the user interface.



1.2.2 M17 and M30 program end

M17

M17	Subroutine end
Format	M17
Explanation	M17 causes a jump back to the calling NC program. If there is no calling program M17 has the same effect as M30. Not required if Q25 bit 5=1.

M30

M30	Program end
Format	M30
Explanation	Ends the NC program. Controller switches to RESET operating state. Not required if Q25 bit 5=1.

1.2.3 Initialization program

%nnnnnnnn	Initialization program
Format	%nnnnnnnn nnnnnnnn = 8-digit decimal number, default setting is 99999999
Explanation	Initialization program for setting parameters at CNC START.
Note	The initialization program runs before main program START. You can select any program number. The default setting is 99999999. The program number must be entered in Q130 in the configuration data of the subsystem. The initialization program must end with M17. If no program number is entered (Q130 = 0) the active CNC program is started directly at CNC start.

Example	%99999999
	N10 G11 X (Home to X)
	N20 G11 Y (Home to Y)
	. (Synchronize other axes)
	.
	N100 F1000 (Velocity presetting)
	N110 M17 (End subroutine)

1.2.4 BN and BNR unconditional program branches

BN	Unconditional program jump
Format	BNnnnn+/- nnnn = record number, +/- = search direction)
Note	The + symbol can be omitted.

Example	
	BN10- (Jump to record no. 10, search up to program start)
	BN120 (Jump to record no. 120, search down to program end)

BNR	Unconditional program jump parameterized
Format	BNRnnnn+/- nnnn = record number, +/- = search direction)
Note	The + symbol can be omitted.

Example	
	BNR10- (Jump to record no. in arithmetic parameter 10, search up)
	BNR20 (Jump to record no. in arithmetic parameter 20, search down)



1.2.5 B% Unconditional subroutine call

B%	Unconditional subroutine call
Format	B%nnnnnnnn nnnnnnnn = program number, 8-digit decimal number
Explanation	Program execution continues in the called program.
Note	

B%nnn.. R	Unconditional subroutine call with loop count
Format	B%nnnnnnnn R nnnnnnnn = program number
Explanation	The called NC program is repeated by the number of times indicated in the arithmetic parameter.
Note	The value in the arithmetic parameter is decremented on each repetition. The value must be a positive whole number when the call is made. If the value is ≤ 1 the subroutine will be executed once.

B%R	Unconditional subroutine call parameterized
Format	B%R R = arithmetic parameter
Explanation	The program number of the calling program is in the arithmetic parameter.
Note	You can calculate and call an 8-digit program number by specifying a max. 8-digit number as offset value with a max. 7-digit R parameter value. The following arithmetic functions are allowed: B%[Offset + R[R]xyz], B%[Offset - R[R]xyz], B%[R[R]xyz + Offset], B%[R[R]xyz - Offset].



The nesting depth of subroutines is 4.
Subroutines may not call themselves (recursive) or a previously called NC program.



The program number of a subroutine must be a positive whole number.
The subroutine must end with M17 (program end).
After the end of the subroutine program execution continues with the next record of the calling program.

1.2.6 Conditional program execution, comparisons

If comparisons are programmed in a record the following parts of the record will only be executed if the result of comparison is "true". If the result is "not true" only the part of the record before the comparison will be executed.

With comparisons you can create conditional program jumps and subroutine calls.

Comparisons with arithmetic parameters

Comparing arithmetic parameters	
R . . < Value	R less than value
R . . <= Value	R less than or equal to value
R . . = Value	R equal to value
R . . <> Value	R not equal to value
R . . >= Value	R greater than or equal to value
R . . > Value	R greater than value
Explanation	
Note	Arithmetic Parameters for ProNumeric (R parameters) are word flags which are used to save values in the NC program. For more on arithmetic parameters see section 7.

Example	
	N10 R1 < 10 BN100 (If R1 < 10 jump to record 100)



Parameter comparison is executed at the time of record decoding. Parameter changes between record decoding and record execution will be ignored.

G and \$ words and T calls will be executed regardless of the comparison.



Comparisons with bit variables

E...=	Requesting bit variables directly at the start of record execution
Format	Ennn=1 Ennn=0 nnn 3-digit decimal number in range 0 to -255)
Explanation	E 0 = <i>cnc.Mem.comSect.abFlg[0]</i> E 127 = E 128 = E 255 = <i>cnc.Mem.comSect.abFlg[255]</i> These bit variables are also used for CNC instructions SE, RS, WA and WN.
Note	Comparison is executed at the time of record change from the preceding record. The following records are not decoded until the comparison has been executed.
Example	N10 X100 E0 = 1 B%9000 (Program %9000 is called if E0 = 1; otherwise the CNC program continues in the next line.)



Comparison is executed at the time of record change from the preceding record. The following records are not decoded until the comparison has been executed.

1.2.7 Conditional skipping of parts of records

/	Conditional skipping of following part of record
Format	/
Explanation	You can exclude part of a record from execution using bit variables <i>cncMem.comSect.flgP2N.bBlkFade</i> for all NC records or <i>cncMem.sysSect[n].flgP2N.bBlkFade</i> for subsystem n.
Note	This function requires a PLC program.
Example	N10 SE01 / G11 X (The part of the record following / will not be executed if the bit variable = 1.)



The bit variable is requested at the time of record decoding. Changes to the bit variables between record decoding and record execution will be ignored.

1.2.8 Indirect programming with arithmetic parameters

The constants in a record can be replaced with arithmetic parameters. The arithmetic parameters are evaluated when the record is prepared.

Example	
	N10 GR0 XR1001 YR1002 FR1003 SER1
	N20 B% R2500
	N30 BN R10-

Axes X and Y move to the positions indicated in R1001 and R1002. The feedrate is taken from parameter R1003. The number of the G function is given by the content of R0 and the bit variable with the number from R1 is set. Then a jump is made to the program with the number from R2500.



Only positive whole R parameter values are valid for R parameters that replace whole number constants (e.g. SExx, BN%xx). Integer R parameters should be used for these (R0-R999, R2000-R5999).

The controller operating system does not round the decimal places of real R parameters!

If R0 = 1,001 in the above example program execution will be aborted with error message "Ungültige G Funktion" ["Invalid G function"].

\$ functions cannot be parameterized.

1.2.9 Indexed programming

As well as replacing a constant in indirect programming, arithmetic parameters can also be used as a pointer to another arithmetic parameter.

Example	
	N10 XRR1 (R1 = pointer to coordinate)
	N20 R1 := R1 + 1 R1 >= 20 R1 := 10

X-coordinate ←

R1

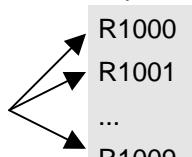


Table with X coordinates in R parameters

R1000
R1001
...
R1009

Each time the described subroutine is called it moves the X axis to the next position in the table. After 10 calls it starts again with the 1st position. For the sake of clarity start initialization and constraints have been omitted.



1.3 Calculations in the record

1.3.1 Calculations

Calculations		
R0	:= 100	Assigns a constant to an arithmetic parameter.
R0	:= R1	Assigns an arithmetic parameter to another arithmetic parameter.
R0	:= -R1	Negated assignment
R0	:= R1 + R2	Addition
R0	:= R1 - R2	Subtraction
R0	:= R1 * R2	Multiplication
R0	:= R1 : R2	Division
R0	:= ABS R1	Absolute value of R1
R0	:= SQR R1	Square root of the absolute value of R1
R0	:= SIN R1	Sine of R1 in degrees.
R0	:= COS R1	Cosine of R1 in degrees.
R0	:= TAN R1	Tangent of R1 in degrees.
R0	:= ATA R1	Arc tangent of R1 in degrees.
R0	:= R1 MOD R2	Division of R1 by R2. The whole number remainder of division is entered in R0.
Note		<p>The maximum number of assignments to one arithmetic parameter in one record is 8. Arithmetic Parameters for ProNumeric are variables which are used to save values in the NC program.</p> <p>For more on arithmetic parameters see section 7.</p> <p>If several assignments are programmed in a record they will always be executed from left to right.</p> <p>If several calculations are programmed in an assignment the calculations will always be carried out from right to left (reverse chain calculation).</p> <p>Brackets cannot be set (brackets indicate comments). Example:</p> <p>R1:= R2*R3+R4*R5+R6 corresponds to R1:= R2*(R3+(R4*(R5+(R6))))</p> <p>R7:= -R8+R9*R1 corresponds to R7:= -(R8+(R9*(R1)))</p>



In trigonometric functions the angle is indicated in degrees (0 to 360). Typical error is $1 \cdot 10^{-5}$ near the quadrant transitions, otherwise $1 \cdot 10^{-6}$.

1.3.2 Coordinate calculation

Axis coordinates can be calculated in the record, e.g. scale factors and offset.

Example	
	N10 X100 * R1001
	N20 Y200 + R1002



Parameter calculations with negative axis coordinates plus negative parameter value are calculated as follows:

N10 X-35+ R1003 (Content R1003 = -3)
 $X = -(35+(-3))$
 $X = -32$

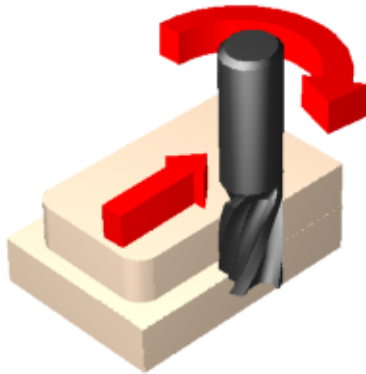
1.3.3 Constants

In all calculations arithmetic parameters can be replaced with a constant.

Example	
	$R1001 = \frac{(R1002 + R1003) * 2,5}{R1005}$



2 Feedrate, Acceleration and Spindle Speed



Feedrate in MANUAL mode

Path feedrate with G0

Path feedrate with G1

Path feedrate with G2/G3

Feedrate with G10

2.1 Programming Feedrate F (Path Feedrate)

The feedrate depends on the mode, the selected interpolation type and the machine data presets. The default setting is mm/min.

Feedrate in MANUAL mode

In MANUAL mode the axes move at the set conventional velocity (set in Q.000).
If rapid overlay is also used the axes move at the set rapid feed velocity (set in Q.028).

Path feedrate with G0

Programmed rapid feed. Path feedrate is calculated so that the slowest axis moves at its rapid feed velocity (set in Q.029).

Path feedrate with G1

With G1 all axes programmed in a record will be interpolated so that the resulting path feedrate corresponds to the programmed feedrate F. The unit for F depends on G94 (mm/min) and G95 (mm/spindle revolution).

Example
N10 G1 X100 Y50 Z20 F5000

Path feedrate with G2/G3

With circular interpolation the programmed feedrate F relates to the circular path. If other axes are programmed in this record they will be interpolated as straight lines and their velocity will be calculated so that they reach their target coordinates at the same time as the circular movement.

Feedrate with G10

Point-to-point positioning in rapid feed. Each axis moves at its rapid feed velocity (Q.029) to the programmed coordinate.

G94 feedrate / path feedrate in mm/min.

The feedrates/path feedrates programmed with the F word are calculated in mm/min.

G94 is the default setting.

G95 feedrate in mm/rev. of main spindle

The path feedrate programmed with the F word is calculated in mm/spindle revolution. A spindle with an actual value system is required for G95.



2.2 Programming Acceleration ACC

ACC	Ramp type and acceleration override
Format	ACCtnnn t = type of ramp 0 = Linear ramp 1 = Sin ² ramp 2 = Velocity reduction before record change (linear) nnn = Acceleration override 0 - 200%
Explanation	Acceleration is programmed as acceleration override in % of the preset acceleration value.
Note	<p>The programmable acceleration is self-maintaining, until M30 or CNC-RESET.</p> <p>Ramp type and ramp override can be changed with G64. If the ramp type is velocity reduction before record change, with G64 the linear ramp is always used (effective record by record).</p> <p>With the function for record transitions with any axes (RD-programming), if the programmed rounding distance is reduced the set velocity for the transition records is also reduced.</p> <p>If ACC2000+100 (2000 = ramp type linear + acceleration 100%) is programmed deceleration to record change velocity will take place before record change.</p> <p>Record change velocity is always the lower velocity of the two records.</p> <p>ACC2100 FF50%.....</p> <p>The ACC function cannot be applied to independent axes and special functions like G33, G63 and oscillating.</p> <p>Ramp type for automatic mode is preset in Q37, bit 4. 0= linear ramp 1 = sin² ramp</p> <p>Manual mode always uses linear ramp.</p>

Example		
	N10 G1 X100 Y500 F2000 ACC50	(Acceleration with 50 % linear ramp)
	N20 G1 X100 Y650 F500 ACCR1	(Acceleration value in R1)
	N30 G1 X350 Y650 F1500 ACCRR0	(Parameter no. for acceleration value in R0)

Example		
	N110 G1 G64 X10 ACC0050 F100	(Acceleration with 50 % linear ramp)
	N120 X100 ACC1100 F2000	(Acceleration with 100% sine ramp)
	N120 X150 ACC 2050	(Deceleration with linear ramp to record change velocity F100)
	N130 G60 X160 ACC1100 F100	(Deceleration with sine ramp to standstill)

2.3 Programming Spindle Speed S

S	Spindle speed
Format	Snnnnn SXnnnnn nnnnn = 5-digit decimal number X = any axis letter
Explanation	The programmed value is evaluated as the spindle speed (default setting) in r.p.m. or cutting speed m/min. If there are several spindles in a subsystem, one axis can be selected as the spindle by entering an axis letter.
Note	<p>With G97 the speed is in r.p.m.</p> <p>With G96 the S word is the cutting speed (circumferential speed) in m/min. The radius associated with the circumference is formed from the actual value of an axis specified with \$34.</p> <p>The speed of other spindles is programmed with S"axis name".</p> <p>The value programmed in S is entered in the shared RAM variable <i>cncMem.axSect[n].wrN2P.lPrgSVal</i>.</p> <p>Variable <i>cncMem.axSect[n].flgN2P.bSFctMod</i> is set to TRUE as the modification signal. This variable must be acknowledged by the PLC user program.</p> <p>If no axis in the subsystem is specified as spindle or rotary axis (Q.054) the content of the S word will be saved in the variables <i>cncMem.sysSect[n].wrN2P.lSFct</i> for processing by a PLC program. Variable <i>cncMem.axSect[n].flgN2P.bSFctMod</i> then serves as the modification signal.</p>
Example	
	N10 G1 X100 Y100 S3500



3 G Functions

According to DIN 66025 Part 2, G functions are CNC functions that describe the interpolation context of the NC axes. In this overview the G words are organized in groups.

Only one function from each group can be active.

Normally the functions remain active until they are deselected by another function from the same group.

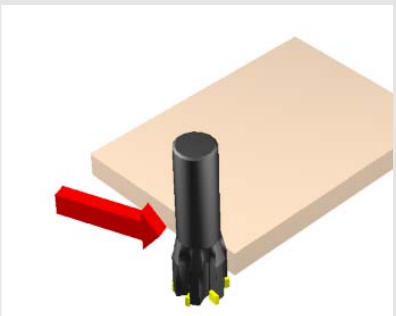
In individual cases a function is active for only one record (property = S). Some functions are default settings (property = D).

Group	Properties D = Default setting S = Active for 1 record		Meaning
1		G0	Contour control in rapid feed.
	D	G1	Linear interpolation
		G2	Clockwise circular-helix interpolation
		G3	Anticlockwise circular-helix interpolation
		G10	Point-to-point positioning in rapid feed
		G11	Home to reference point
		G12	Clockwise spiral interpolation
		G13	Anticlockwise spiral interpolation
		G25	Online curve interpolation OCI without tangential transition
		G26	Online curve interpolation OCI with tangential transition
		G27	Freeform interpolation of CNC programs created offline
		G32	Tapping with controlled spindle
		G33	Thread cutting
		G63	Tapping without compensating chuck
		G76	Thread cycle
		G77	Tapping cycle without compensating chuck
2	S	G4	Dwell time
3	D	G17	Plane selection X-Y
		G18	Plane selection X-Z
		G19	Plane selection Y-Z
4	S	G39	Interrupt record preparation
5	D	G40	Switch off tool radius compensation
		G41 G42	Tool radius compensation left/right
		G43 G44	Tool radius compensation positive/negative
	S	G50	Tool radius compensation without transition contour
6		G45 G46	Feedrate correction
7	D	G53 to G59	Zero point offset

Group	Properties D = Default setting S = Active for 1 record		Meaning
8	S	G9	Exact positioning
	D	G60	Record change after exact stop boundary reached
		G61	Record change after elimination of set-actual deviation
		G62	Record change with acceleration monitoring
		G64	Record change without loss of velocity
		G66	Synchronization of IPO support points
9	S	G67	Special function for oscillating
10		G70	Measurements in inches; the last used function applies
		G71	Measurements in millimeters
11	D	G80 to G89	Machining cycles
12	D	G90	Absolute measurements
		G91	Incremental measurements
13		G92	Reference point offset
14	D	G94	Feedrate given in mm/min (in/min)
		G95	Feedrate given in mm/rev. (in/rev.)
15		G96	Constant cutting speed
	D	G97	Spindle speed given in r.p.m.
16	D	G98	Accept self-maintaining preparatory functions
		G99	Do not accept self-maintaining preparatory functions

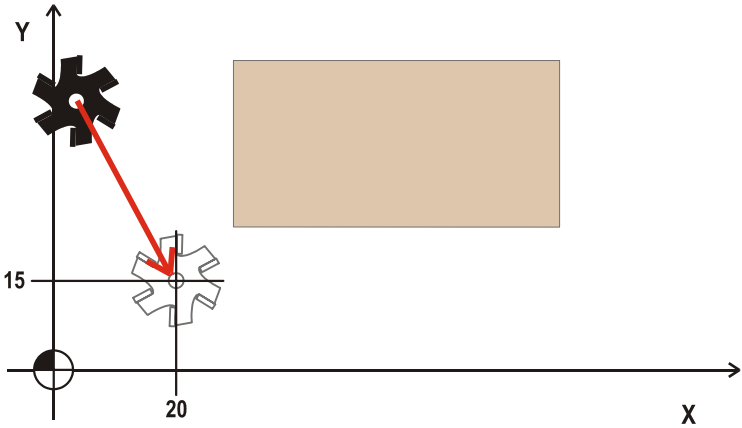


3.1 G0 Contour Control with Rapid Feed Velocity

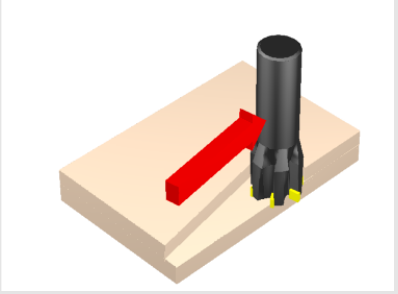
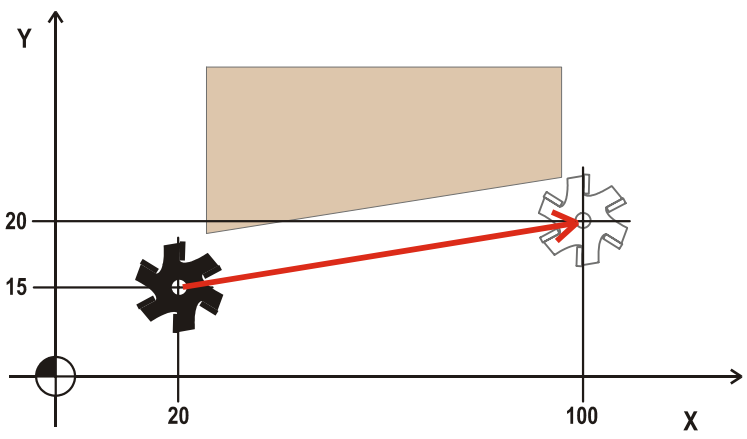
G0	Contour control with rapid feed velocity and linear interpolation	
Format	G0 X Y X, Y = any axis letter	
Explanation	All axes reach the programmed end position simultaneously. The path feedrate is calculated in the controller so that the shortest positioning time is achieved without exceeding the axis-specific rapid feed velocity (Q.029).	
Note	<p>Record change does not occur until exact position has been reached on all axes, regardless of the exact positioning level selected with G60 to G64.</p> <p>If this behaviour is not desired you can set bit 2 in configuration parameter Q38.</p> <p>The programmed feedrate F is not active but is retained and reactivated after G0.</p> <p>As long as G0 is active the shared RAM variable <i>cncMem.sysSect[n].flgN2P.bG0Act</i> is set to TRUE.</p>	



G0 is not suitable for workpiece machining.

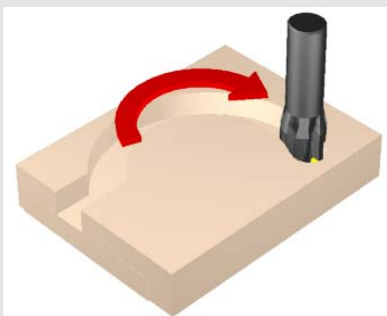
Example	
	N10 G0 X20 Y15
	

3.2 G1 Contour Control with Linear Interpolation

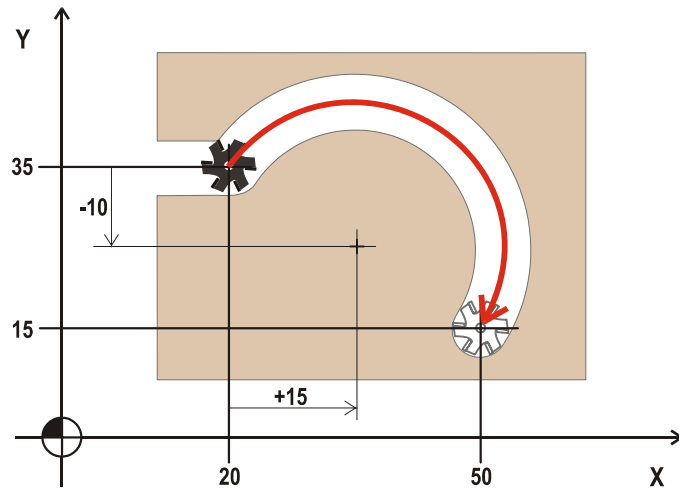
G1	Contour control with linear interpolation	
Format	G1 X Y F X, Y = any axis letter F = path feedrate	
Explanation	All axes reach the programmed end position simultaneously on a straight line. The path feedrate is identical with the current programmed feedrate F.	
Note	Linear interpolation is permissible on all axes simultaneously, n-dimensionally. The maximum achievable path feedrate is restricted by the slowest axis so it cannot be slower than the velocity programmed in F. As long as G1 is active the shared RAM variable <i>cncMem.sysSect[n].flgN2P.bG1Act</i> is set to TRUE.	
Example	N10 G1 X100 Y20 F1000 (The end position is approached on a straight line at a path feedrate of 1000 mm/min.)	
		



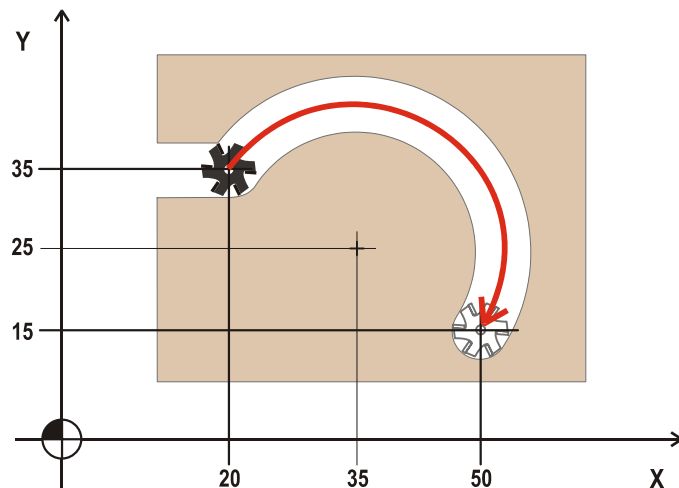
3.3 G2, G3 and RC Circular and Helix Interpolation

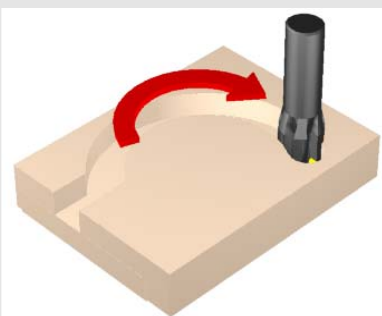
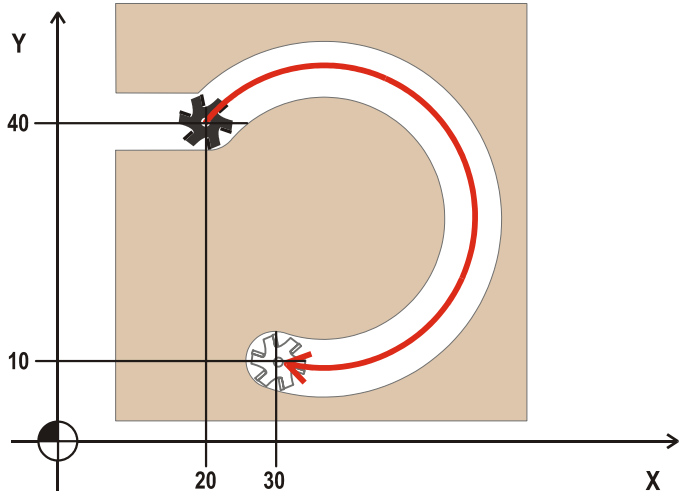
G2	Clockwise circular and helix interpolation		
G3	Anticlockwise circular and helix interpolation		
Format	G2 X Y I J F G3 X Z I K F G2 X Y RC F X, Y, Z = axis letters I, J, K = auxiliary coordinates F = path feedrate RC = radius		
Explanation	Circular and helix interpolation with specified circle centre.		
Note	<p>The circle centre is programmed using the auxiliary coordinates I, J, K or indicated in RC. The auxiliary coordinates are assigned to the axes:</p> <p>Axes parallel to X = I Axes parallel to Y = J Axes parallel to Z = K</p> <p>You can set the reference points of the auxiliary coordinates: relative, relating to the record start position (Q25 bit 2= 0) or absolute, relating to the currently selected coordinate system (Q25 bit 2= 1).</p> <p>Circular interpolation can only be carried out in one plane. The circle plane must concur with the selected working plane G17/G18/G19. If this is not the case an error will be indicated and program execution will be interrupted.</p> <p>The corresponding dimensional coordinates must be assigned to the axes involved in circular interpolation. This is done by setting a bit (0, 1 or 2) in configuration parameter Q.054. If the same dimensional coordinates are assigned to several axes one must be selected with \$47.</p> <p>The end coordinates can be absolute or incremental, depending on the preparatory function (G90 or G91).</p> <p>The circle end position must achieve the precision set in Q.106, otherwise an error message will be output. Permissible deviations of the circle end position will be compensated by spiral interpolation in the circular path. If other axes are programmed as well as the circular axes, these axes will be included in the interpolation context so that they reach the end position at the same time as the circular axes (helix interpolation). The path feedrate programmed in F relates to the resulting path (see \$38).</p> <p>If the start and end positions of the circle are identical a full circle will be interpolated unless the circle centre was specified with RC.</p>		

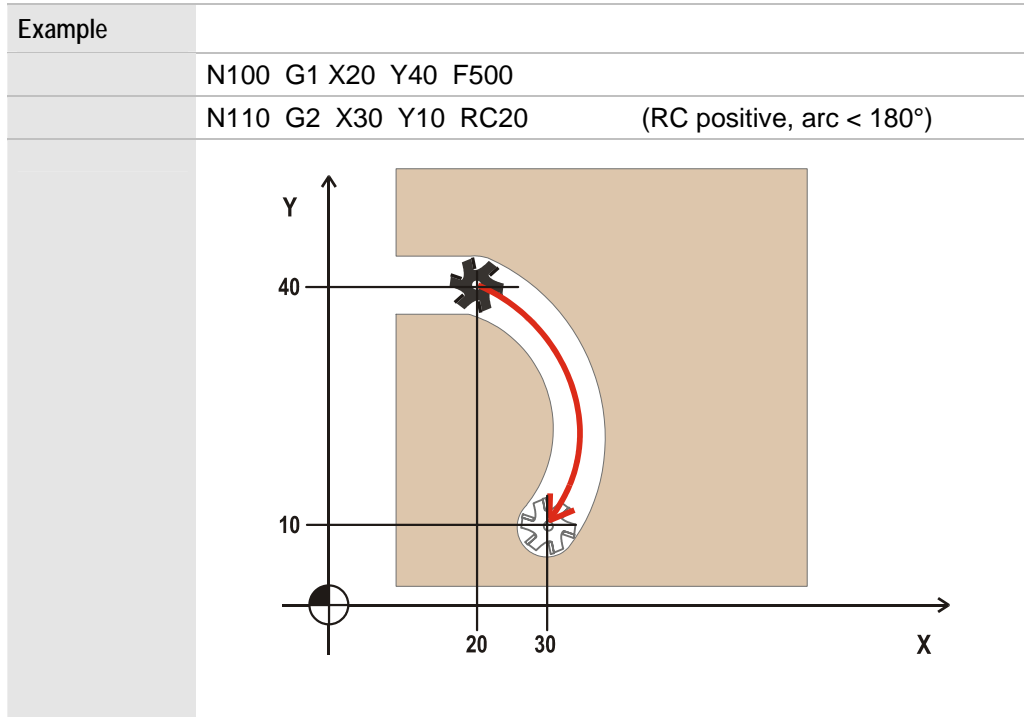
Example	Auxiliary coordinates with incremental reference (Q25 bit 2=0)
	N10 G1 X20 Y35 F1000
	N20 G2 X50 Y15 I15 J-10 (The absolute programmed end position is approached clockwise on a circular path at a constant path feedrate of 1000 mm/min.) The circle centre coordinates are relative, relating to the start position.)



Example	Auxiliary coordinates with absolute reference (Q25 bit 2=1)
	N10 G1 X20 Y35 F1000
	N20 G2 X50 Y15 I35 J25 (The absolute programmed end position is approached clockwise on a circular path at a constant path feedrate of 1000 mm/min.) The circle centre coordinates are absolute, relating to the programmed zero point.)



RC	<p>Circular and helix interpolation with radius programming</p> 
Format	<p>RCnnn RCRnnn</p> <p>nnn = decimal number Rnnn = arithmetic parameter</p>
Explanation	<p>Circular and helix interpolation with specified arc radius.</p>
Note	<p>Only the end coordinates and the radius have to be programmed: If $RC < 0$ (negative), an arc with angle at circumference $> 180^\circ$ will be made. If $RC > 0$ (positive), an arc with angle at circumference $< 180^\circ$ will be made. A full circle can only be programmed as two parts.</p>
Example	<p>N100 G1 X20 Y40 F500</p> <p>N110 G2 X30 Y10 RC-20 (RC negative, arc $> 180^\circ$)</p> 





3.4 G4 and TI Dwell Time

G4	Dwell time
Format	G4 F G4 R F = dwell time in seconds R = arithmetic parameter contains dwell time in seconds

Example	
	N10 G4 F1.2 (dwell time 1.2 seconds)
	N10 G4 R1002 (dwell time in R parameter R1002)
	N10 G4 FR1002 (dwell time in R parameter R1002)

TI	Dwell time
Format	TI nnn TI R nnn = decimal number (integer double word 2,147,483,647), unit sec. R = arithmetic parameter contains dwell time in seconds
Explanation	TI can be programmed parallel to the motion
Note	If a TI dwell time is programmed parallel to a movement the time will run in parallel to the movement. The record changes when both conditions have been met: target coordinate reached and time expired. The dwell time is effective record by record. Can also be programmed with G4.

Example	
	N10 TI 2.5 (dwell time 2.5 seconds)
	N10 TI R1002 (dwell time in R parameter R1002)
	N10 G1 X0.5 F500 TI2.5 (dwell time 2.5 seconds during G1)

3.5 G9, G60 Exact Positioning

G9	Exact positioning, effective record by record
Format	G9
Explanation	Record change occurs when the following error of all axes in the subsystem is less than the respective stop tolerance range. (see configuration data Q.048)
Note	Exact positioning with G9 is effective for just one record. In the next record the previously programmed record change condition applies. As long as the axis is not in exact position the shared RAM variable <i>cncMem.axSect[n].flgN2P.blInPos</i> is set to FALSE.

G60	Exact positioning, self-maintaining
Format	G60
Explanation	Record change occurs when the set position has been reached and the following error of all axes in the subsystem is less than the respective stop tolerance range (Q.048).
Note	As long as the axis is not in exact position the shared RAM variable <i>cncMem.axSect[n].flgN2P.blInPos</i> is set to FALSE. G60 is the default setting. It can be deselected with G61 or G64.

Example	
	N10 G60 G1 X1000 F1000
	N20 X2000 F500



3.6 G10 Point-to-point Positioning in Rapid Feed

G10	Point-to-point positioning in rapid feed
Format	G10 X Y X, Y = any axis letter
Explanation	In contrast to G0, all axes move at their axis-specific rapid feed velocity, so they do not normally reach the end position simultaneously.
Note	Record change does not occur until exact position has been reached on all axes, regardless of the exact positioning level selected with G60 to G64. Feedrates programmed in F are retained and are reactivated when G10 is deselected.



G10 is not suitable for workpiece machining.

Example	
	N10 G10 X40 Y15

3.7 G11 Homing


G11	Homing
Format	G11 X X = any axis letter
Explanation	The selected axis homes to its reference point
Note	<p>The axes are not interpolated and move at their specific velocities. If the axis is not synchronized home position search velocities will always be used.</p> <p>If the axis is synchronized it will move to the home position coordinate at programmed velocity F or rapid feed. The velocity must not be programmed in the G11 record.</p> <p>G11 is effective record by record.</p> <p>NC record preparation is stopped until the NC record has been processed (implicit G39).</p>

Example	
	N10 G0
	N20 G11 X
	or
	N10 F1000
	N20 G11 X

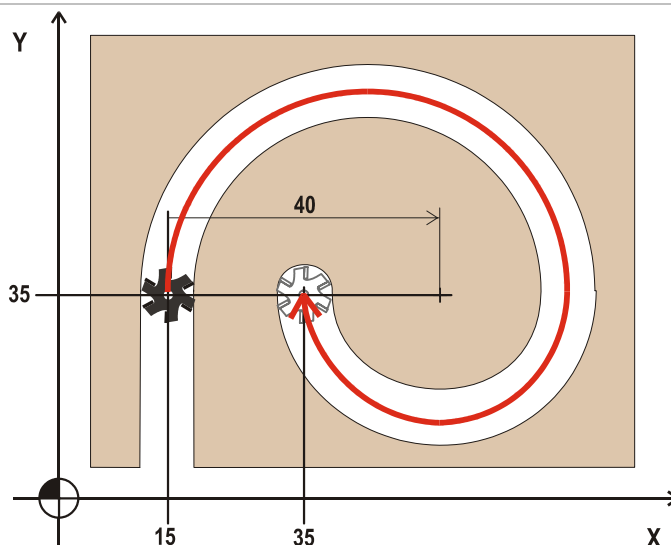
If several axes are moved in one record with G11 the reference coordinate of the axes must be specified.

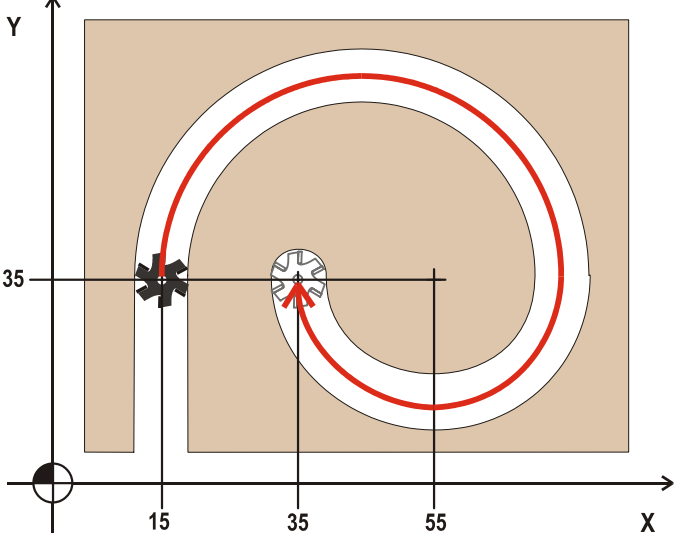
Example	
	N10 G0
	N20 G11 X0 G11 Y0

3.8 G12 and G13 Spiral Interpolation

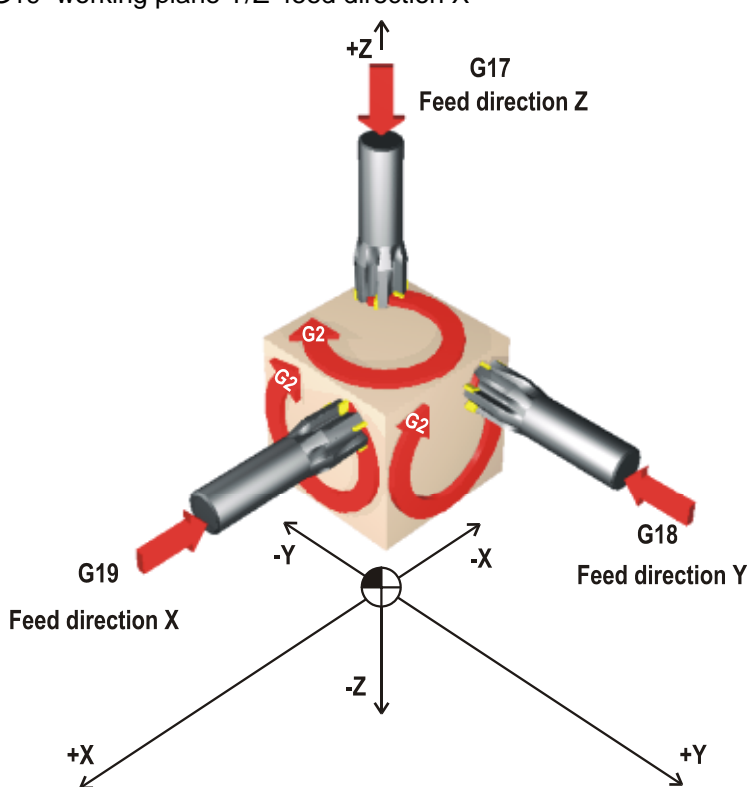
G12	Clockwise spiral interpolation	
Format	G12	
G13	Anticlockwise spiral interpolation	
Format	G13	
Explanation	G12/G13 programming corresponds to G2/G3. In spiral interpolation the difference between start radius and end radius is travelled at path angle, generating an Archimedean spiral.	
Note	As in circular interpolation, the coordinates of the centre can be specified with I, J, K in absolute or relative terms (Q25 bit 2 = 1 absolute, bit 2 = 0 relative). If other axes are programmed as well as the spiral axes, these axes will be included in the interpolation context so that the third axis reaches the end position at the same time as the spiral axes (helix interpolation).	

Example	Auxiliary coordinates relative to start position (Q25 bit 2=0)	
	N10 G1 X15 Y35 F1000	
	N20 G12 X35 Y35 I40 J0	(The programmed end position is approached clockwise on a spiral path at a constant path feedrate of 1000 mm/s) The spiral centre coordinates are relative, relating to the start position.)



Example	Auxiliary coordinates absolute (Q25 bit 2=1)
	N10 G1 X15 Y35 F1000
	N20 G12 X35 Y35 I55 J35 The spiral centre coordinates are absolute, relating to the programmed zero point.)
	

3.9 G17, G18 and G19 Selecting the Working Planes

G17	Plane selection X/Y
G18	Plane selection X/Z
G19	Plane selection Y/Z
Format	G17 G18 G19
Explanation	<p>G17 working plane X/Y feed direction Z G18 working plane X/Z feed direction Y G19 working plane Y/Z feed direction X</p> 
Note	<p>Machine axes are assigned to dimensional coordinates in configuration parameters Q.054 bit 0 – 2.</p> <p>If several axes in one subsystem are assigned to the same dimensional coordinate you can select with \$47.</p> <p>Plane selection also defines the plane for the following functions:</p> <ul style="list-style-type: none"> • Circular and helix interpolation (in the figure the direction of rotation is shown for G2/G12) • Tool length and radius compensation <p>G17 is the default setting.</p>

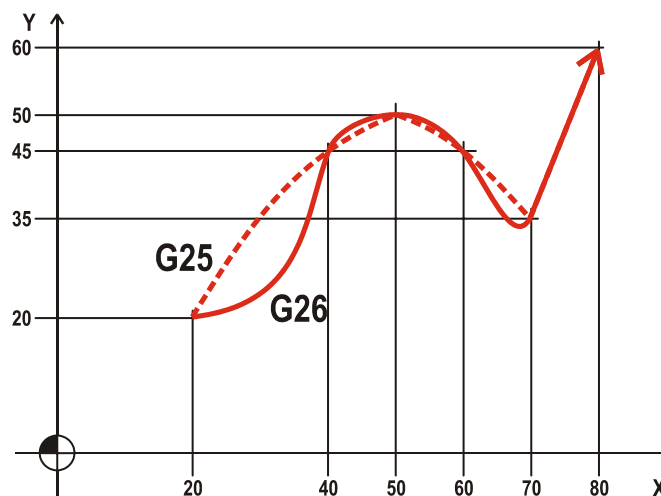
3.10 G25 and G26 Online Curve Interpolation OCI

G25	Online curve interpolation OCI without tangential transition
G26	Online curve interpolation OCI with tangential transition
Format	G25 X Y Z G26 X Y Z X, Y, Z = any axis letters
Explanation	Contour control for smooth, stepless paths.
Note	You can use the select/deselect condition to specify whether there will be a tangential transition to preceding or following programmed paths. The online curve interpolator requires interpolation points in the form of CNC records. G1 or G2/G3 can be programmed at any point to generate sharp corners or straight lines. OCI can be used simultaneously with all axes in the NC subsystem, so you can generate three-dimensional curves of unlimited complexity.



After OCI has been deselected by any G function (G0/G1....) at least 2 motion records must follow for OCI to operate.
G39 and NC functions containing an implicit G39 are not permissible in OCI.
Tool path compensation with G40 to G44 are not permissible in connection with OCI.

Example	G25 OCI without tangential transition.	G26 OCI with tangential transition.
	N10 G1 X20 Y20	N10 G1 X20 Y20
	N20 G25 X40 Y45	N20 G26 X40 Y45
	N30 X50 Y50	N30 X50 Y50
	N40 X60 Y45	N40 X60 Y45
	N50 X70 Y35	N50 X70 Y35
	N60 G1 X80 Y60	N60 G1 X80 Y60
	N70 X125 Y65	N70 X125 Y65

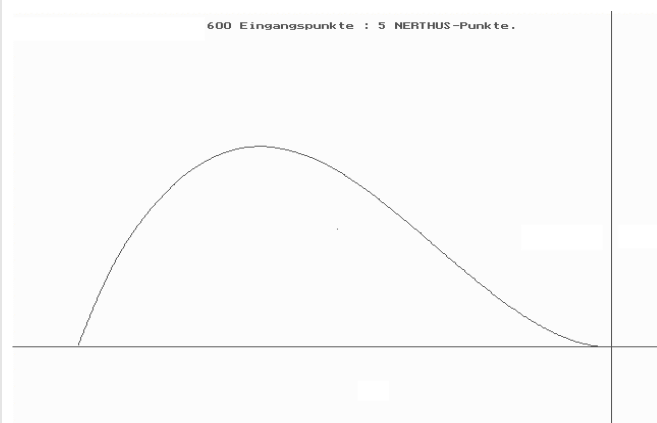




3.11 G27 Freeform Interpolation

G27	Free form interpolation of CNC programs created offline
Format	G27
Explanation	Contour control based on NERTHUS* interpolation point reduction *NERTHUS is a Schleicher software product
Note	<p>This function requires offline programming (e.g. CAM system) with subsequent processing by NERTHUS software.</p> <p>The NERTHUS software prepares the table of axis coordinates for the G27 function.</p> <p>The table can include up to 6 axes of a freeform. It is used to create the CNC subroutine (reduced with NERTHUS), which may not be altered in the CNC controller.</p> <p>If a contour correction is required, offline programming and preparation with the NERTHUS software has to be repeated.</p> <p>Calculations of compensation and transformation must be done during offline programming.</p> <p>Zero point offsets are permissible.</p> <p>Please refer to the NERTHUS software manual to ensure correct operation.</p>

Example	A CNC program created from initially 600 points after processing with the NERTHUS software
	<pre>%1 N0 G1 G90 X-37.937 Y.169 N1 G27 G64 X-31.16 Y12.503 IX7.1 IY18.858 JX17.964 JY18.858 N2 X-24.067 Y15.399 JX10.845 JY-1.575 N3 X-13.617 Y8.802 JX15.977 JY-14.615 N4 G61 X0 Y0 JX20.819 JY-.099 N5 M17</pre>



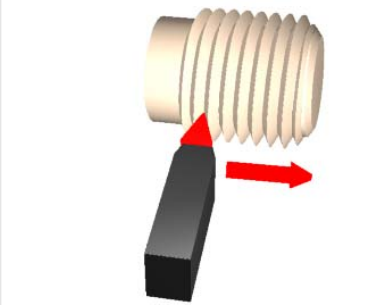
3.12 G32 Tapping with Controlled Spindle

G32	Tapping with controlled spindle
Format	G32 Z I Z = any axis letter I = pitch
Explanation	<p>In contrast to tapping with G63, in this function the spindle is interpolated with the lead axis. This requires a position-controlled spindle.</p> <p>The thread pitch I can be positive (tapping with M3) or negative (tapping with M4). I is only be programmed in the first G32 record.</p> <p>G32 is especially suited for blind holes, because the exact depth is achieved.</p>
Note	<p>G32 must be called with the spindle stopped (M5).</p> <p>The lead axis must be specified with \$33 before G32 is called.</p> <p>The spindle speed must be programmed in S.</p> <p>M3, M4 and M5 must not be used.</p> <p>Single record mode, speed override and stop key are not locked. All other modes are locked.</p> <p>A thread can also be programmed with several G32 records. Record change conditions G60, G61 or G64 apply. This makes it possible, for example, to output an M function during tapping.</p>

Example	#
	N10 \$33 Z S2000 M5 Lead axis, 2000 r.p.m., spindle stop
	N20 G0 Z200 (C90) Start position (possibly also for spindle)
	N30 G32 Z190 I2 Thread with M3, pitch 1 mm
	N40 Z200 Z back, spindle reversed
	N50 G0 Continue with G0



3.13 G33 Thread Cutting Single Record

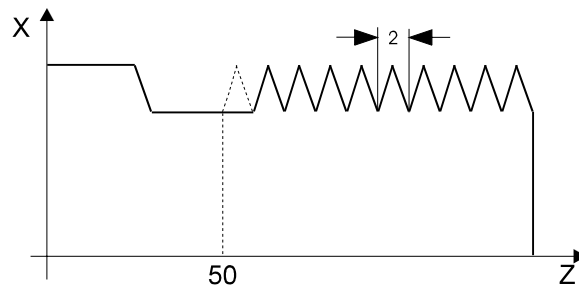
G33	Thread cutting single record	
Format	G33 X Z K X, Z = axis letters I, J, K = auxiliary coordinates	
Explanation		
Note	<p>Function G33 requires a spindle with a positioning transducer. The spindle can be operated as a controlled, uncontrolled or PLC-controlled spindle.</p> <p>Before G33 is called:</p> <ul style="list-style-type: none"> the direction of rotation of the spindle and the speed must be programmed, the lead axis must be declared via the \$33 function. <p>Record change must not occur until the spindle is turning in the programmed direction. When G33 is active right-hand or left-hand thread is decided by the direction of rotation of the spindle and the travel direction. A later alteration is not possible.</p> <p>If the direction of rotation of the spindle changes the axis returns to the record start position and stops there.</p> <p>The thread pitch is programmed by an auxiliary coordinate (I, J, K) matched to the axis (X = I, Y = J, Z = K).</p> <p>Cycle G76 is available for thread cutting.</p>	



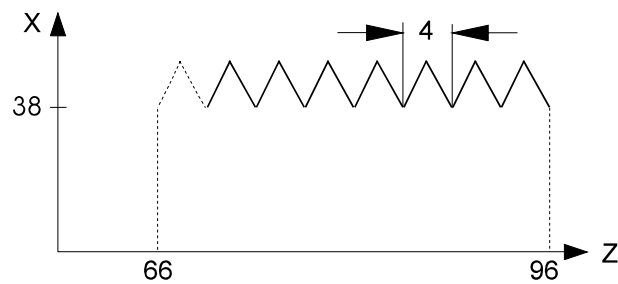
Locking with G33:

- Override is set to 100%.
- Stop key is locked.
- In single record mode stop is not until after the last G33 record.
- In single record mode stop is not until after the last G33 record.

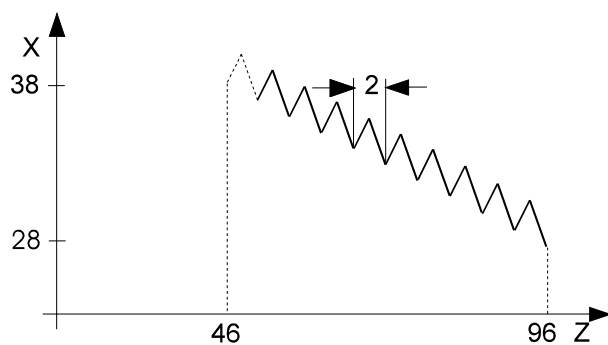
Example	Cylindrical thread	
	N10 M03 S700	(Spindle on, speed 700 r.p.m.)
	N20 \$33 Z	(Z is the lead axis)
	N30 G33 Z50 K2	(Pitch is 2 mm)



Example	Cylindrical thread	
N10	M03 S700	(Spindle on, speed 700 r.p.m.)
N20	\$33 Z	(Z is the lead axis)
N30	G0 X38 Z96	(Home to start position)
N40	G91 G33 Z-30 K4	(Thread length 30 mm with 4 mm pitch)
N50	G0 G90 X35 Z98	(Move to end position)
N60	M05	(Spindle off)



Example	Conical thread	
N10	M03 S900	(Spindle on, speed 900 r.p.m.)
N20	\$33 Z	(Z is the lead axis)
N30	G0 X28 Z96	(Home to start position)
N40	G91 G33 X10 Z-50 K2	(Cone 10x50, pitch 2 mm)
N50	G0 G90 X40 Z98	(Move to end position)
N60	M05	(Switch spindle off)



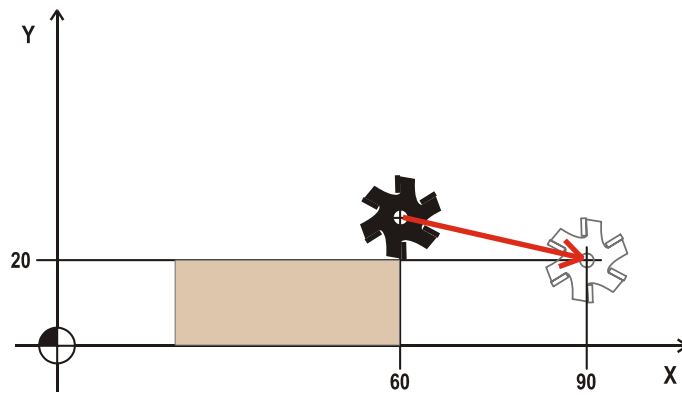


3.14 G39 Stop Record Preparation

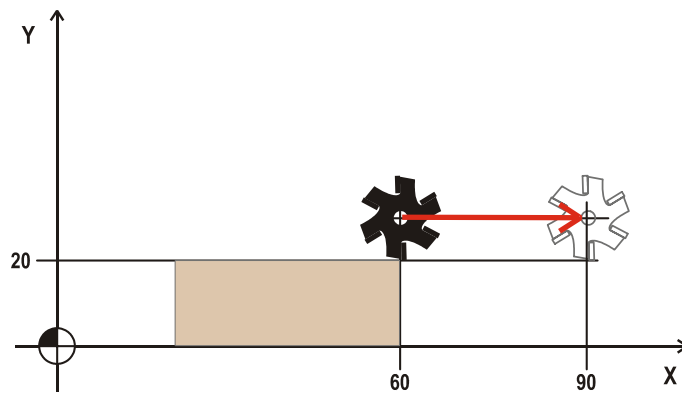
G39	Stop record preparation
Format	G39
Explanation	Record preparation (decoding) stops until the buffer is empty and the last prepared record has been processed.
Note	G39 is activated automatically in the following functions: G11 homing E1 = 0 or 1 communication flag comparison Read/write Q parameter by record Change NC axes between NC subsystems \$1stop an axis \$25 switch off follow-up operation \$28 reintegrate axis in record change \$32 if bit 1 =1 in Q37 \$40 oscillation off

3.15 G40 Switch Off Tool Radius Compensation

G40	Switch off tool radius compensation
Format	G40 [X Y Z F] X, Y, Z = any axis letters F = path feedrate
Explanation	Tool radius compensation is switched off
Note	If G40 is programmed with a motion the tool radius compensation is activated on the path. If G40 is programmed without a motion the tool centre becomes the actual position of the axis. Tool radius compensation is also switched off with M30.
Example	G40 with motion
	N30 G41 X... F500
	N40 G41 X60 F500
	N50 G40 X90 F500



Example	G40 without motion
	N30 G41 X... F500
	N40 G41 X60 F500
	N50 G40
	N60 G1 X90





3.16 T Word Tool Selection for Tool Compensation

T	Tool selection
Format	Tnn nn = number of Tool Data Memory, 2-digit decimal number
Explanation	The Tool Data Memory nn is selected and activated.
Note	<p>Tool data, stored in Tool Data Memory, is taken into consideration in the travel instruction. It remains valid until another tool is selected or tool compensation is switched off with T0.</p> <p>The number of the selected tool data memory is continuously displayed in shared RAM variable <i>cncMem.sysSect[n].wrnN2P.IToolMem</i>.</p> <p>The T function is also displayed by a modification signal in shared RAM variable <i>cncMem.sysSect[n].flgN2P.bTFctMod</i>. Record change does not occur until this variable has been set to FALSE by a user program.</p>
Example	
	N10 G1 X100 Y50 T0001 Tool 0, Tool Data Memory 1 selected
	N20 G0 X0 Y0 T0 Tool compensation switched off



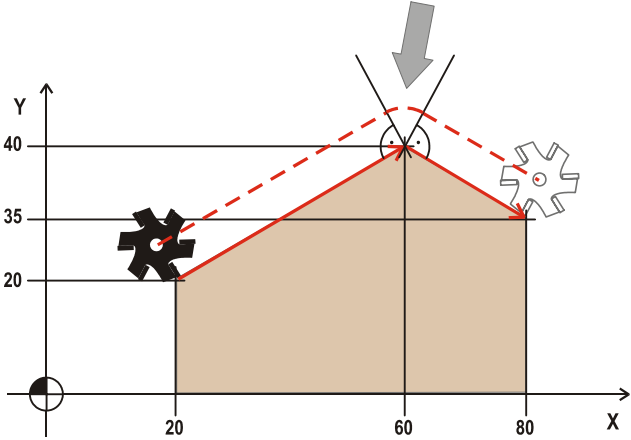
The tool is regarded as deselected after M30 or a program abort through RESET. So if the NC program is restarted tool selection must be specified in the first positioning record.

The actual value display is corrected accordingly.



If an NC program is to be executed in several subsystems ensure that the Tool Data Memory with corresponding tool data is entered in each subsystem when you select tools.

3.17 G41/G42 Tool Radius Compensation

G41	Tool radius compensation (WRK) left of contour
G42	Tool radius compensation (WRK) right of contour
Format	G41 X Y Z G42 X Y Z F X, Y, Z = any axis letters F = path feedrate
Explanation	With functions G41 and G42 you can carry out tool path compensation regardless of the tool data.
Note	<p>You can compensate tool radius WRK (default setting) or tool nose radius SRK. To activate SRK you have to select a compensation quadrant and enter it in the Tool Data Memory. See "Quadrant assignment" in the annex.</p> <p>Tool selection Before WRK a tool must be selected with the T word. Tool compensation data for the tool must be stored in the corresponding Tool Data Memory.</p> <p>Plane A machining plane for the WRK must be selected with instructions G17, G18, G19. The machining plane cannot be altered while WRK is active.</p> <p>Approach When a WRK has been selected the tool radius is activated in the first positioning record. The selection must be outside the machining contour and the approach path must be clear. See also "Approach and Departure Strategies" in the annex.</p> <p>Compensation is parallel to the contour. The axes move so that the tool centre is perpendicular to the programmed contour.</p> <p>Depending on the programmed contour, transition radii may be inserted by the record decoder.</p>  <p>The transition radius is a separate record, displayed under the number of the preceding record. These records are not included in the NC program, merely saved in the buffer. G50 operates without inserted intermediate records.</p> <p>The tool centre is always displayed in the actual and set value displays.</p>

Feedrate calculation can be switched with G45 or G46.



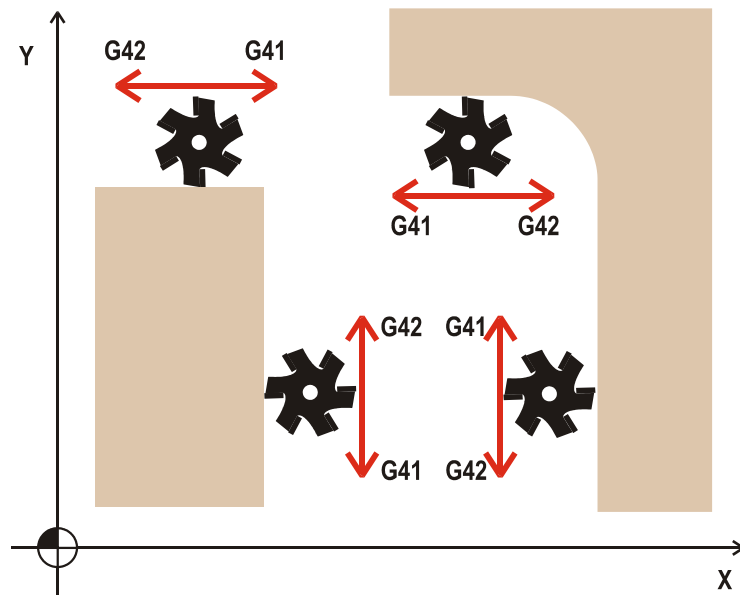
When tool radius compensation is active the following restrictions must be considered.

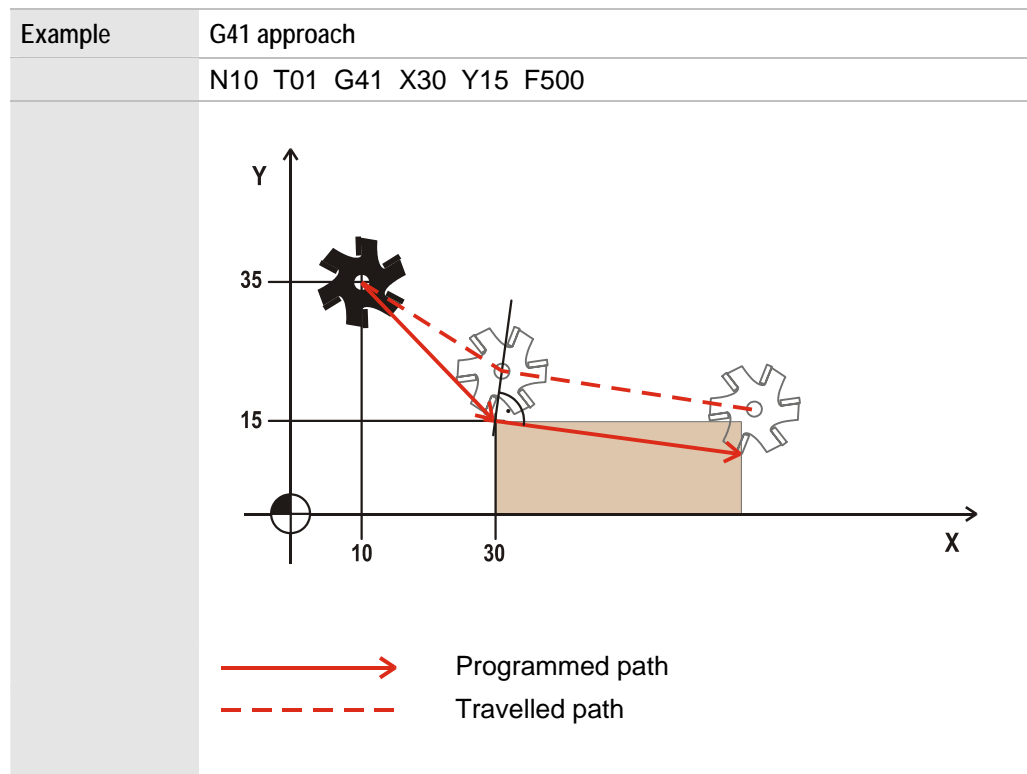
- If there are several sequential NC records without motion the program may stop **without** an error message. In this case the number of NC records without motion must be reduced.
- Sequential NC records must not contain identical coordinates (Error 0x21300005).
- The programmed radius of the workpiece contour must be greater than the tool radius.
- On inside corners ensure that the tool fits into the corner (Error 0x21300003).
- Tool and second tool function cannot be changed.
- The machining plane cannot be altered.
- G39 or a function containing an implicit G39 therefore cannot be used

You may have to deselect tool radius compensation with G40.

Example

G41 and G42



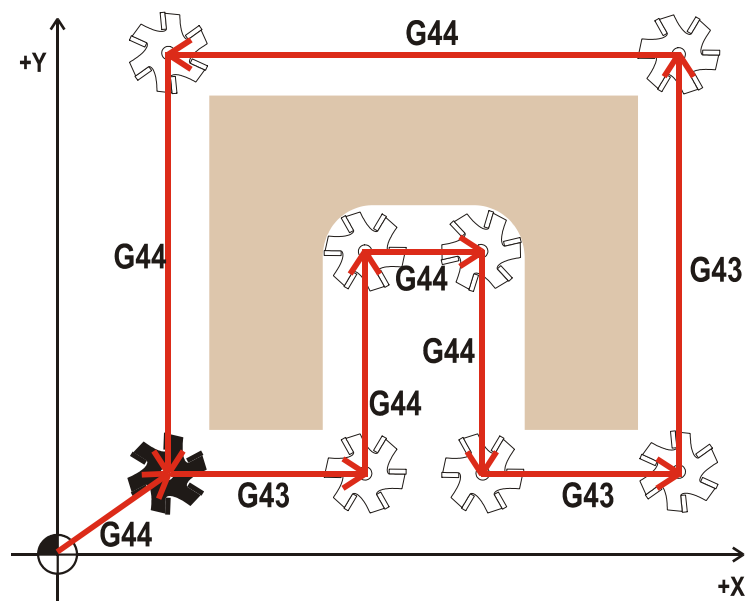


3.18

G43 / G44 Tool Radius Compensation, Positive/Negative

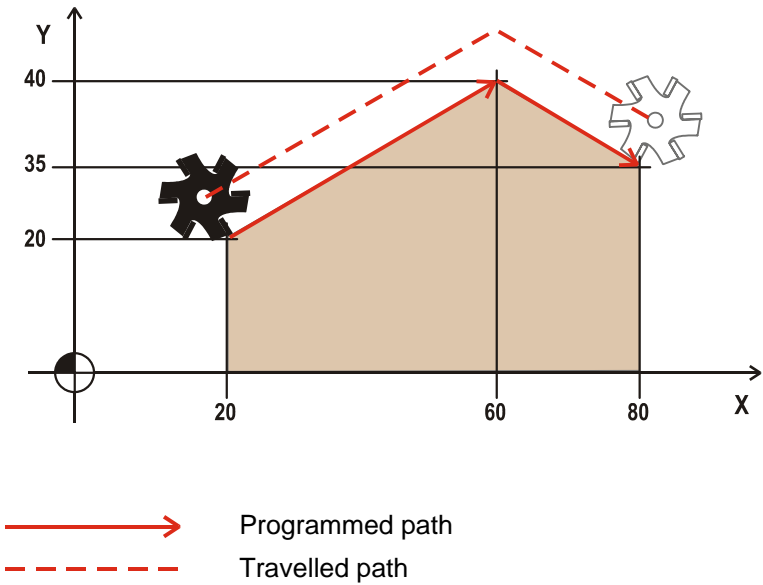


G43	Tool radius compensation positive
G44	Tool radius compensation negative
Format	G43 G44
Explanation	Tool radius compensation parallel to coordinate axes
Note	Tool selection, plane selection and restrictions in programming are identical to functions G41 and G42.

Example



3.19 G50 Tool Radius Compensation Without Transition Contour

G50	Tool radius compensation between straights without transition radius
Format	G50
Explanation	No transition radius is inserted at a straight-straight transition on an outside corner.
Note	The start and end coordinates are recalculated. G50 is effective record by record.

Example	
	N60 G41 X20 Y20
	N60 G50 X60 Y40
	N70 X80 Y35
	 <p>  Programmed path  Travelled path </p>

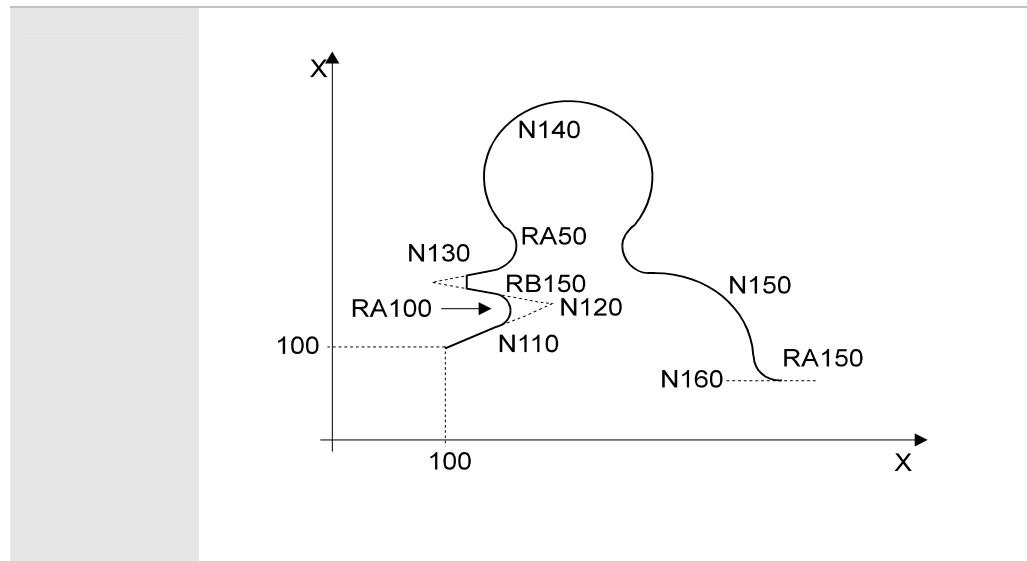


3.20 G45/G46 Path Feedrate Compensation

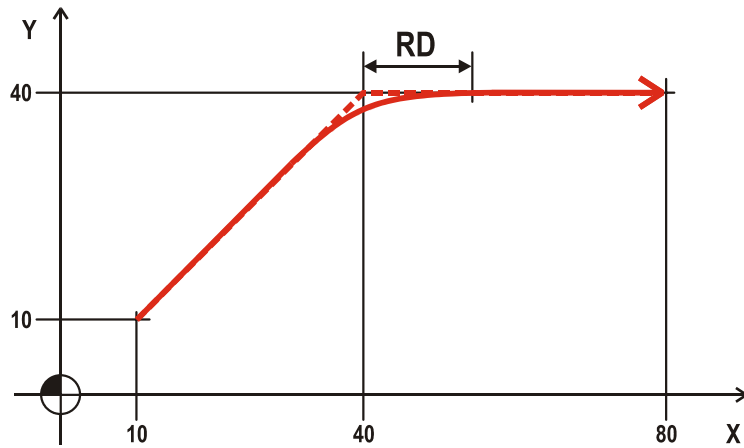
G45	Switch off path feedrate compensation
G46	Switch on path feed rate compensation
Format	G45 G46
Explanation	Path feedrate is calculated on the programmed contour, not in relation to the tool centre. The resulting velocity is restricted to the range 50 % to 200 % of programmed velocity. E.g. transition radii on outside corners are executed at 200 %.
Note	G46 is effective only with active tool radius compensation. G45 is the default setting.

3.21 RA, RB, RD, RF Rounding

RA	Rounding with transition radius between arc and straight line
RB	Rounding with chamfer between straight lines
Format	RAnnnn RBnnnn nnnn = decimal number, radius / chamfer length
Note	For RA and RB a working plane must be selected (e.g. G18) for XZ plane. G39 or a function resulting in an implicit G39 must not be programmed immediately after or in the following record.
Example	
	N100 G1 F100 X1000 Y100
	N110 X900 Y250 RA100
	N130 X300 Y550 RB150
	N130 X700 Y650 RA50
	N140 G2 X1020 Y650 RC-330 RA100
	N150 X1370 Y100 RC330 RA150
	N160 G1 X1700



RD	Rounding with parabola between straight lines
Format	RDnn
Explanation	A parabola is inserted in the straight-straight transition.
Note	<p>Applies to any axes, between two straight lines (G0/G1) without velocity reduction (G64).</p> <p>The parameter indicates the distance from the start and end positions of the inserted parabola to the vertex.</p> <p>If RD = 0 RD will not be executed.</p> <p>If the value of RD is greater than 40 % of the path length of one of the two NC records, RD will be limited to 40 % of the path length of the shorter record.</p> <p>The path feedrate can be specified as percent of the path feedrate programmed in F using FFnnn. See example 2. If ramp type 2000 (ACC2100) this path feedrate will be achieved at the start of the transition record.</p> <p>G39 or a function resulting in an implicit G39 must not be programmed immediately after, or in the following record.</p>
Example 1	
	N10 G0 X10 Y10
	N20 G1 G64 X40 Y40 RD20 F1000
	N30 X80



Example 2	Influencing velocity with FF	
	N10 G1 X0 Y0 F2000	Start position, path feedrate 2000 mm/min
	N20 G64 X20 Y100 RD20 ACC2100 FF40	Smoothing with RD, path feedrate 40 %
	N30 X40 Y0	Continue with 100 %

RF	Axis specific rounding with smooth acceleration																									
Format	RFxnn x = axis letter nn = feed rate (the max. axis acceleration will be reached with this)																									
Explanation	<p>The RF function is self-maintaining.</p> <p>The RF feed rate may be greater than the programmed feed rate. In this case the axis will not reach the permissible acceleration. The movement is smoother.</p> <p>If the programmed feed rate is greater than the RF feed rate, the permissible acceleration will be exceeded (G64) or automatically reduced (G62).</p> <p>Deactivate RF with RFx 0 command for one axis (x = axis letter) or RF 0 for all axes.</p>																									
Note	<p>Use the RF function only with G1.</p> <p>If RF is active, RA, RB and RD can not be used.</p> <p>RF is not usable if the robot transformation is active.</p> <p>RF needs calculation time for axis with no movement also. Therefore use RF referring to axes only when necessary.</p> <p>RF is effective only with G62 and G64.</p> <p>If G61 and G64 are programmed, altering the RF parameters remains self-maintaining.</p> <p>With G9 and G39 RF is depressed for this record.</p>																									
Example	<table> <tr><td>N100</td><td>G64 RFZ2000 F2000</td><td></td></tr> <tr><td>N500</td><td>C20 Z5</td><td></td></tr> <tr><td>N600</td><td>C20 Z15</td><td></td></tr> <tr><td>N700</td><td>G61 C20 Z12</td><td>(RF N700 N800 no active)</td></tr> <tr><td>N800</td><td>G64 C20 Z10</td><td>(RF N800 N900 no active)</td></tr> <tr><td>N900</td><td>G9 C20</td><td>(RF N900 N1000 no active)</td></tr> <tr><td>N1000</td><td>C20 Z15</td><td>(RF N1000 N1100 no active)</td></tr> <tr><td>N1100</td><td>G61 C20</td><td></td></tr> </table>		N100	G64 RFZ2000 F2000		N500	C20 Z5		N600	C20 Z15		N700	G61 C20 Z12	(RF N700 N800 no active)	N800	G64 C20 Z10	(RF N800 N900 no active)	N900	G9 C20	(RF N900 N1000 no active)	N1000	C20 Z15	(RF N1000 N1100 no active)	N1100	G61 C20	
N100	G64 RFZ2000 F2000																									
N500	C20 Z5																									
N600	C20 Z15																									
N700	G61 C20 Z12	(RF N700 N800 no active)																								
N800	G64 C20 Z10	(RF N800 N900 no active)																								
N900	G9 C20	(RF N900 N1000 no active)																								
N1000	C20 Z15	(RF N1000 N1100 no active)																								
N1100	G61 C20																									



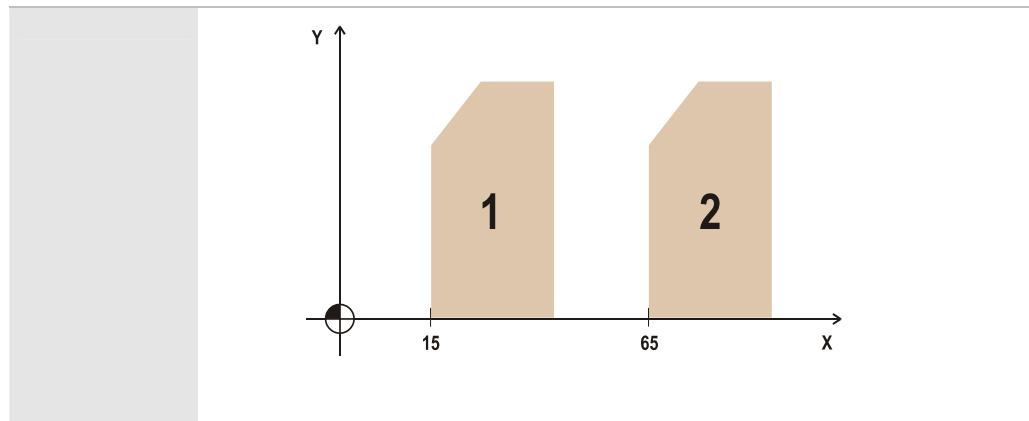
3.22 G53 to G59 Zero Point Offset

G54	Zero point offset 1 (parameters starting R10001)
G55	Zero point offset 2 (parameters starting R10101)
G56	Zero point offset 3 (parameters starting R10201)
G57	Zero point offset 4 (parameters starting R10301)
G58	Zero point offset 5 (parameters starting R10401)
G59	Zero point offset 6 (parameters starting R10501)
G53	Deselect zero point offset
Explanation	There are 6 zero point offsets (G54 to G59), which are normally used to describe the workpiece zero point. With each zero point offset the zero point can be displaced for all axes simultaneously.
Note	The zero point offset values are saved on R parameters. They can be written and read by the NC program. The R parameters have fixed assignments to the G words and the axes. (See also arithmetic parameters Zero Point Offsets R10001 to R10564.) Example for G54: R10001 = 1st axis, R10002 = 2nd axis, ... R10064 = 64th axis Functions G54 to G59 cancel each other. Functions G54 to G59 and G92 are executed simultaneously. Zero point offsets G54 - G59 and reference point offset G92 are deselected with G53. Deselection with G53 is effective record by record, self-maintaining deselection can be set (Q38 bit 6 = 1). Zero point offset is also deselected with M30.



If zero point offset is selected with motions the zero point offset is taken into consideration for the target coordinates.
If a zero point offset is selected without motion only the displayed values for axis set and actual positions will be converted.

Example	Specification: X-axis is first axis
	N10 G54 X0 G0 B% 4711 (Processing part 1 R10001=15)
	N20 G55 X0 G0 B% 4711 (Processing part 2 R10101=65)
	N30 G53 X0 G0 (Deselect with motion)

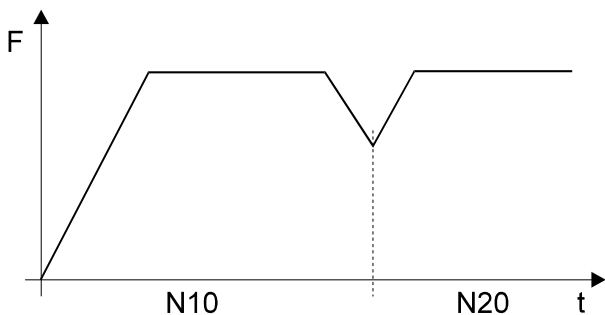




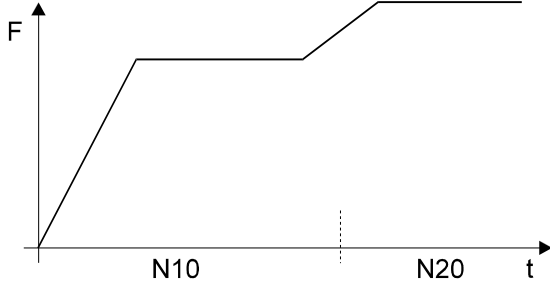
3.23 G61, G64 Smoothing

G61	Smoothing 1
Format	G61
Explanation	Record change occurs when the set position is reached (set-actual deviation = 0).
Note	<p>The axes follow the position values from the controller, displaced by the following error. Record change occurs regardless of following error when the set position of each axis is equal to the programmed coordinate.</p> <p>G61 can be deselected with G60 or G64 or overwritten record-by-record by G9.</p>

Example	#
	N10 G61 G1 X1000 F1000
	N20 X2000



G64	Smoothing without loss of velocity
Format	G64
Explanation	Record change occurs without braking ramp if set-actual deviation = 0. And residual travel on the interpolator is taken into the next record, so there is no loss of velocity.
Note	<p>Acceleration monitoring is activated at the same time as G64. If necessary, it reduces the path feedrate so that none of the involved axes exceeds the maximum acceleration set in Q.025 / 26. This applies both to discontinuous transitions (corners) and to arcs (small radii).</p> <p>If G64 is selected waiting functions (WA, WN, TI) should not be used, because they prevent acceleration monitoring.</p> <p>Record change is executed independently of PLC enable for M and T functions.</p> <p>G64 can be deselected with G60 or G61 or overwritten record-by-record by G9.</p>

Example	#
	N10 G64 G1 X1000 F1000
	N20 X2000 F1200
	

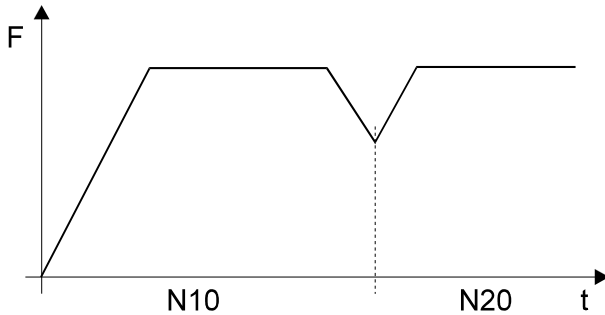


On non-tangential contour transitions (e.g. angle between 2 consecutive straight lines $> 7^\circ$) G64 may act like G61 due to the acceleration monitoring.



3.24 G62 Record change with acceleration monitoring

G62	Record change with acceleration monitoring
Format	G62
Explanation	Record change occurs if set-actual deviation = 0. And residual travel on the interpolator is taken into the next record, so there is no loss of velocity.
Note	<p>If necessary, it reduces the path feedrate so that none of the involved axes exceeds the maximum acceleration set in Q.025 / 26 respectively the acceleration set by ACC function. That is permissible for all ramp type functions (ACC0100, ACC1100, ACC2100, ACC3100).</p> <p>This applies to discontinuous transitions (corners), transitions with RD (between G0 or G1) and small arcs (G02, G03, interpolated records in tool nose radius compensation). In this case the velocity will be reduced so that no exceeds the maximum acceleration.</p> <p>If G64 is selected waiting functions (WA, WN, TI) should not be used, because they prevent acceleration monitoring.</p> <p>Record change is executed independently of PLC enable for M, H and T functions.</p> <p>Calls of subroutines and returns are possible without loss of velocity if : B%xxx respectively M17 is programmed in the last path record and robot transformations are inactive.</p> <p>G62 can be deselected with G60 or G61 or overwritten record-by-record by G9.</p>

Example	
	N10 G62 G1 X500 Y200 F1000
	N20 X700 Y500
	

3.25 G63 Tapping without compensating chuck

G63	Tapping without compensating chuck as single record
Format	G63
Explanation	
Note	<p>Function G63 requires a spindle with a positioning transducer. The spindle can be operated as a controlled or uncontrolled spindle. Before G63 is called in the record the lead axis must be declared via the \$33 function.</p> <p>Before the first G63 record is called the direction of rotation and speed of the spindle must be programmed. Record change must not occur until the spindle is turning in the programmed direction (M bit acknowledge). When G63 is active right-hand or left-hand thread is decided by the direction of rotation of the spindle and the travel direction. A later alteration is not possible.</p> <p>If the direction of rotation of the spindle changes the axis returns to the record start position and stops there.</p> <p>Thread pitch is programmed using the auxiliary coordinates I, J, K. With a controlled spindle G32 can be used for interpolating in place of G63.</p>

Example	#
	N10 G0 \$33 Z0 M00 M03 S500
	N20 G63 Z200 I2 M03
	N30 Z220 M05
	Reversing record: The programmed thread depth must not be reached. Record change occurs when spindle stops.
	N40 Z20 M04
	N50 Z0 M05 BN20-



Locking with G63:
 Override is set to 100%.
 Stop key is locked.
 In single record mode stop is not until after the last G63 record.
 Mode change is not possible until after the last G63 record.
 With G63 and NC reset the spindle stops and the CNC program is deselected. Gear coupling remains engaged and all modes apart from automatic are locked. G63 and thread pitch remain self-maintaining. Spindle is set to M05 S00.
 Deselect G63 locking:
 If an error occurs and the return program is not possible, e.g. borer is broken off, G63 locking can be cancelled by programming G0 or G1. After G0/G1 has been processed the RESET key must be operated.



After programming new programs with G63 or program cycles with G77 we recommend running the program without the workpiece.
 If the "Thread Error" message appears at the thread return point the program will stop. The "Thread Error" message appears if the spindle cannot stop within the calculated distance.
 In this case the thread depth must be corrected in the program.



3.26 G66 Synchronization of IPO support points

G66	Synchronization of IPO support points
Format	G66
Explanation	<p>With G66 the velocity over several records will be corrected so that the end point of path is reached in IPO cycle. Therefore beats in program loops without stop are avoided.</p> <p>Mit G66 wird die Geschwindigkeit über mehrere Sätze so korrigiert, dass der Satzendpunkt im IPO-Takt erreicht wird. Dadurch können Schwebungen bei Programmschleifen ohne Halt vermieden werden.</p>
Note	<p>G66 should be programmed only once.</p> <p>Is a stop in the program loop, G66 must not be programmed.</p> <p>G66 sollte nur einmal programmiert werden.</p> <p>Bei einem Stop der Achsen in der Programmschleife ist G66 überflüssig.</p>

3.27 G67 Special Function for Oscillating

G67	Special function for oscillating
Format	G67
Explanation	Influences reversing behaviour of oscillating axis or first feed axis.
Note	<p>G67 is effective record-by-record and has no effect without \$40 to \$44.</p> <ul style="list-style-type: none"> Reversing behaviour without G67. <p>Oscillating axis remains at reversal point until respective feed axis has completed its infeed increment and selected exact position condition is met. Infeed begins when the oscillating axis is at the reversal point and meets the exact position condition.</p> <p>The same conditions apply for the first feed axis if a second feed axis is programmed.</p> <ul style="list-style-type: none"> Reversing behaviour with G67 <p>The oscillating axis initiates the feed process by reaching the reversal point and meeting the selected exact position conditions, but changes direction before feed has been completed. The same conditions apply for the first feed axis if a second feed axis is programmed.</p>

3.28 G70 and G71 Inch/Metric Switching

G70	Dimensions in inches
Format	G70
G71	Dimensions in mm
Format	G71
Note	<p>Inch/mm switching relates only to the programmed coordinates.</p> <p>Zero point offsets, tool compensations and system parameters are not converted. They are always interpreted according to the machine data setting.</p> <p>The program sequence figure in the monitor displays set and actual values and set-actual deviation in the selected system of units (mm or inches).</p> <p>Actual values and coordinates are displayed in the selected system of units. The internal parameters are saved in floating point format, but displayed in the IPO resolution, i.e. with G70 in inches, with G71 in mm. You can set conversion of F, S in configuration parameter Q25 bit 4.</p>

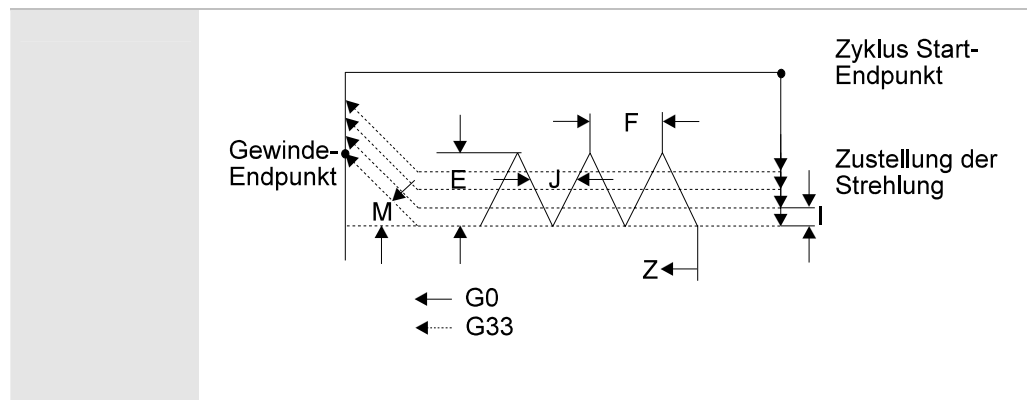


G70/G71 is self-maintaining even through controller on/off.



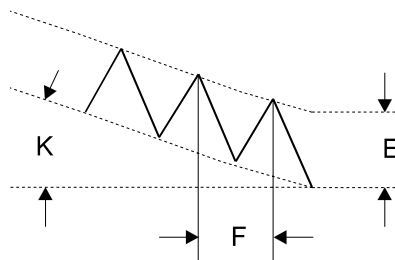
3.29 G76 Thread Cutting Cycle

G76	Thread cutting cycle	
Format	G76 Z X	
	Z, X = axis letters	
Explanation	Thread cutting in cycle	
Note	<p>The syntax for thread cycle is programmed in 2 records. Here an example for thread in Z direction: N.. X.. Z.. \$33 Z</p> <p>X and Z define the start and end position of the cycle (outside the workpiece). \$33 Z defines the pitch in Z direction</p> <p>N... G76 X.. Z.. F.. E.. H.. [I..] [J..] [K..] [M..]</p> <p style="padding-left: 40px;">X.. Z.. Thread end position F Thread pitch E Thread depth H Number of cuts I (opt.) Final machining allowance J (opt.) Thread angle in degrees K (opt.) Conical angle in degrees M (opt.) Overrun angle in degrees</p> <p>Parameters I-M are optional. If not programmed default values are: I = 0.0 J = 0.0 K = 0.0 M = 0.0</p> <p>External thread if cycle start (X) > thread end position (X) Internal thread if cycle start (X) < thread end position (X) Active preparatory function at cycle end is G0 All parameters can be parameterized with R parameters</p>	
Example	%1	
	N10 T01 M03 S700	(Switch spindle on, speed 700 r.p.m., select tool 1)
	N20 \$33 Z	(Z is the lead axis)
	N30 G0 X38 Z0	(Home to start position)
	N40 G76 X20 Z-50 F2.5 E5 H5 I0.5	(Thread depth 5 mm, 2.5 mm pitch)
	N70 M17	(End subroutine)



Gewinde-Endpunkt = Thread end position, Zyklus Start-Endpunkt = Cycle start and end position, Cut infeed

Example	Conical thread
	%1
N10 T01 M03 S700	(Switch spindle on, speed 700 r.p.m., select tool 1)
N20 \$33 Z	(Z is the lead axis)
N30 G0 X38 Z0	(Home to start position)
N40 G76 X20 Z-50 F2.5 E5 H5 I0.5 K20	(Thread conical 20 degrees)
N70 M17	(End subroutine)



If $J > 0$ then:

Feed direction is half thread angle, so that from second cut on only one cutting edge is engaged.

$$\Delta Feed = CurrentDepth \cdot \tan\left(\frac{J}{2}\right)$$

The above formula ensures constant-volume cut segmentation, so that cutting forces are as constant as possible.

If final machining allowance $I = 0$ (default setting) a non-cutting pass will be carried out.

Cut segmentation

$$CurrentDepth = \frac{E - 1}{\sqrt{H - 1}} \cdot \sqrt{CurrentCutNumber}$$

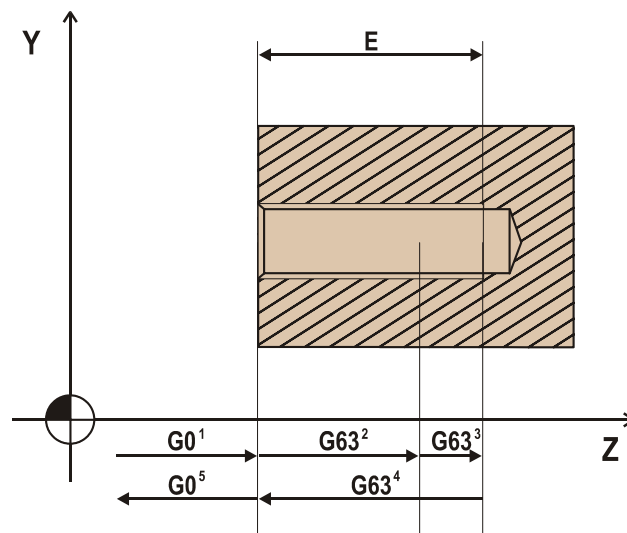
$$= E \text{ if } (H = 1) \text{ or last cut}$$



3.30 G77 Tapping without compensating chuck cycle

G77	Tapping without compensating chuck cycle
Format	see note below
Explanation	G77 controls the complete G63 sequence
Note	<p>G77 contains the following working steps:</p> <p>Z defines the cycle start position (Start position outside workpiece)</p> <p>G77 Z.. E.. [J..] F.. [S..] [TI..]</p> <p>Z.. Thread start position E Thread depth F Thread pitch J (opt.) Approach angle S (opt.) Return speed TI (opt.) Dwell time when reversing</p> <p>Active preparatory function at cycle end is G0.</p> <p>All parameters can also be parameterized with R parameters</p> <p>The syntax for thread cycle is programmed in 2 records.</p>

Example	
	N10 G0 Z10 M4 S250
	N20 G77 Z13 E25 J0 F2 S200 TI0,5
	N30 G0 Z10
	N40 M5 M17



¹⁾ Approach with G0 to acceleration path before start of thread

²⁾ Switch on G63 contouring compensation

G63 tapping to spindle braking path

³⁾ G63 tapping with decelerating spindle

Record change at spindle stop. The thread depth must not be reached. (Error message thread error)

- 4) Return to thread start at return speed and
5) Decelerate at start position



After programming new programs with G63 or program cycles with G77 we recommend running the program without the workpiece. If the "Thread Error" message appears at the thread return point the program will stop. The "Thread Error" message appears if the spindle cannot stop within the calculated distance. In this case the thread depth must be corrected in the program.

3.31 G80 to G89 Machining Cycles G80 to G89

G80	Cancel machining cycle	
G81	Machining cycle 1	(: %99999981)
G82	Machining cycle 2	(: %99999982)
G83	Machining cycle 3	(: %99999983)
G83	Machining cycle 4	(: %99999984)
G85	Machining cycle 5	(: %99999985)
G86	Machining cycle 6	(: %99999986)
G87	Machining cycle 7	(: %99999987)
G88	Machining cycle 8	(: %99999988)
G89	Machining cycle 9	(: %99999989)
Format	G81	
Explanation	A machining cycle is carried out after execution of each record containing a motion.	
Note	The machining cycle is programmed as a subroutine with the corresponding program number (%99999981 - %99999989). The machining cycle call is self-maintaining. I.e. once programmed the machining cycle is executed after each motion record until it is overwritten by calling another machining cycle or cancelled by programming G80.	
Example		
	N100 G00 X50 Y50 G81	Call machining cycle %99999981, execute after reaching programmed position.
	N110 X100 Y100	Execute %99999981 after reaching the programmed position.
	N130 Y120 G80	Cancel programmed machining cycle, %99999981 is <u>no longer</u> executed.



3.32 G90, G91 Measurements Absolute/Incremental

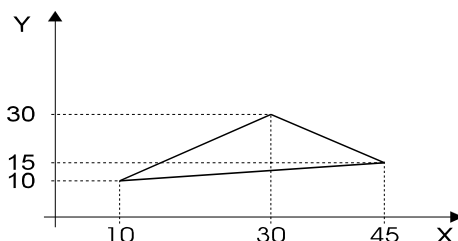
You can program axis-specific measurements with \$90/\$91.

G90	Absolute measurements
Format	G90
Explanation	All measurements relate in absolute terms to the current zero point.
Note	<p>If no zero point offset is active, this is the zero point defined by the reference point coordinate. It can be altered by zero point offset G54 to G59, G92 or with Zero Overlays R10601 to R10664.</p> <p>G90 is the default setting.</p> <p>This function can also be programmed for individual axes (see \$90)</p>

Example	
	N10 G0 G90 X10 Y10
	N20 G1 X30 Y30 F1000
	N30 X45 Y15
	N40 X10 Y10

G91	Incremental measurements
Format	G91
Explanation	The programmed value corresponds to the distance to be travelled.
Note	<p>The auxiliary coordinates (I, J, K) for circle programming are not affected by G90/G91. The setting in the configuration parameter always applies.</p> <p>Programming of G91 can be prevented by setting bit 0 in configuration parameter Q25.</p> <p>This function can also be programmed for individual axes (see \$91).</p>

Example	
	N10 G0 G90 X10 Y10
	N20 G1 G91 X20 Y20 F1000
	N30 X15 Y-15
	N40 X-35 Y-5



3.33 G92 Reference Point Offset

G92	Reference point offset	
Format	G92 X Y X, Y any axis letters	
Explanation	With G92 you can set the reference point for individual axes.	
Note	<p>In the rest of the program all axis coordinates used will relate the coordinates set with G92.</p> <p>The difference between actual value and reference point offset is entered in R parameters and can be read by the NC program. (See arithmetic parameters R10701 to R10764 Reference Point Offset.)</p> <p>A reference point offset programmed with G92 is inactive as long as G53 is active.</p> <p>The actual value memory is deleted with M30 or RESET.</p> <p>Functions G54 to G59 and G92 are executed simultaneously.</p>	
Example		
	N10 G00 X100 Y7.5	
	N20 G92 X0 Y100	<p>X-axis is at 100 and is set to zero.</p> <p>Y-axis is at 7.5 and is set to 100.</p>



3.34 G94, G95 Evaluation of F Word

G94	Feedrate/path feedrate in mm/min
Format	G94 F
Explanation	The feedrates/path feedrates programmed with the F word are calculated in mm/min.
Note	G94 is the default setting. Example see G95

G95	Feedrate in mm per revolution of main spindle
Format	G95 F
Explanation	The path feedrate programmed with the F word is interpreted as mm/revolution of the main spindle. The resulting path feedrate in mm/min is the product of speed (S) and feedrate (F).
Note	This feedrate evaluation mode requires a spindle with an actual value system.

Example	
	N10 G1 X10 F500 M3 S1000 (X-axis moves at 500 mm/min, spindle speed 1000 r.p.m.)
	N20 G95 X30 F1.5 (G95 X-axis moves at 1.5 mm per spindle revolution. Resulting path feedrate is 1500 mm/min)
	N30 G94 X40 F500 (G94 feedrate in mm/min again)
	N50 M5 M17

3.35 G96, G97 Evaluation of S Word

G96	Constant cutting speed
Format	G96 S
Explanation	<p>The S word is interpreted as the circumferential speed in m/min. The radius associated with the circumference is formed from the actual value of an axis specified with \$34. You can set a radius offset for these axes in configuration parameter Q.019.</p> <p>The current radius is given by actual position – tool compensation – Q.019.</p>
Note	<p>If no radius axis is specified, the radius will be taken from parameter Q.019 of the spindle.</p> <p>The cutting speed can also be given in m/sec. In this case set bit 1 of parameter Q38.</p> <p>See also Programming Spindle Speed S.</p>

G97	Spindle speed in r.p.m.
Format	G97 S
Explanation	The S word is interpreted as a constant speed in r.p.m.
Note	<p>Programming of a constant spindle speed can be locked by setting bit 1 in configuration parameter Q25.</p> <p>See also Programming Spindle Speed S.</p>



3.36 G98, G99 Self-maintaining Preparatory Functions in Subroutines

When a subroutine is called the self-maintaining preparatory functions remain effective. If self-maintaining preparatory functions are programmed in a subroutine you can use G98 and G99 to decide whether they will remain effective after a return, or whether the previously valid preparatory functions will be restored.

G98	Use self-maintaining preparatory functions programmed in the subroutine after return to main program
Format	G98
Note	The preparatory conditions activated in the main program will not be restored after return from subroutine. For the sake of clarity G98 should only be programmed in main programs. G98 is the default setting.

G99	Do not use self-maintaining preparatory functions programmed in the subroutine after return to main program
Format	G99
Note	The preparatory conditions activated in the main program will be restored after return from subroutine. For the sake of clarity G99 should only be programmed in main programs.

Example	
	N10 G0 X100 G99
	N20 X200
	N30 B%9000 Subroutine number
	N40 X220 Axis moves with G0 and G90
	N50 M30
	#9000
	N10 G1 G91 X10 F100 Self-maintaining preparatory functions G1/G91 are not effective after return to main program.
	N20 M17

4 \$ Functions

The \$ functions are additional preparatory functions for expanding the standard preparatory functions.

The additional preparatory functions are arranged in function groups.

Only one function from each group can be active.

Normally the functions remain active until they are deselected by another function from the same group.

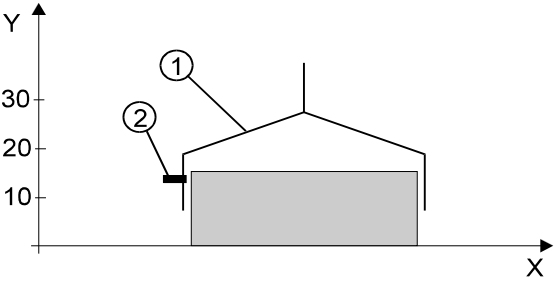
In individual cases a function is active for only one record (property = S). Some functions are default settings (property = D).

Group	Properties D = Default setting S = Active for 1 record		Meaning
1	S	\$1	Stop axis motion
	S	\$53 - \$54	Abort motion
2	S	\$20	Handwheel enable for velocity superposition
	S	\$21	Handwheel enable for path superposition
3		\$23	Switch on internal follow-up operation
		\$24	Switch on follow-up operation
		\$25	Switch off follow-up operation
4		\$26	Switch on independent axis with individual feedrate
		\$27	Switch off independent axis with individual feedrate
		\$28	Integrate independent axis in record change
		\$29	Do not integrate independent axis
5		\$31	Switch on synchronous operation
		\$32	Switch off synchronous operation
6		\$33	Lead axis for thread cutting
7		\$34	Radius axis for v = constant
8			Path length calculation
		\$38	Switch on contouring axis in IPO context
		\$39	Switch off contouring axis in IPO context
9		\$40	Switch oscillation off
		\$41	Switch on oscillation with continuous infeed
		\$42	Switch on oscillation with infeed on both sides
		\$43	Switch on oscillation with infeed right
		\$44	Switch on oscillation with infeed left
10		\$47	Alternative machining plane
11		\$48	Give back system axis
12		\$90	Absolute measurements
		\$91	Incremental measurements



4.1 \$1 Stop Axis Motion Without Ramp

\$1	Stop axis motion without ramp
Format	\$1 X F = feedrate, X = any axis letter
Explanation	Axis motion is interrupted when the PLC signal provided for this purpose is active. Assignment to PLC signal: <i>cncMem.axSect[n].FlgP2N.bAxStop</i> (n=axis number)
Note	The motion is aborted immediately in the interpolation cycle, without velocity ramp. The following error will be corrected. If the axis interpolates with other axes interpolation will be aborted on all axes. The NC axis assigned to the input signal must be programmed directly after the preparatory function (\$1 X...). The coordinate value of the NC axis indicates the maximum permissible travel. If the PLC signal is not active during the programmed path record change will occur when the programmed coordinate value is reached. An implicit G39 is executed in this function. For stopping an axis motion with an interrupt see \$53 and \$54.

Example	
	N100 G0 Y20
	N110 G1 \$1 Y5 F500
	N120 E1 = 0 BN ... (no part)
	N130 SE ... (close gripper)
	N140 G0 Y100
	
	1 Gripper, 2 Range finder

4.2 \$20 Handwheel Enable for Velocity Superposition

\$20	Handwheel enable for velocity superposition
Format	\$20 X X = any axis letter
Explanation	You can alter the velocity of the specified axes with a handwheel. The superposition is added to the programmed velocity.
Note	Handwheel pulses can be saved in shared RAM variable <i>cncMem.axSect[n].wrdP2N.IValHdWhl</i> . The evaluation of the pulses can be saved in <i>cncMem.axSect[n].wrdP2N.fRateHdWhl</i> .

4.3 \$21 Handwheel Enable for Path Superposition

\$21	Handwheel enable for path superposition
Format	\$21 X X = any axis letter
Explanation	You can alter the programmed end position of the specified axes with a handwheel. The superposition is added to the end coordinate and to the programmed velocity.
Note	Handwheel pulses can be saved in shared RAM variable <i>cncMem.axSect[n].wrdP2N.IValHdWhl</i> . The evaluation of the pulses can be saved in <i>cncMem.axSect[n].wrdP2N.fRateHdWhl</i> .

4.4 \$23 Internal Follow-up Operation On

\$23	Internal follow-up operation on
Format	\$23 X X = any axis letter
Explanation	The specified axes go into internal follow-up operation. The axes remain in position control and can, for example, be moved via the table interpolator (TIPO PS04) or via PLC position superposition. The CNC follows the axis position in the display. A set position for this axis must not be programmed as long as \$23 is active.
Note	This function is independent of selection of a table interpolator. So the axis can first be moved without \$23 using the TIPO or the PLC. With \$23 (or as before with \$24) the resulting position difference must be calculated back before the TIPO is deselected. \$23 is deselected with \$25.



4.5 \$24 Follow-up Operation On

\$24	Follow-up operation on
Format	\$24 X X = any axis letter
Explanation	The specified axis goes into internal follow-up operation. Follow-up operation is used to temporarily interrupt position control, controlled by the program. This is always necessary when the axis is mechanically jammed or displaced by external factors, for example on an injection moulding machine by the discharger when parts are removed.
Note	The position control circuit is opened, the "controller enable" relay releases, all increments in the actual value system are recorded and transferred to the set position. A set position for this axis must not be programmed as long as \$24 is active.
Example	See \$25

4.6 \$25 Switch Off Follow-up Operation

\$25	Switch off follow-up operation
Format	\$25 X X = any axis letter
Explanation	Cancel programmed follow-up operation for one axis.
Note	An implicit G39 is executed in this function.
Example	X-axis is displaced by an external discharger and must therefore be taken out of position control.
	N100 G0 X100 Gripper to part
	N110 \$24 X Follow-up operation on
	N120 SE1 Move discharger forward (request to PLC)
	N130 WN1 Discharger retracted (acknowledgement from PLC)
	N140 G0 \$25 X50 Follow-up operation off, gripper with part back

4.7 \$26 Exclude Axes from Interpolation Context

\$26	Exclude axes from interpolation context
Format	\$26 X FX FX = feedrate, X = any axis letter
Explanation	With this \$ function you can exclude individual axes from the interpolation context and from the record change criterion (\$29). They become "independent" axes.
Note	The selected axes move at the axis-specific feedrate Fx in mm/min, regardless of the path feedrate. Independent rotary axes move at feedrate F"axis name" in °/min. Record change occurs when all axes in the NC subsystem meet the valid exact position condition. Independent axes do not normally reach their programmed end position at the same time. Record change without loss of velocity with G64 should not be used for independent axes.
Example	X and Y interpolate on a straight line, Z is independent. N10 G1 X100 Y100 F500 \$26 Z500 FZ1000 See also \$29.



If \$26 and \$29 are used together and the G condition for feedrate changes in one of the following records, e.g. from G1 to G0, while the independent axis is still moving, G0 applies to all axes in the system. The feedrate of the independent axis will switch to the axis-specific G0 rapid feed velocity.

**4.8 \$27 Include Independent Axes in Interpolation Context**

\$27	Include independent axes in interpolation context
Format	\$27 X X = any axis letter
Explanation	The independent axis is reintegrated in interpolation and record change. \$27 cancels function \$26.
Note	If \$29 is active \$28 must be set in the previous NC record.
Example	See \$29

4.9 \$28 Include Independent Axis in Record Change

\$28	Reintegrate independent axis in record change
Format	\$28 X X = any axis letter
Explanation	The independent axis is reintegrated in the record change, but not in the interpolation context.
Note	Record change occurs when all interpolating axes <u>and</u> the independent axes meet their exact position condition. This word is the default setting, or cancels function of \$29.
Example	See \$29

4.10 \$29 Do Not Include Independent Axis in Record Change

\$29	Do not include independent axis in record change
Format	\$29 X X = any axis letter
Explanation	
Note	Record change occurs when the interpolating axes meet their exact position condition, regardless of the position of the independent axis. If the independent axis has not yet reached its end coordinates the motion of the independent axis will continue in the next record. If program ends with M17 or M30 the independent axis will stop too, regardless of its position.

Example	
	N100 G1 X100 Y100 Z100 F1000
	N110 G1 X120 Y80 F500 \$26 Z200 FZ50 \$29 Z
	N120 X140 Y60
	N130 X160 Y40
	N140 \$28 Z
	N150 G1 X100 Y100 \$27 Z100 F1000
	N160 X50 Y120 Z90

The Z-axis is excluded from interpolation (\$26) and record change condition (\$29).

Z-axis moves independently of the axes programmed here.

Reintegrate Z-axis in record change (\$28). Record change occurs when Z-axis has reached target position.

Reintegrate Z-axis in interpolation from this record on (\$27).

Path interpolation restarts for the Z-axis.



If \$26 and \$29 are used together and the G condition for feedrate changes in one of the following records, e.g. from G1 to G0, while the independent axis is still moving, G0 applies to all axes in the system. The feedrate of the independent axis will switch to the axis-specific G0 rapid feed velocity.



Function \$29 can only be called after \$26. Cancelling with \$28 must occur at least one record before \$27.

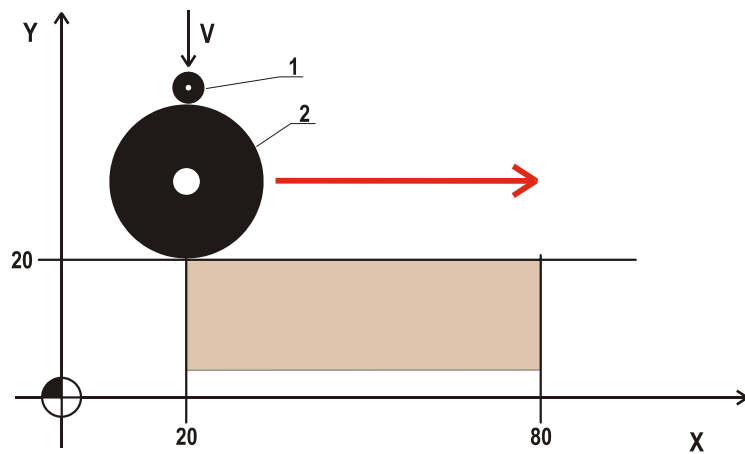
**4.11 \$31 Switch On Synchronous Operation**

\$31	Switch on synchronous operation
Format	\$31 X Y X, Y = any axis letter
Explanation	Synchronous operation allows synchronous operation of several axes to be programmed for a time.
Note	<p>The first axis named after \$31 is the lead axis and all subsequently named axes are the following axes. The motion of the first named axis is also effective for the subsequent axes. If a motion is programmed for the following axes, the motion of the lead axis will be added to this.</p> <p>There is no check on whether the following axes can reach the required velocity.</p> <p>The distance moved by the lead axis does not appear in the actual value display of the following axes. Instead it is saved as an internal zero point offset. The displayed actual value of a following axis contains only the actual position of the distance programmed in this axis.</p> <p>The function is deselected with \$32, M30 or reset. M30 or reset deletes the content of zero point offset, the absolute actual value is displayed.</p> <p>If bit 1=1 is set in Q37, \$32 will delete the content of internal zero point offset (actual value display changes). In this case G39 is executed automatically.</p>
Example	See \$32

4.12 \$32 Switch Off Synchronous Operation

\$32	Switch off synchronous operation
Format	\$32
Explanation	This function cancels \$31.
Note	If bit 1=1 is set in Q37, \$32 will delete the content of internal zero point offset (actual value display changes). In this case the controller executes an automatic \$39.

Example	Continuous dressing of a grinding wheel	
	N100 G0 X20 Y10	Position grinding wheel
	N110 VR1000	Position dressing roll
	N120 \$31 VY	V superposes Y
	N130 \$26 VR1001 FR102 \$29 V	Switch on dressing, V = independent axis
	N140 X70 F100	Grinding
	...	
	N210 G91 \$32 V1	Lift off dressing roll
	N220 G90 \$28 V	V-axis in position
	N230 \$27 V	V-axis in interpolation



1 Dressing roll 2 Grinding wheel

**4.13 \$33 Select Lead Axis for Thread Cutting**

\$33	
Format	\$33 Z Z = axis letter
Explanation	Specify lead axis for thread cutting / tapping with G33, G63.
Note	\$33 is self-maintaining and only has to be programmed again if the lead axis changes.
Example	See G33, G63

4.14 \$34 Select Radius Axis

\$34	Radius axis selection for G96
Format	\$34 X X = any axis letter
Explanation	The actual position of the selected axis enters the main spindle circumferential velocity calculation as the radius.
Note	<p>An additional offset can be entered in Q.019 of the axis selected with \$34. The sign of Q.019 (OFFSET RADIUS) is taken into consideration in calculation of the constant cutting speed. The spindle speed is limited to the set maximum spindle speed.</p> <p>If no radius axis is selected with \$34 Q.019 of the spindle axis will be interpreted as the radius. If Q.019 = 0 no speed will be output. Q.019 can be altered at any time by the PLC program or, if enabled, by the CNC program.</p> <p>The radius determined for calculation is formed from: Radius = actual position – tool compensation – Q.019.</p>

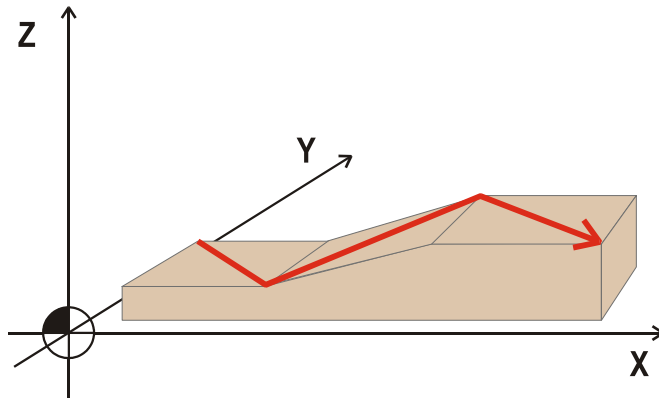
4.15 \$37 Path length calculation

\$37	
Format	\$37
Explanation	<p>With \$37 (alternatively and together with \$38 and \$39) the path length calculation is executed by principle of lead axis.</p> <p>The programmed feedrate is related to the axis with the longest path length (* 1000 / Q1079). Standardized (like \$39) the path length is the root-mean-square value of the included axes.</p>
Note	see also \$38 and \$39

4.16 \$38 and \$39 Axis Selection for Path Feedrate Calculation

\$38	Exclude axes from path feedrate calculation
\$39	Include axes in path feedrate calculation
Format	\$38 X \$39 X X = any axis letter
Explanation	With these functions you can exclude individual axes from the path feedrate calculation.
Note	<p>The affected axes follow in the interpolation context. There is no check on whether the following axes can reach the required velocity.</p> <p>This function has no effect on axes which are involved in circular interpolation with G2/G3 or G12/G13.</p> <p>In the case of helix interpolation the third axis can be excluded from the path feedrate calculation with \$38. Then the path feedrate is effective not on the spatial path but on the flat circular path (projection).</p>

Example	
	N100 G0 X10 Y40 Z10 Start position
	N110 G1 X40 Y10 F100
	N120 X60 Y40 \$38 Z25 Exclude axis Z
	N130 X100 Y10
	N150 G0 \$39 Z30 X0 Y0 Include axis Z





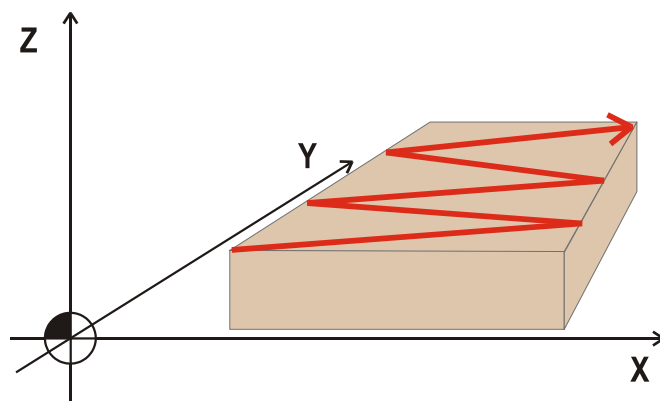
4.17 \$40 Switch Oscillation Off

\$40	Switch oscillation off with sparking out strokes	
Format	\$40 X \$40 Xn Xn = axis letter with specified number of sparking out strokes	
Explanation	Oscillation switched off, the number of sparking out strokes for an oscillating axis can be specified.	
Note	When the end coordinate is reached the axis moves to the next reversal point. From this point the specified number of sparking out strokes is executed. One sparking out stroke is the distance between reversal points. With \$40 the controller inserts an automatic G39.	
Example		
	N10 \$40 X5	Axis X executes 5 sparking out strokes

4.18 \$41 Oscillation With Continuous Infeed

\$41	Oscillation with continuous infeed on one axis
Format	\$41 X Y X, Y = any axis letters
Explanation	The oscillating axis oscillates between the start position (the axis position when the oscillation function is called) and the coordinate programmed in the oscillation record. Infeed is continuous, at path feedrate.
Note	The oscillating axis is always the first axis programmed after the \$ word. The feedrate of the oscillating axis is programmed with F"axis name". The feed axis is the second programmed axis. The target coordinate is programmed. When the oscillate function is selected the oscillating axis automatically becomes an independent axis (corresponding to \$26). When the oscillate function is deselected it returns to the interpolation context. The reversing behaviour of the oscillating axis can be controlled with G67.

Example	
	N100 G0 X0 Y100
	N110 G1 \$41 X200 Y95 FX1000 FY5 Oscillation on
	N120 \$40 X3 Oscillation off with 3 sparking out strokes
	N130 G0 Y150

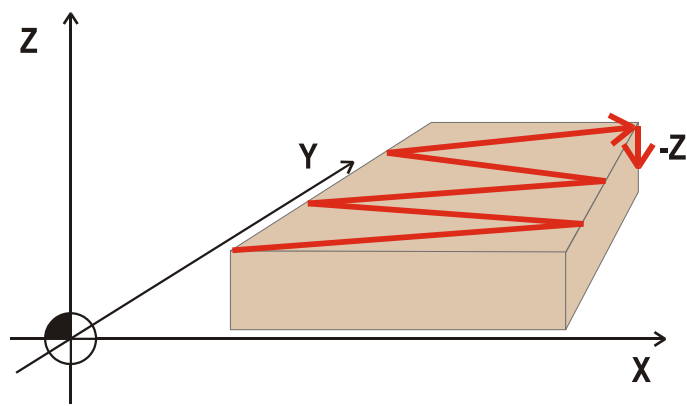


\$41	Oscillation with continuous infeed on one axis and two feed axes
Format	\$41 X Y Z R1001 R2 X, Y, Z = any axis letters R1001, R2 = arithmetic parameter addresses
Explanation	A second feed axis is added to oscillating with one feed axis. This makes it possible to machine surfaces that are broader than the width of the grinding wheel.



Note	<p>The oscillating motion in X is as before. In Y there is continuous infeed until the programmed coordinate is reached. Now Z is fed by the content of R1001 in relation to R2. Y reverses and moves to the opposite reversal point (etc.).</p> <p>Record change occurs when Z has reached the final dimension and the valid exact position condition is met.</p>
------	--

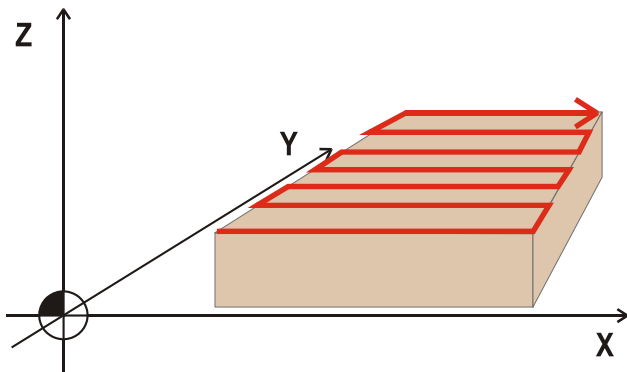
Example	
	N100 G0 Y0 Z100 R1001:= 0.5 R2:= 1
	N110 G1 \$41 X200 Y50 Z95 R1001 R2 FX1000 FY10 FZ500
	N120 \$40 X3 Y0
	N130 G0 Z150



Record N110:	G1	Straight interpolation
	\$41	Select oscillate function with continuous infeed.
	X	Is the oscillating axis. The programmed position is the 1st reversal point. The 2nd reversal point results from the position of X at the start of oscillating.
	Y	Is the 1st feed axis. The programmed coordinate is the reversal point. Infeed in Y is continuous.
	Z	Is the 2nd feed axis. The programmed coordinate is the final dimension. Infeed in Z occurs at the reversal point of Y, dependant on R2.
	R1001	Contains the infeed increment for Z
	R2	controls infeed in Z: R2 =0 Infeed at front (smaller) reversal point of Y. R2 =1 Infeed at both reversal points of Y. R2 =2 Infeed at rear (greater) reversal point of Y.
	FX	Is the feedrate of the oscillating axis
	FY	Is the feedrate of the 1st feed axis
	FZ	Is the feedrate of the 2nd feed axis

4.19 \$42 Oscillating With Infeed at Both Reversal Points

\$42	Oscillating with infeed on one axis at both reversal points
Format	\$42 X Y R X, Y = any axis letter R = arithmetic parameter
Explanation	As \$41 but with infeed at the corresponding reversal points. The respective feed increment is programmed in the arithmetic parameter.
Note	A reversal dwell time can be programmed in the oscillation record with T1. The reversal dwell time starts as soon as infeed has occurred. The oscillating axis remains at the reversal point until the dwell time has expired. The reversing behaviour of the oscillating axis can be controlled with G67.

Example	
	N100 G0 X0 Y100 R1001 := 0,5
	N110 G1 \$42 X200 Y95 R1001 FX1000 FY500
	N120 \$40 X3
	N130 G0 Y150
	



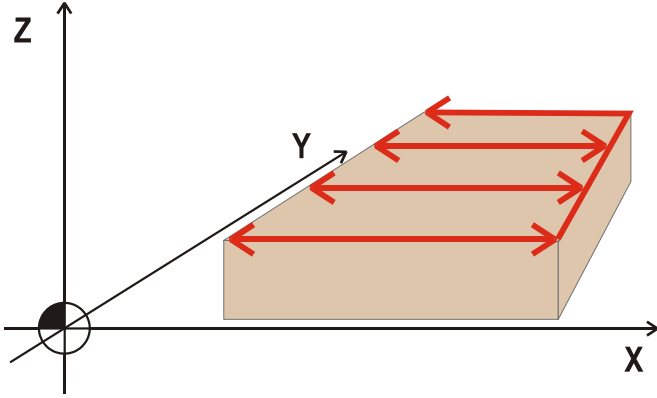
\$42	Oscillating with infeed at both reversal points and a second feed axis
Format	\$42 X Y Z R1001, R1002, R3 X, Y, Z = any axis letters R1001, R1002, R3 = arithmetic parameter addresses
Explanation	A second feed axis is added to oscillating with one feed axis.

Example	
	N100 G0 X0 Y0 Z100 R1001:= 10 R1002:= 0.5 R3:= 1
	N110 G1 \$42 X200 Y50 Z95 R1001 R1002 R3 FX1000 FY500 FZ500
	N120 \$40 X3 Y0
	N130 G0 Z150

Record N110:	G1	Straight interpolation
	\$41	Select oscillate function with continuous infeed
	X	Is the oscillating axis. The programmed position is the 1st reversal point. The 2nd reversal point results from the position of X at the start of oscillating.
	Y	is the 1st feed axis. The programmed coordinate is the reversal point. Infeed in Y is continuous.
	Z	is the 2nd feed axis. The programmed coordinate is the final dimension. Infeed in Z occurs at the reversal point of Y, dependant on R2.
	R1001	Contains the infeed increment for Z
	R1002	Contains the infeed increment for Z
	R3	controls infeed in Z: R3 =0 Infeed at front (smaller) reversal point of Y. R3 =1 Infeed at both reversal points of Y. R3 =2 Infeed at rear (greater) reversal point of Y.
	FX	is the feedrate of the oscillating axis
	FY	is the feedrate of the 1st feed axis
	FZ	is the feedrate of the 2nd feed axis

4.20 \$43 Oscillating With Infeed Only at Right Reversal Point

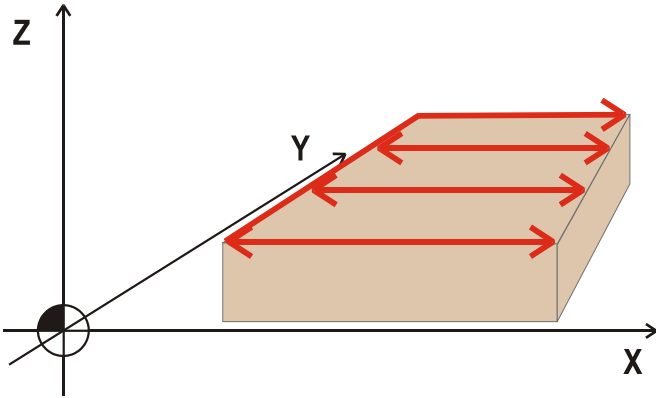
\$43	Oscillating with infeed only at right reversal point
Format	\$43 X Y R X, Y = any axis letter R = arithmetic parameter
Explanation	Function and example as \$42.
Note	As in \$42, a second feed axis can be used. A reversal dwell time can be programmed in the oscillation record with T1. The reversal dwell time starts as soon as infeed has occurred. The oscillating axis remains at the reversal point until the dwell time has expired.

Example	
----------------	---



4.21 \$44 Oscillating With Infeed Only at Left Reversal Point

\$44	Oscillating with infeed only at left reversal point
Format	\$44 X Y R X, Y = any axis letter R = arithmetic parameter
Explanation	Function and example as \$42.
Note	As in \$42, a second feed axis can be used. A reversal dwell time can be programmed in the oscillation record with T1. The reversal dwell time starts as soon as infeed has occurred. The oscillating axis remains at the reversal point until the dwell time has expired. The reversing behaviour of the oscillating axis can be controlled with G67.

Example	
---------	---

4.22 \$47 Define Machining Plane

\$47	Define machining plane
Format	\$47 U V U, V = any axis letters
Explanation	Machining planes are defined by two axes, specified in configuration parameter Q.054 as axes parallel to X, Y or Z. If several axes in a subsystem are defined parallel to the same dimensional coordinate you can select the axes for the current machining plane with \$47.
Note	The machining plane is activated with G17, G18 or G19. \$47 selects the axes that define the plane, if this is not clear from parameter Q.054.

4.23 \$48 Release Axis for Subsystem Change

\$48	Release axis
Format	\$48 X X = any axis letter
Explanation	<p>If the controller is configured for running several subsystems each axis must be assigned to one of the subsystems. Then an NC program can be started for each subsystem and these NC programs can run in parallel.</p> <p>It may be necessary to program one of the axes in several subsystems (example: multispindle lathes). In this case an axis can be "borrowed" from a subsystem by assigning any axis letter to the axis number.</p> <p>See also: Lending NC Axes Between NC Subsystems.</p> <p>If this axis is then no longer required in this subsystem it must be released in order to avoid jamming situations.</p> <p>With this function the borrowing system gives the "borrowed" axis back to the original system.</p>
Note	<p>Axes are borrowed from other systems by programming X = axis number (X = any axis letter not used for another axis).</p> <p>M functions M90 to M99 are available to ensure a degree of processing sequence.</p>

In the example the controller's third axis is assigned to subsystem #1. The NC program runs in subsystem #2, where axis letter U is no longer used.

Example	#2
	N20 U:=3 Assign 3rd axis to address U N30 G1 U100 Y20 F50 and move in interpolation context with Y. N30 Y60 \$48 U Axis U released



4.24

\$53 - \$54 Abort Motion

\$53	Abort motion with following error compensation
Format	\$53 X X = any axis letter
Explanation	Stop axis motion through interrupt signal
Note	<p>With active interrupt signal the axis motion is aborted immediately and record change is carried out.</p> <p>The current position of the axes at the record change corresponds to that at the time of the interrupt.</p> <p>The function is effective record by record.</p> <p>See also \$54.</p>
\$54	Delete remaining distance through interrupt signal
Format	\$54X I X = any axis letter, I = remaining distance data
Note	<p>The axis position of the corresponding axis at the time of receipt of the interrupt signal is saved, the associated path position is determined. The distance programmed in "I" is travelled from this path position. The resulting difference between the programmed record end position and the actual record end position is saved in an internal zero point offset. This zero point offset remains until the next G39 (including implicit G39 e.g. \$1, E1 = 1). Record change can occur with G64. That also means at maximum velocity without braking. The internal zero point offset can be taken into consideration later in a G39 record.</p> <p>I = residual distance of interrupt (measured position) to record end on path. I must be at least as great as the distance travelled in three interpolation cycles. If record change is programmed with G9 the braking distance must also be taken into consideration.</p> <p>The function is effective record by record.</p> <p>See also \$53.</p>
Example	
	N100 G1 \$54 X400 I100



\$54 is only permissible in connection with G1.

4.25 \$90, \$91 Absolute/Incremental Measurements, Axis-specific

\$90	Absolute measurements
\$91	Incremental measurements
Format	\$90 X \$91 X X = any axis letter
Explanation	\$90 Absolute measurements for this axis \$91 Incremental measurements for this axis With these \$ functions G functions G90 or G91 can be superposed for individual axes to mix absolute and incremental dimensions in one record.
Note	\$90 and \$91 are self-maintaining until program end or until the dimensions are changed by programming G90 or G91 . The two \$ functions cancel one another, programming G90 or G91 deletes all programmed \$90 or \$91 functions. The default setting is absolute.
Example	
	N110 G90 X100 Y35 F1050 All axes with absolute position
	N120 X120
	N130 X125 \$91 YR256 X-axis moves on absolute coordinate, Y-axis moves incrementally with content of R256
	N140 G91 X15 Y5 All axes move incrementally
	N150 X10 \$90 Y75 X-axis moves incrementally, Y-axis moves on absolute coordinate



5 M Functions

M functions can be used to program logic functions.

M0	Programmed stop
M1	Optional stop
M3	Spindle rotation clockwise (and special case M"axisname")
M4	Spindle rotation anticlockwise (and special case M"axisname")
M5	Spindle stop (and special case M"axisname")
M17	Subprogram end see M17 and M30 program end
M30	Program end / reset see M17 and M30 program end
M90, M91 to M98	Synchronization of NC subsystems

Up to max. 3 M functions can be programmed in each record. M functions which are not predefined can be evaluated at will in the PLC. M functions in a record can lead to record change delays, because the PLC user program processes the signal and has to give an enable for the next record. This process takes at least two interpolation cycles. Record change without loss of velocity (G64) is carried out without enable from the PLC user program.

5.1 M0

M0	Programmed stop
Format	M0
Explanation	Stop after record execution. Program can be continued with CNC start.
Example	
	N120 G0 X100 M0

5.2 M1

M1	Optional stop
Format	M1
Explanation	Stop after record execution if function activated from PLC program. Program can be continued with CNC start.
Example	
	N120 G0 X100 M1

5.3 M3 and M4

M3	Clockwise spindle rotation
Format	M3
Explanation	Starts an NC axis declared as main spindle or a PLC-controlled spindle.
Note	<p>M"axisname" 3</p> <p>If one or more axes in an NC subsystem are declared as rotary axes they can be operated as controlled spindles with M"axisname" 3. The speed is then programmed with S"axis name".</p>
M4	Anticlockwise spindle rotation
Format	M4
Explanation	Starts an NC axis declared as main spindle or a PLC-controlled spindle
Note	<p>M"axisname" 4</p> <p>If one or more axes in an NC subsystem are declared as rotary axes they can be operated as controlled spindles with M"axisname" 4. The speed is then programmed with S"axis name".</p>



5.4

M5

M5	Spindle stop
Format	M5
Explanation	Stops an NC axis declared as main spindle or a PLC-controlled spindle
Note	<p>Switching a controlled spindle from spindle mode to rotary axis mode is programmed with M5, and in the next record G39 for synchronizing up the actual position.</p> <p>Alternatively to M5 and G39 a controlled spindle can also be stopped at a target position by programming the target position after the axis letter of the spindle.</p> <p>M"axisname" 5</p> <p>If one or more axes in an NC subsystem are declared as rotary axes they can be operated as controlled spindles with M"axisname" 5.</p>

Example	
	N10 M3 S500 Spindle start
	N20 X... Y...
	N30 M5 Spindle stop
	N40 G39 Synchronize up actual position
	N50 C45 F300 Position spindle as C-axis

Example	
	N10 M3 S500 Spindle start
	N20 X... Y...
	N30 C45 Positioned spindle stop



On M3, M4 and M4

- A spindle is declared with Q.054 bit 3 = 1.
- If there is no main spindle the system-specific variable `cncMem.sysSect[n].flgN2P.bM345Act` is set and must be acknowledged by the PLC program.
The PLC program can control a spindle with the system-specific PLC variables `cncMem.sysSect[n].wrdN2P.lMFct1` to `lMFct3`.

5.5 M90 to M98 Synchronization of NC Subsystems

M90	Synchronization of all subsystems
M91 to M98	Synchronization with subsystem 1 to 8
Format	M90 M95
Explanation	<p>These functions are required for controllers with several subsystems. In this case several NC programs (one per subsystem) can run at the same time.</p> <p>It is often necessary to divide these NC programs into parts that can be executed in parallel and parts which must be executed sequentially.</p> <p>Functions M90 to M98 are synchronization labels for controlling execution of the NC programs. Execution of the NC program stops at a synchronization label until the NC program of the corresponding subsystem has also reached a label. Then execution continues in all involved subsystems.</p>

The following example shows synchronization of two subsystems with functions M91 and M92. If the controller is configured for exactly two subsystems all labels can be replaced with M90.

Example	Subsystem 1	Subsystem 2
	N10..... (#1 and #2 parallel)	
	N20.....	N10.....
	N30 M92 (sync with #2)	N20 M91 (sync with #1)
	N40.....	
	N50..... (only #1 active)	(#2 waits at N30)
	N60.....	
	N70 M92 (sync with #2)	N30 M91 (sync with #1)
	N80..... (parallel again)	
	N90.....	N40.....
		N50.....



Inappropriate use of synchronization labels can lead to deadlock situations (jamming) in NC program processing.



6 CNC-PLC Interface

E	Request a bit variable
SE	Set a bit variable at the start of record execution
RS	Reset a bit variable at the start of record execution
WA	Wait for bit variable = 1
WN	Wait for bit variable = 0

6.1 E

E	Request a bit variable
Format	Ennn = 1 Ennn = 0 nnn = number of bit variable, 3-digit decimal number in range 0 - 255)
Note	PLC access path to bit variable is: <i>cncMem.comSect.abFlgPNRw[n]</i> (n = number of bit variable 0-255) Bit variables are executed at the time of record change from the preceding NC record. The controller executes an automatic G39.
Example	N10 E0=1 B%9000 (If bit variable 0 = 1 jump to subroutine %9000.)

6.2 SE

SE	Set a bit variable at the start of record execution
Format	SEnnn nnn = number of bit variable, 3-digit decimal number in range 0 - 255)
Note	PLC access path to bit variable is: <i>cncMem.comSect.abFlgPNRw[n]</i> (n = number of bit variable 0-255)
Example	N10 SE0

6.3 RS

RS	Reset a bit variable at the start of record execution
Format	RSnnn nnn = number of bit variable, 3-digit decimal number in range 0 - 255)
Note	PLC access path to bit variable is: <i>cncMem.comSect.abFlgPNRw[n]</i> (n = number of bit variable 0-255)
Example	N10 RS0

6.4 WA and WN

WA	Wait for bit variable = 1
Format	WAnnn nnn = number of bit variable, 3-digit decimal number in range 0 - 255)
Explanation	Record change to next record only if bit signal = 1. Bit variable checked at end of any axis motion.
Note	PLC access path to bit variable is: <i>cncMem.comSect.abFlgPNRw[n]</i> (n = number of bit variable 0-255) Bit variable checked at end of any axis motion.
WN	Wait for bit variable = 0
Format	WNnnn nnn = number of bit variable, 3-digit decimal number in range 0 - 255)
Explanation	Record change to next record only if bit signal = 0.
Note	PLC access path to bit variable is: <i>cncMem.comSect.abFlgPNRw[n]</i> (n = number of bit variable 0-255) Bit variable checked at end of any axis motion.
Example	N10 G0 X100 WN0 Motion executed regardless of instruction WN0. WN instruction evaluation and possible record change not until position X = 100.



7

Arithmetic Parameters for ProNumeric

General Arithmetic Parameters R2000 to R5999 (Integer Values)

General Arithmetic Parameters R6000 to R9999 (Real Values)

System-specific Arithmetic Parameters R000 to R999 (Integer Values)

System-specific Arithmetic Parameters R1000 to R1999 (Real Values)

Zero Point Offsets R10001 to R10564

Zero Overlays R10601 to R10664

R10701 to R10764 Reference Point Offset

All parameters are in the shared RAM and can be read and written by the CNC and PLC.

Function assignment is defined by the NC program.

Real values can be programmed and entered in decimal form with up to 7 positions plus sign.

Integer values are positive or negative whole numbers.

The active system of units (G70 and G71) is taken into consideration when substituting coordinate values and velocities.

7.1 General Arithmetic Parameters R2000 to R5999 (Integer Values)

Number	Type
R2000 to R5999	Global arithmetic parameters, identical in all CNC subsystems

7.2 General Arithmetic Parameters R6000 to R9999 (Real Values)

Number	Type
R6000 to R9999	Global arithmetic parameters, identical in all CNC subsystems

7.3 System-specific Arithmetic Parameters R000 to R999 (Integer Values)

Number	Type
R000 to R999	Local arithmetic parameters, which exist once per CNC subsystem

7.4 System-specific Arithmetic Parameters R1000 to R1999 (Real Values)

Number	Type
R1000 to R1999	Local arithmetic parameters, which exist once per CNC subsystem

7.5 Zero Point Offsets R10001 to R10564

6 zero point offsets are available.

The zero point offsets are called with G54 to G59.

Each axis is assigned to a parameter number.

Parameter	
R10001	1st axis zero point offset 1 (G54)
to	
R10064	64th axis zero point offset 1 (G54)
R10101	1st axis zero point offset 2 (G55)
to	
R10164	64th axis zero point offset 2 (G55)
R10201	1st axis zero point offset 3 (G56)
to	
R10264	64th axis zero point offset 3 (G56)
R10301	1st axis zero point offset 4 (G57)
to	
R10364	64th axis zero point offset 4 (G57)
R10401	1st axis zero point offset 5 (G58)
to	
R10464	64th axis zero point offset 5 (G58)
R10501	1st axis zero point offset 6 (G59)
to	
R10564	64th axis zero point offset 6 (G59)



7.6 Zero Overlays R10601 to R10664

In these R parameters you can set a permanent zero overlay independent of the program.

A parameter is assigned each axis.

Parameter	
R10601	1st axis zero overlay
to	
R10664	64th axis zero overlay

The monitor display is altered according to this data. The internal controller actual value and software limit switch functions are unaffected.

If value = 0 no zero overlay occurs.



The content of this parameter is effective as zero overlay after homing.

For axes with absolute value encoder:

A value entered in these parameters must not be less than the reference point coordinate (Q.034).

7.7 R10701 to R10764 Reference Point Offset

The difference between actual value and reference point offset (G92) is entered in these parameters. This means they can be read by the CNC.

A parameter is assigned each axis.

Parameter	
R10701	1st axis zero point offset (G92)
to	
R10764	64th axis zero point offset (G92)

8 Arithmetic Parameters for XCx (in Preparation)

 General Arithmetic Parameters R2000 to R5999

 General Arithmetic Parameters R6000 to R9999

 System-specific Arithmetic Parameters R000 to R999

 System-specific Arithmetic Parameters R1000 to R1999

 Zero Point Offsets R10001 to R10564

 Zero Overlays R10601 to R10664

 R10701 to R10764 Reference Point Offset

All parameters are in the shared RAM and can be read and written by the CNC and PLC.

Function assignment is defined by the NC program.

Real values can be programmed and entered in decimal form with up to 7 positions plus sign.

Integer values are positive or negative whole numbers.

The active system of units (G70 and G71) is taken into consideration when substituting coordinate values and velocities.

8.1 General Arithmetic Parameters R2000 to R5999 (Integer Values)

Number	Type
R2000 to R5999	Global arithmetic parameters, identical in all CNC subsystems

8.2 General Arithmetic Parameters R6000 to R9999 (Real Values)

Number	Type
R6000 to R9999	Global arithmetic parameters, identical in all CNC subsystems

8.3 System-specific Arithmetic Parameters R000 to R999 (Integer Values)

Number	Type
R000 to R999	Local arithmetic parameters, which exist once per CNC subsystem



8.4 System-specific Arithmetic Parameters R1000 to R1999 (Real Values)

Number	Type
R1000 to R1999	Local arithmetic parameters, which exist once per CNC subsystem

8.5 Zero Point Offsets R10001 to R10564

6 zero point offsets are available.

The zero point offsets are called with G54 to G59.

Each axis is assigned to a parameter number.

Parameter	
R10001	1st axis zero point offset 1 (G54)
to	
R10064	64th axis zero point offset 1 (G54)
R10101	1st axis zero point offset 2 (G55)
to	
R10164	64th axis zero point offset 2 (G55)
R10201	1st axis zero point offset 3 (G56)
to	
R10264	64th axis zero point offset 3 (G56)
R10301	1st axis zero point offset 4 (G57)
to	
R10364	64th axis zero point offset 4 (G57)
R10401	1st axis zero point offset 5 (G58)
to	
R10464	64th axis zero point offset 5 (G58)
R10501	1st axis zero point offset 6 (G59)
to	
R10564	64th axis zero point offset 6 (G59)

8.6 Zero Overlays R10601 to R10664

In these R parameters you can set a permanent zero overlay independent of the program.

A parameter is assigned each axis.

Parameter	
R10601	1st axis zero overlay
to	
R10664	64th axis zero overlay

The monitor display is altered according to this data. The internal controller actual value and software limit switch functions are unaffected.

If value = 0 no zero overlay occurs.



The content of this parameter is effective as zero overlay after homing.

For axes with absolute value encoder:

A value entered in these parameters must not be less than the reference point coordinate (Q.034).

8.7 R10701 to R10764 Reference Point Offset

The difference between actual value and reference point offset (G92) is entered in these parameters. This means they can be read by the CNC.

A parameter is assigned each axis.

Parameter	
R10701	1st axis zero point offset (G92)
to	
R10764	64th axis zero point offset (G92)



9 Overview Tables

9.1 Overview of G Words

In this overview the G words are organized in groups.

Only one function from each group can be active.

Normally the functions remain active until they are deselected by another function from the same group.

In individual cases a function is active for only one record (property = S). Some functions are default settings (property = D).

Group	Properties D = Default setting S = Active for 1 record		Meaning
1		G0	Contour control in rapid feed.
	D	G1	Straight interpolation
		G2	Clockwise circle-helix interpolation
		G3	Anticlockwise circle-helix interpolation
		G10	Point-to-point positioning in rapid feed
		G11	Home to reference point
		G12	Clockwise spiral interpolation
		G13	Anticlockwise spiral interpolation
		G25	Online curve interpolation OCI without tangential transition
		G26	Online curve interpolation OCI with tangential transition
		G27	Free form interpolation of CNC programs created offline
		G32	Interpolated tapping
		G33	Thread cutting
		G63	Tapping without compensating chuck
		G76	Thread cycle
		G77	Tapping cycle without compensating chuck
2	S	G4	Dwell time
3	D	G17	Plane selection X-Y
		G18	Plane selection X-Z
		G19	Plane selection Y-Z
4	S	G39	Interrupt record preparation
5	D	G40	Switch off tool radius compensation
		G41 G42	Tool radius compensation left/right
		G43 G44	Tool radius compensation positive/negative
	S	G50	Tool radius compensation without transition contour
6		G45 G46	Feedrate correction
7	D	G53 to G59	Zero point offset

Group	Properties D = Default setting S = Active for 1 record		Meaning
8	S	G9	Exact positioning
	D	G60	Record change after exact stop boundary reached
		G61	Record change after elimination of set-actual deviation
		G64	Record change without loss of velocity
9	S	G67	Special function for oscillating
10		G70	Units in inches; the last used function applies
		G71	Units in millimetres
11	D	G80 to G89	Machining cycles
12	D	G90	Absolute measurements
		G91	Incremental measurements
13		G92	Reference point offset
14	D	G94	Feedrate in mm/min (in/min)
		G95	Feedrate in mm/rev. (in/rev.)
15		G96	Constant cutting speed
	D	G97	Spindle speed given in r.p.m.
16	D	G98	Accept self-holding preparatory functions
		G99	Do not accept self-holding preparatory functions



9.2 Overview of \$ Words

In this overview the \$ words are organized in groups.

Only one function from each group can be active.

Normally the functions remain active until they are deselected by another function from the same group.

In individual cases a function is active for only one record (property = S). Some functions are default settings (property = D).

Group	Properties D = Default setting S = Active for 1 record		Meaning
1	S	\$1	Stop axis motion
	S	\$53 - \$54	Abort motion
2		\$23	Switch on internal follow-up operation
		\$24	Switch on follow-up operation
		\$25	Switch off follow-up operation
3		\$26	Switch on independent axis with individual feedrate
		\$27	Switch off independent axis with individual feedrate
		\$28	Reintegrate independent axis in record change
		\$29	Do not integrate independent axis
4		\$31	Switch on synchronous operation
		\$32	Switch off synchronous operation
5		\$33	Lead axis for thread cutting
6		\$34	Radius axis for v = constant
7		\$38	Switch on contouring axis in IPO context
		\$39	Switch off contouring axis in IPO context
8		\$40	Switch oscillation off
		\$41	Switch on oscillation with continuous infeed
		\$42	Switch on oscillation with infeed on both sides
		\$43	Switch on oscillation with infeed right
		\$44	Switch on oscillation with infeed left
9		\$47	Define machining plane
10		\$48	Give back system axis
11		\$90	Absolute measurements
		\$91	Incremental measurements

9.3 M functions

M0	Programmed stop
M1	Optional stop
M3	Clockwise spindle rotation
M4	Anticlockwise spindle rotation
M5	Spindle stop
M17	Subroutine end
M30	Program end/reset
M90, M91 to M98	Synchronization of NC subsystems

9.4 CNC-PLC Interface

E	Request a bit variable
SE	Set a bit variable at the start of record execution
RS	Reset a bit variable at the start of record execution
WA	Wait for bit variable = 1
WN	Wait for bit variable = 0

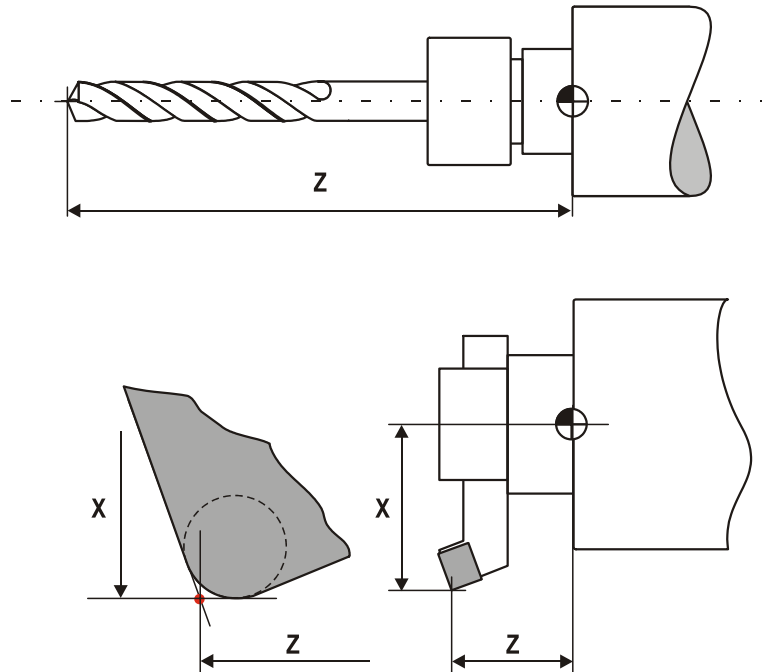
10 Appendix

10.1 Tool Compensations

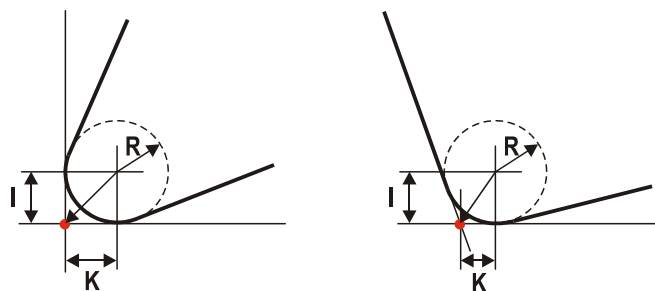
10.1.1 Measuring tools

Determining tool length

The tool length is the distance between the tool reference point and the theoretical cutting point



Determining tool length compensation values I and K

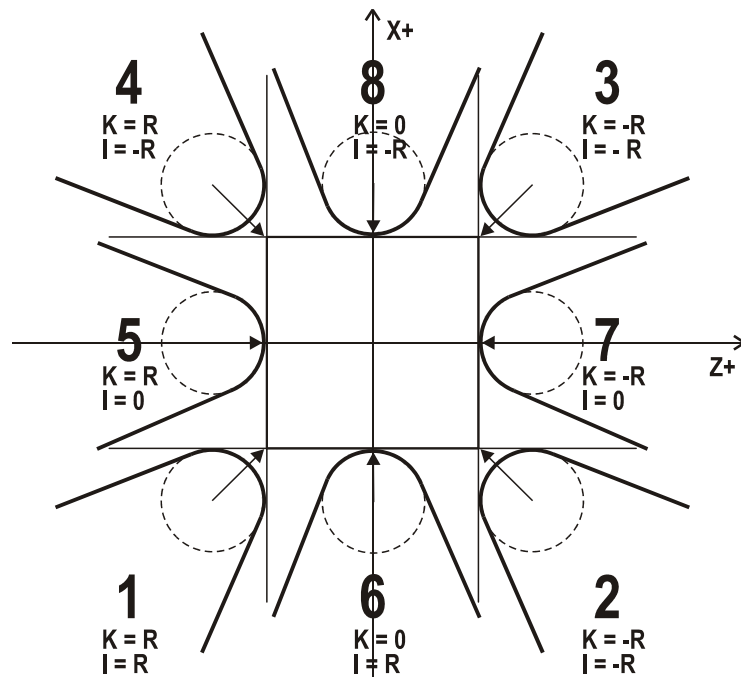


R = tool nose radius
 I , K = compensation values

10.1.2 Quadrant assignment for tool nose radius compensation

Quadrant assignment

for tool nose radius compensation behind and in front of turning centre.
Using example of XZ plane (XZ plane selection with G18).



R = tool nose radius
I, K = compensation values



The quadrant number must be entered in the Tool Data Memory if tool nose radius compensation is to be activated.

For cutter radius compensation quadrant number 0 or 9 must be entered.



10.2 Tool Data Memory

The 99 tools data memories are shown on arithmetic parameters and begin in each case from:

R20000	1. tool data memory (Selection with T01)
R20100	2. tool data memory (Selection with T02)
..	..
R29800	99. tool data memory (Selection with T99)

The tool data memories are system-specific.

Structure of tool data memory			
R parameters	Format	Designation	Explanation
R2xx00	0000000000	IZ	Actual time - wear monitoring in min
R2xx01	0.000	X	Tool length in X-direction in mm
R2xx02	0.000	Y	Tool length in Y-direction in mm
R2xx03	0.000	Z	Tool length in Z-direction in mm
R2xx04	0.000	I	Tool length compensation value for X-direction in mm
R2xx05	0.000	J	Tool length compensation value for Y-direction in mm
R2xx06	0.000	K	Tool length compensation value for Z-direction in mm
R2xx07	0.000	R	Tool radius in mm
R2xx08	0.000	Q	Quadrant
R2xx09	0000000000	SZ	Service life in min
R2xx10	0000000000	VS	Tool wear, if value = 1
R2xx11	0000000000	IH	Tool call frequency, actual number
R2xx12	0000000000	SH	Tool call frequency, set number
R2xx13 to R2xx14	0000000000	---	reserved
R2xx15 to R2xx19	0.000	---	reserved
R2xx20 to R2xx24	0.000	User data 01 to User data 05	User data
R2xx25 to R2xx29	0000000000	User data 06 to User data 10	User data

xx = With the T function (nn-1) selected tool number

10.2.1 Tool monitoring

Tool monitoring of the CNC includes tool wear and tool call frequency monitoring.

Tool service life monitoring registers the effective operating time of the tool (not with G0, G4 and Tl) and compares this with the given limit value.

The actual time (in minutes) is written in IZ (R2xx00), the limit value (service life) in SZ (R2xx09).

The tool service life monitoring takes place only if the service life in SZ (R2xx09) is greater than zero.

The tool call frequency monitoring registers the call frequency of the tool and is incremented by the call of the T function. The actual call frequency is registered in IH (R2xx11), the maximum call frequency in SH (R2xx12).

The tool call frequency monitoring takes place only if the maximum call frequency in SH (R2xx12) is greater than zero.

The error message (0x02100008) 'Tool worn (system n)' occurs if one of the three conditions is fulfilled:

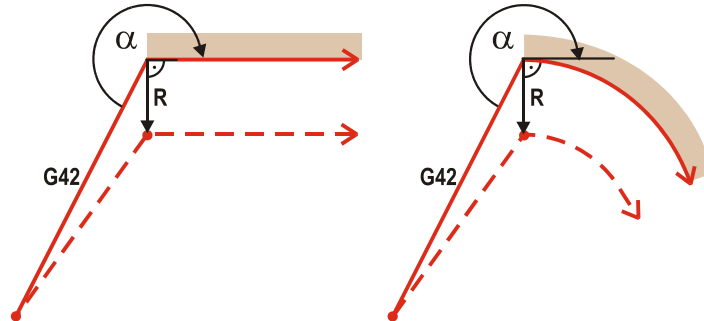
- The actual time is equal or greater than the service life
- The actual call frequency is equal or greater than the max. call frequency
- With a PLC signal (shared RAM variable *cncMem.sysSect[n].flgP2N.bToolWornExt* is TRUE)

In addition a One is written into VS (R2xx10) and the shared RAM variable *cncMem.sysSect[n].flgN2P.bToolWorn* is set to TRUE.

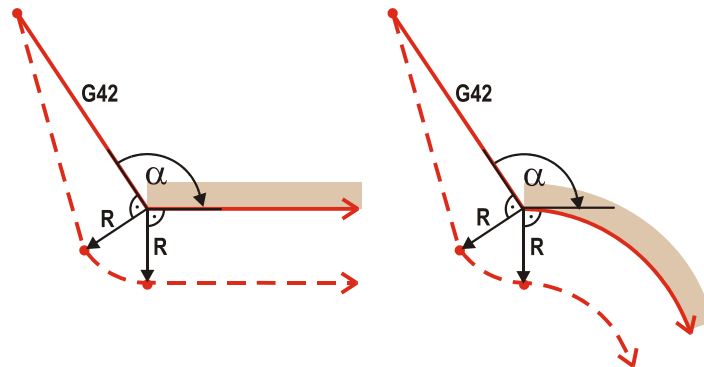
10.3 Approach and Departure Strategies

Approach
at various angles

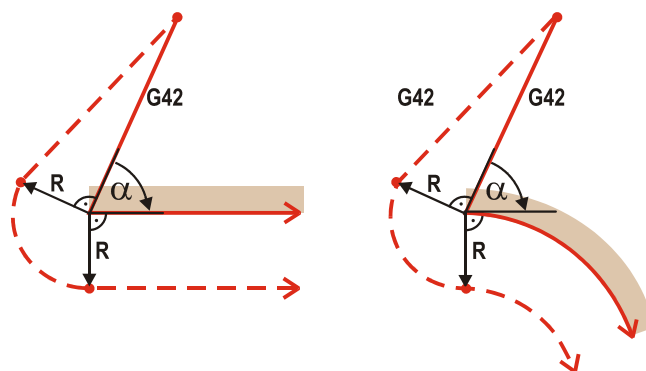
$\alpha > 180^\circ$

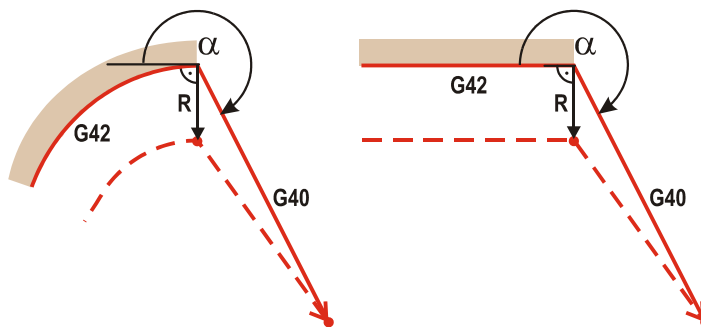
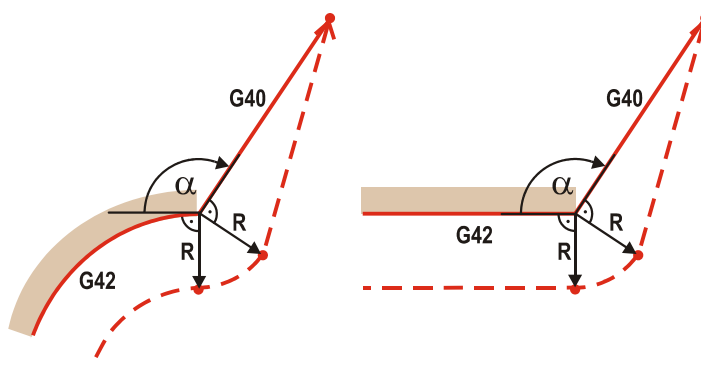
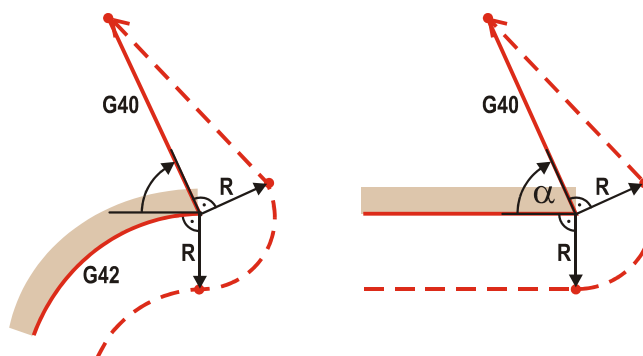


$90^\circ \leq \alpha \leq 180^\circ$



$\alpha < 90^\circ$



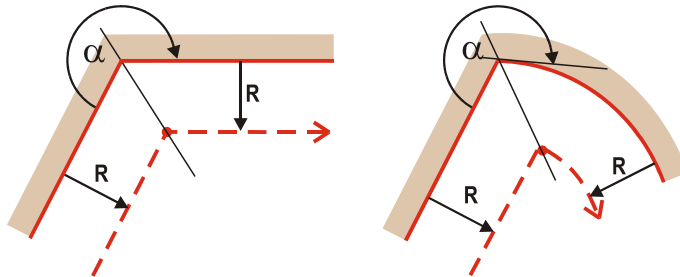
Departure
at various angles $\alpha > 180^\circ$  $90^\circ \leq \alpha \leq 180^\circ$  $\alpha < 90^\circ$ 

10.4 Contour Transitions

Straight - straight

Straight - arc

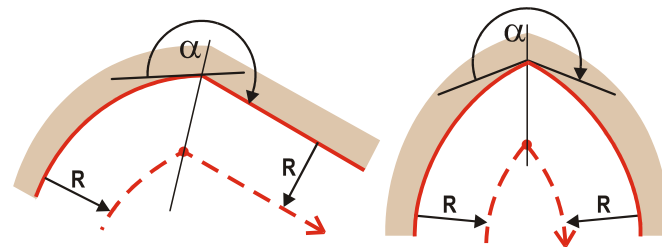
$\alpha > 180^\circ$



Arc - straight

Arc - arc

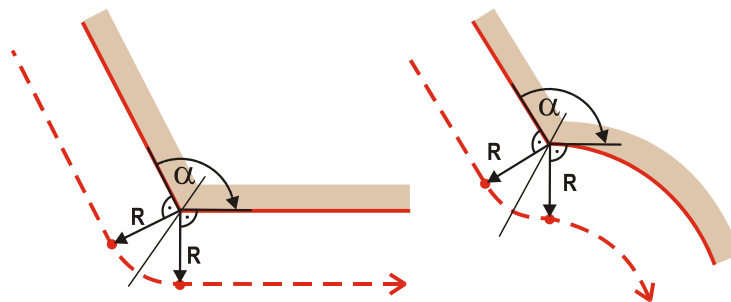
$\alpha > 180^\circ$



Straight - straight

Straight - arc

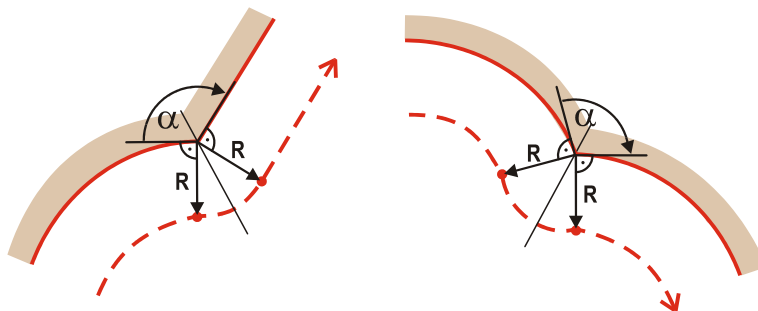
$90^\circ \leq \alpha \leq 180^\circ$



Arc - straight

Arc - arc

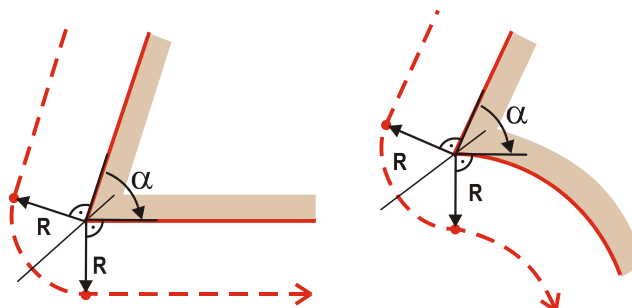
$$90^\circ \leq \alpha \leq 180^\circ$$



Straight - straight

Straight - arc

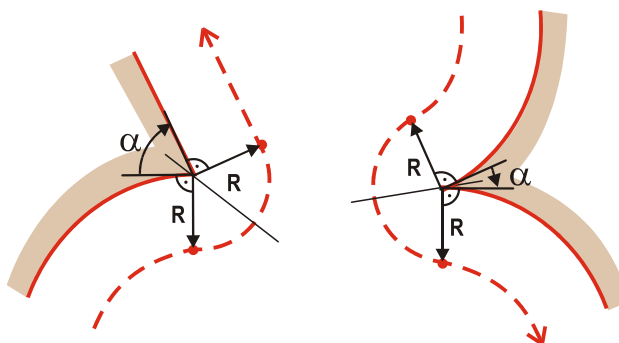
$$\alpha < 90^\circ$$



Arc - straight

Arc - arc

$$\alpha < 90^\circ$$

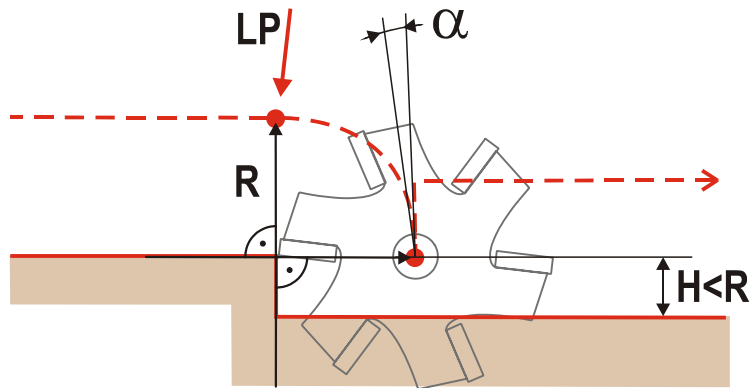


Contour transitions with error messages and STOP

Stop is activated as soon as the contour transition has been interpreted. Interpretation is predictive so the position where STOP occurs may be far before the contour transition. The last position (LP) can be approached by repeating START.

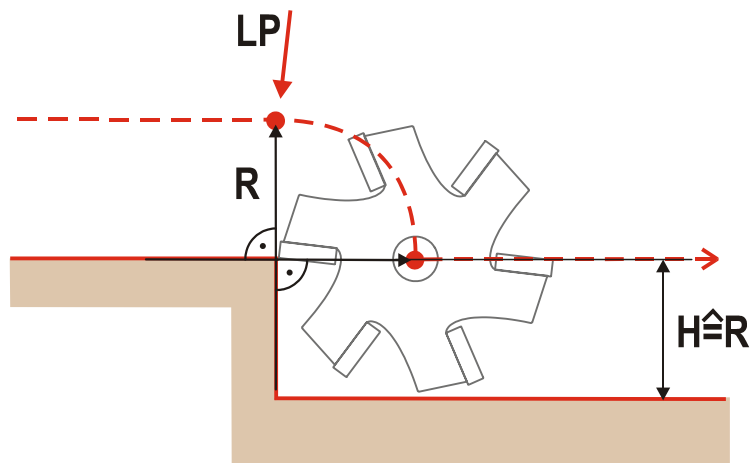
Error 0x21300003

"Angle too sharp"

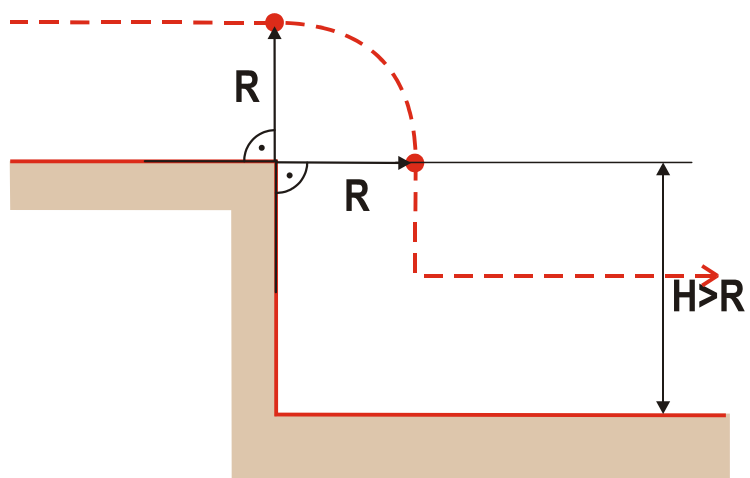


Error 0x21300005

"Start position = end position"



No error message, contour and tool radius are OK

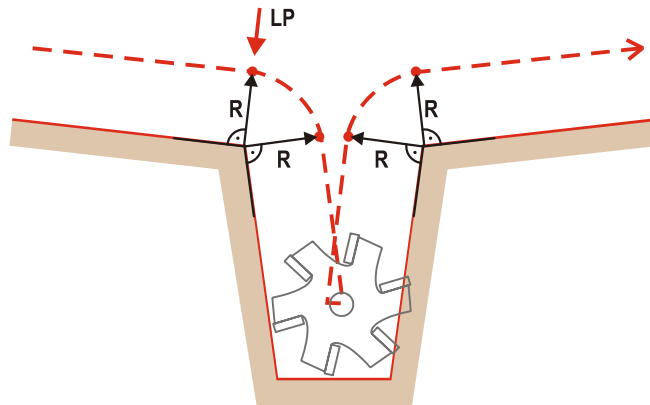


LP = Last position that can be approached

H = Height

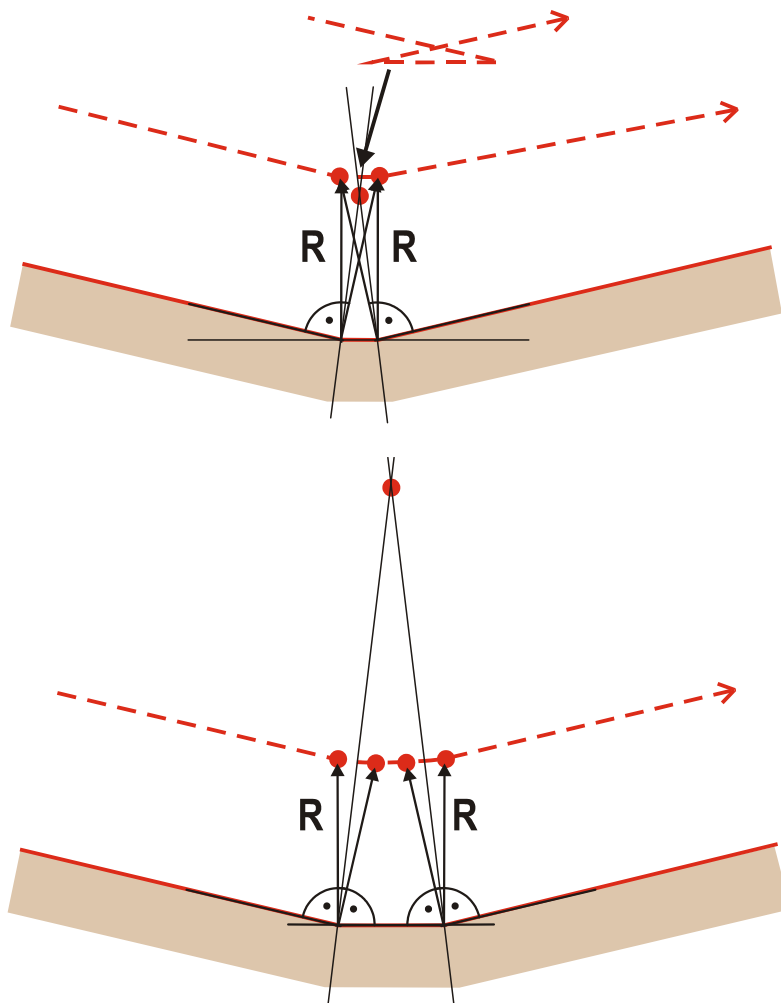
R = Tool radius

Error 0x21300004
"Tool radius too large"



LP = Last position that can be approached
R = Tool radius

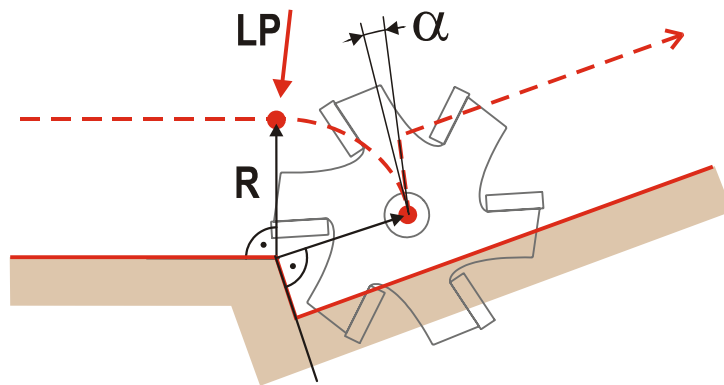
Error 0x21300004
"Tool radius too large"



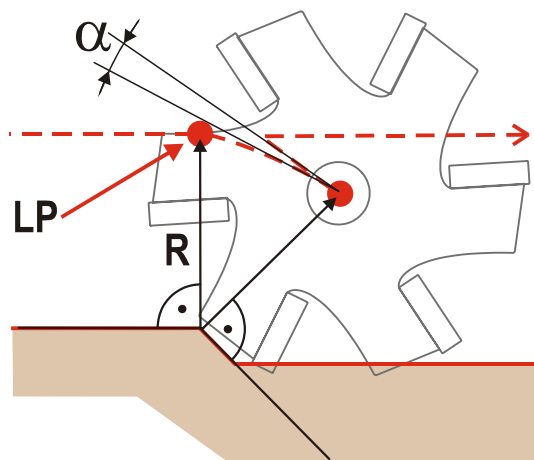
No error message, contour and
tool radius are OK

R = Tool radius

Error 0x21300003
"Angle too sharp"



Error 0x21300003
"Angle too sharp"



LP = Last position that can be approached
R = Tool radius

10.5 Lending NC Axes Between NC Subsystems

In order to interpolate axes of one NC subsystem with axes of another you can assign an axis number to an axis letter (A, B, C, D, L, O, P, U, V, W, X, Y, Z).

Example:

Axis X is the controller's 3rd axis and normally belongs to NC subsystem 1. If this axis is to move in NC subsystem 2 you have to assign a number to the axis. (See also M90)

NC subsystem 1	
N10 G0 X1000	
N20 M90	(Wait for NC subsystem 2)
N30 (X must not be used)	(Axis X currently moving in NC subsystem 2)
N40 M90	(Wait for NC subsystem 2)
N50 G0 X0 M17	
NC subsystem 2	
N10 M90	(Wait for NC subsystem 1)
N20 X:=3	(Controller's 3rd axis from NC subsystem 1 moving as X-axis in subsystem 2)
N30 G1 X500 F2000	
N40 \$48 X	(X released again with \$48)
N40 M90	(Wait for NC subsystem 1)
N50 M17	



11 Index

\$

\$ functions	73
\$1 stop axis motion	74
\$20 handwheel enable for velocity superposition	75
\$21 handwheel enable for path superposition	75
\$23 internal follow-up operation on	75
\$24 follow-up operation on	76
\$25 switch off follow-up operation	76
\$26 exclude axes from interpolation context	77
\$27 include axes in interpolation context	78
\$28 include independent axis in record change	78
\$29 do not include independent axis in record change	79
\$31 switch on synchronous operation	80
\$32 switch off synchronous operation	81
\$33 selecting a lead axis	82
\$34 axis selection for constant cutting speed	82
\$37 Path length calculation	82
\$38 and \$39 axis selection for path feedrate calculation	83
\$40 switch oscillation off	84
\$40 to \$44	63
\$41 oscillation with continuous infeed	85
\$42 oscillating with infeed at both reversal points	87
\$43 oscillating infeed right	89
\$44 oscillating infeed left	90
\$47 define machining plane	90
\$48 subsystem change axes	91
\$53 \$54 measuring sensor	92
\$90, \$91 absolute/incremental measurements	93

%

% character	11
-------------------	----

A

Absolute measurements	68
Absolute value	18
Acceleration	22
Acceleration monitoring	58
Acceleration monitoring see G62	60
Acceleration override	22
Addition	18
Additional preparatory functions \$ functions	73
Address identifier \$	7
Approach strategy tool nose radius compensation	114
Arc tangent	18
Arithmetic parameters	
system-specific for ProNumeric	100
system-specific for XCx	103
Arithmetic parameters	
arithmetic zero point offsets for ProNumeric	101
arithmetic zero point offsets for XCx	104
general for ProNumeric	100
general for XCx	103
Reference point offsets for ProNumeric	102
Reference point offsets for XCx	105
zero overlays for ProNumeric	102
zero overlays for XCx	105
Axis selection for constant cutting speed	82
Axis specific rounding see RF	55
Axis-specific feedrate	77

C

Calculations	18
Calculations in the NC record	18
absolute value	18
addition	18
arc tangent	18
cosine	18
division	18
multiplication	18
negated assignment	18
sine	18
square root	18
subtraction	18
tangent	18
C-axis	96
Circle centre coordinates with G2/G3	28
Circular interpolation G2/G3	28
CNC-PLC interface overview	109
Comment	9
Comparisons	15
Compensating chuck	61
Compensation values for tool length	110
Conditional program executions	15
Conical thread	43
Constants	19
Contour control with rapid feed velocity G0	26
Controlled spindle	95
Coordinate calculation	19
Cosine	18
Cutting point	110
Cylindrical thread	43

D

Define machining plane \$47	90
Delete remaining distance with interrupt	92
DIN 66025	7
Division	18
Dwell time G4	32
Dwell time T1	32

E

E . . . = -instruction	16
E request a bit variable	98
Empty buffer G39	44
Exact positioning	
self-maintaining G60	33
Exact positioning G9	33

F

F word	70
Feedrate in mm per revolution	70
Follow-up operation	
internal switch on \$23	75
switch off	76
switch on \$24	76
Freeform interpolation G27	40

G

G functions.....	24
G words	
Overview.....	106
G0 contour control with rapid feed velocity.....	26
G1 contour control with linear interpolation.....	27
G10 point-to-point positioning in rapid feed.....	34
G11 home to reference point.....	35
G12 clockwise spiral interpolation.....	36
G13 anticlockwise spiral interpolation.....	36
G17 working plane X/Y.....	38
G18 working plane X/Z.....	38
G19 working plane Y/Z.....	38
G2 clockwise circular and helix interpolation.....	28
G25 online curve interpolation OCI.....	39
G26 online curve interpolation OCI.....	39
G27 freeform interpolation.....	40
G3 anticlockwise circular and helix interpolation.....	28
G32 thread.....	41
G33 thread cutting.....	42
G39 empty buffer.....	44
G4 dwell time.....	32
G40 switch off tool radius compensation.....	44
G41 tool radius compensation right.....	47
G42 tool radius compensation left.....	47
G43 tool radius compensation positive.....	50
G44 tool radius compensation negative.....	50
G45 switch off path feed rate compensation.....	52
G46 switch on path feed rate compensation.....	52
G50 tool radius compensation without transition contour.....	51
G53 to G59 zero point offset.....	56
G60 exact positioning self-maintaining.....	33
G61 smoothing.....	58
G62 Record change with acceleration monitoring.....	60
G63 tapping.....	61
G64 smoothing without loss of velocity.....	58
G66 Synchronization of IPO support points.....	62
G67 special function for oscillating.....	63
G70 dimensions in inches.....	63
G71 dimensions in mm.....	63
G76 thread cycle.....	64
G77 thread cutting cycle.....	66
G80 to G89 machining cycles.....	67
G9 exact positioning.....	33
G90 absolute measurements.....	68
G91 incremental measurements.....	68
G92 reference point offset.....	69
G94.....	21
G94 F in mm/min.....	70
G95.....	21
G95 F in mm/rev.....	70
G96 cutting speed.....	71
G97 spindle speed.....	71
G98 G functions by subroutine.....	72
G99 G functions by subroutine.....	72
General arithmetic parameters for ProNumeric.....	100
General arithmetic parameters for XCx.....	103

H

Handwheel enable for path superposition.....	75
Handwheel enable for velocity superposition.....	75
Helix interpolation G2/G3.....	28
Helix interpolation with G12/G13.....	36
Home position search velocities with G11.....	35
Home to reference point G11.....	35

I

Incremental dimension	
\$91 axes.....	93
Incremental dimensions	
G91 subsystem.....	68
Incremental measurements G91.....	68
Independent axes	
interpolate again.....	78
not in record change.....	79
with individual feedrate.....	77
Independent axis	
include in record change.....	78
Indexed programming.....	17
Indirect programming.....	17
Influence of RD on velocity.....	53
Initialization program.....	12
Intermediate records G50.....	51
Interrupt input.....	92
IPO support points synchronization see G66.....	62

L

Lead axis.....	82
Lending NC axes between NC subsystems.....	121
Lent axis.....	91
Linear interpolation G1.....	27
Logic functions M functions.....	94

M

M functions.....	94
M functions overview.....	109
M0 programmed stop.....	94
M1 optional stop.....	94
M17 subroutine end.....	12
M3 spindle direction of rotation positive.....	95
M30 program end.....	12
M4 spindle direction of rotation negative.....	95
M5 spindle stop.....	96
M90 – M98 synchronizing subsystems.....	97
Machining cycles G80 to G89.....	67
MANUAL.....	20
MANUAL mode	
feedrate.....	20
Measurements absolute/incremental.....	68
Measuring sensor.....	92
Measuring tools.....	110
Metric/inch switching.....	63
Multiplication.....	18

N

NC record structure.....	8
Negated assignment.....	18

O

OCI.....	39
Online curve interpolation.....	39
Optional stop with M1.....	94
Oscillating	
infeed left \$44.....	90
infeed right \$43.....	89
special functions G67.....	63
with infeed at both reversal points \$42.....	87
Oscillation	



with continuous infeed \$41	85
Overview	
\$ words	108
CNC-PLC interface	109
G words	106
M functions	109
Overview of \$ words	108

P

Path feedrate	
evaluation	70
Path feedrate calculation	83
Path feedrate compensation	
switch off G45	52
switch on G46	52
Path length calculation \$37	82
Plane selection G17, G18 and G19	38
Pointer indexed programming	17
Point-to-point positioning in rapid feed G10	34
Positioned spindle stop	96
Program end	12
Program end M30	12
Program name	11
Program number	11
Program structure	11
Programmed stop with M0	94
Programming feedrate	20
Programming path feedrate	20

Q

Quadrant	112
Quadrant assignment	111

R

Radius G96	82
Radius programming RC	30
Rapid feed with G10	34
RC radius programming	30
RD transition parabola	53
Record change for independent axes	79
Record change with acceleration monitoring see G62	60
Record number	9
Record structure NC	8
Reference point offset	
Arithmetic parameters for ProNumeric	102
Arithmetic parameters for XCx	105
Reference point offset G92	69
Requesting PLC signals	16
Reversing behaviour	63
RF rounding	55
Rounding	
RF55	
RS reset a bit variable	99

S

SE set a bit variable	98
Second tool function	46
Selecting a lead axis \$33	82
Selecting the working plane G17, G18 and G19	38
Set actual value G92	69
Set bit variable	98
Sine	18
Skipping parts of records	16

Smoothing	
G61	58
G64	58
Smoothing corners	53
Speed spindle	23
Spindle rotation	95
Spindle speed	23
Spindle stop with M5	96
Spiral interpolation G12/G13	36
Square root	18
SRK	51
Stop axis motion with \$1	74
Structure of NC record	8
Subroutine call	14
Subroutine end M17	12
Subsequent record, programming	10
Subsystem assignment of axes	91
Subsystem change of axes	91
Subsystems	97
Subtraction	18
Surface grinding cycle \$41	85
Switch oscillation off \$40	84
Switching metric/inch	63
Synchronization of IPO support points see G66	62
Synchronization of NC subsystems	97
Synchronous operation	
switch off \$32	81
switch on \$31	80
System-specific arithmetic parameters for ProNumeric	100
System-specific arithmetic parameters for XCx	103

T

T function	46
T word	46
Tangent	18
Thread	
conical	43
cycle	
G76	64
G77	66
cylindrical	43
lead axis	82
pitch	42
tapping with controlled spindle	41
tapping without compensating chuck G63	61
thread cutting G33	42
TI dwell time	32
Tool call frequency	112
Tool data memory	112
quadrant	112
tool call frequency	112
tool length	112
tool length compensation	112
tool quadrant	112
tool radius	112
tool service life	112
tool wear	112
Tool length	110, 112
Tool length compensation	112
Tool memory	112
Tool monitoring	113
Tool nose radius compensation	47
Tool nose radius compensation quadrant assignment	111
Tool quadrant	112
Tool radius	112
Tool radius compensation	47
switch off G40	44



Tool radius compensation G43/G44	50
Tool reference point	110
Tool selection	46
Tool service life	112
Tool wear	112
Transition parabola RD	53

U

Unconditional program branches	13
--------------------------------------	----

V

V-constant radius	82
-------------------------	----

W

WA wait for bit variable = 1	99
Warning signs	6
WN wait for bit variable = 0	99
Working plane G17, G18 and G19	38

Z

Zero overlays	
arithmetic parameters for ProNumeric	102
arithmetic parameters for XCx	105
Zero point offset	
arithmetic parameters for ProNumeric	101
arithmetic parameters for XCx	104
G53 to G59	56