NCT2xx
NCT3xx
Machine Tool Controls

Parameters

From the software version n.14.9

Produced and developed by **NCT Automation Ltd**.
H-1148 Budapest, Fogarasi út 7.
⊠ P. O. Box: 1631 Bp. Pf. 26
☎ Phone: (+36 1) 467 63 00
☎ Fax:(+36 1) 467 63 09
E-mail: nct@nct.hu
Home page: www.nct.hu

Table of Contents

November 26, 2019

1 Definitions

1.1 Channel (L=1...8)

Each channel, independently from the other channels executes a separate part program. As a result of this, certain display screens, memory areas, PLC flags and parameters belong individually to each channel. The block executor and interpolator works separately in each channel. It is advisable to classify certain parameters per channel, i.e., to determine them separately, for instance, which machine group shall the given channel belong to.
Address “L” carries out the classification of parameters repeated per channel.

1.2 Machine group (T=1...8)

Each channel has to be assigned for a given machine group. Address “T” carries out the classification of parameters repeated per machine group.

1.3 Axis (A=1...32)

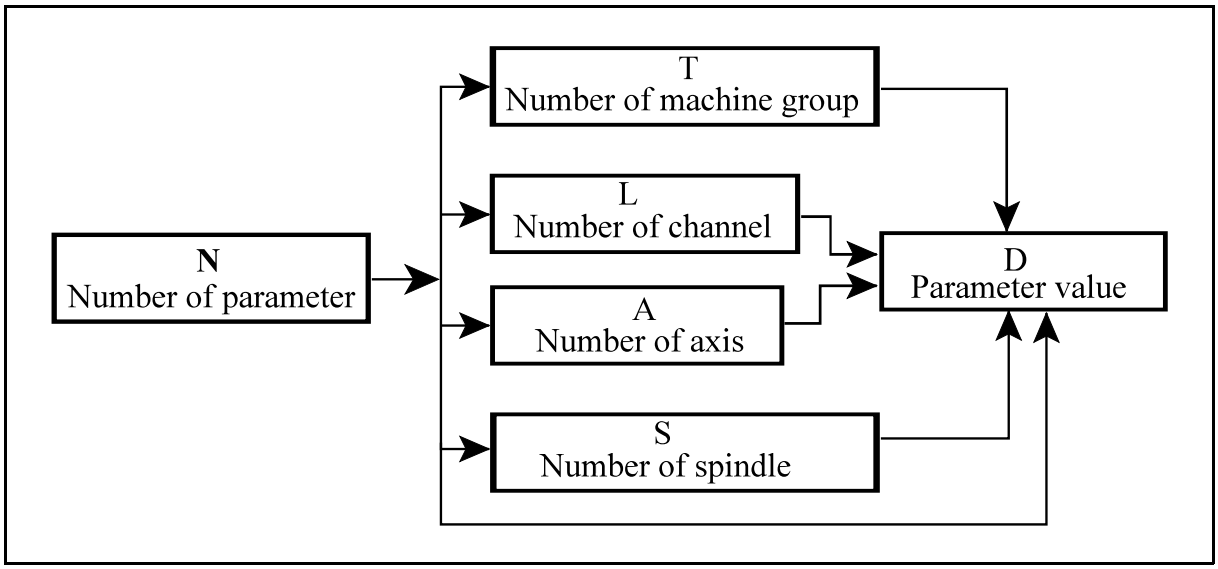
Address “A” carries out the classification of parameters repeated per axis. We distribute the A1...A32 axis addresses, with appropriate parameters between the various channels. The name of axes in the various channels may be the same; however, it may be different, as well. For example, the name of axis 1 in the first channel may be X, but also the name of axis 4 may be X in the 2nd channel.

1.4 Spindle (S=1...16)

Address “S” carries out the classification of parameters repeated per spindle. Spindles S1...S16 are distributed by the PLC between the channels, by writing appropriate registers.

A given parameter is not determined by only its ID or number **(N)**, but it can be classified also based on several aspects, thus they may occur multiple times. Classification aspects are the following:

- Common parameters, valid for all channels, without a specific designation
- Machine group, designation **T**,
- Channel, designation **L**,
- Axis, designation **A**,
- Spindle, designation **S**.



1.5 Parameter types

Parameters can be:

– **Bit type** parameters, value range:
0, or 1

– **Integer parameters** (DWORD), value range:
without a sign: 0. ... 4294967295, or
with a sign: -2147483648... +2147483647

– **Floating-point parameters** (double), value range:
in case of negative numbers. -1.7×10^{308} ... -5.0×10^{-324}
in case of positive numbers. 5.0×10^{-324} ... 1.7×10^{308}

2 System Config Parameter Group

N0000 Machine Group (L 1...8, integer, channel)

It tells which machine group is the given *channel* assigned to.
In case the parameter value is 0: the given channel does not exist.

N0000	Designation	Value	Note
L1	**T**	**1**	Channels 1-2 belong to machine group 1
L2	**T**	**1**	
L3	**T**	**2**	Channel 3 belongs to machine group 2.
L4	**Not used**	**0**	Not used

N0001 Control Type (L 1...8, integer, channel)

It tells what kind of a control shall be in the given *channel:*

=0: lathe
=1: mill

N0001	Designation	Value	Note
L1	**Lathe**	**0**	Channels 1-2 have a lathe control.
L2	**Lathe**	**0**	
L3	**Mill**	**1**	Channels 3-4 have a mill control.
L4	**Mill**	**1**	

N0002 Axis Assign (A1...32, integer, axis)

It tells whether the given axis exists or not and if yes, which channel it does belong to.
In case the parameter **value is 0: the given axis does not exist**!
In case the **parameter** value is **1-8** and the value of the appropriate **AN_DETCHA** PLC flag is =**0** the axis **does exist**:

For the display (position, offsets, etc.), in case the axis has a name, too;
For the block executor, in case the axis has a name, too;
For the PLC, even if it does not have a name;
For the interpolator and measuring system.

As soon as the appropriate **AN_DETCHA** PLC flag turns to **1**, the **axis terminates** and the record of the reference point is deleted!

☞ ***Warning!***

The maximum number of axes that can be assigned to a channel is 16!

In a channel, there could be **axes, which are measured only, that execute direct measurement in the dual position control loop** (linear encoder, angle encoder). It is not necessary to assign these devices to the master axis channel, thus these measured axes do not reduce the maximum 16 axes managed per channel.

If the value of the **parameter** is **99**, it will mean that the given axis is an axis which is measured only. For these axes, there will not be displaying the position.

N0002	Designation	Value	Note
A1	**L**	**1**	Axes 1-2 belong to channel 1
A2	**L**	**1**	
A3	**L**	**2**	Axes 3-4 belong to channel 2
A4	**L**	**2**	
A5	**L**	**3**	Axes 5-8 belong to channel 3
A6	**L**	**3**	
A7	**L**	**3**	
A8	**L**	**3**	
A9	**L**	**4**	Axes 9-10 belong to channel 4
A10	**L**	**4**	
A11	**L**	**4**	
A12	**Not used**	**0**	Not used

N0003 Channel Name (L 1...8, character, channel)

1-8 characters of the name of the given channel.

We refer to the channel name only during the identification of display screens, e.g.:

WORK ZERO-POINTS **Ch. 1**

WORK ZERO-POINTS **Ch. 2**

In case of a position display we mark the position of axes per channel by a subscript, e.g. X_1, X_2, X_3.

N0004 Nr. of Interface Cards (integer)

The parameter shall be filled out only in case of INT2000 or INT100 cards attached to an ISA bus!

It is the number of interface cards joined to ISA bus.

N0005 NC cycle time (integer)

The cycle time of the NC. Its possible values are:

Auto, 1000, 2000, 5000

In the case of Auto, the NC sets the cycle time automatically depending on the number of channels assigned in the parameter N0002 Axis Assign. If the number of channels are increased, the value of the parameter NC cycle time will have to be increased too.

3 Axis Config Parameter Group

N0100 Axis Name1 (A1... 32, integer, axis)

It shows the 1st character of the name of the given axis. It is mandatory to choose from the below values:

name	X	Y	Z	A	B	C	U	V	W
parameter value	88	89	90	65	66	67	85	86	87

In case the parameter value is =0, it is an axis without address and position readout, to which no reference can be made from a part program.
Within a channel the axis names shall be different, independently of the applied number of characters. The axis names may be the same in different channels.
For example, we may call the axes as X and Z both in channel 1 and 2.

For example:

N0100	Designation	Value	Note
A1	**X**	**88**	X (Channel 1, axis X...)
A2	**Z**	**90**	Z (Channel 1, axis Z...)
A3	**X**	**88**	X (Channel 2, axis X...)
A4	**Z**	**90**	Z (Channel 2, axis Z...)
A5	**X**	**88**	X (Channel 3, axis X...)
A6	**Y**	**89**	Y (Channel 3, axis Y...)
A7	**Z**	**90**	Z (Channel 3, axis Z...)
A8	**B**	**66**	B (Channel 3, axis B...)
A9	**Not used**	**0**	Not used

N0101 Axis Name2 (A1... 32, integer, axis)

It shows the 2nd character of the name of the given axis. Possible names: numbers 0,1, ..., 9 (related values: 48, 49, ..., 57, letters of the English alphabet: A, B, C, ..., Z (related values: 65, 66, 67, ..., 90).
In case the parameter value is: 0, there is no second and further (third) axis name.

For example:

N0101	Designation	Value	Note
A1	**M**	**77**	M (channel 1 axis XM...)
A2	**M**	**77**	M (channel 1 axis ZM...)
A3	**S**	**83**	S (channel 2 axis XS...)

N0101	Designation	Value	Note
A4	**S**	**83**	S (channel 2 axis ZS...)
A5	**3**	**51**	3 (channel 3 axis X3...)
A6	**3**	**51**	3 (channel 3 axis Y3...)
A7	**3**	**51**	3 (channel 3 axis Z3...)
A8	**3**	**51**	3 (channel 3 axis B3...)
A9	**Not used**	**0**	Not used

N0102 Axis Name3 (A1... 32, integer, axis)

It shows the 3rd character of the name of the given axis. Possible names: numbers 0,1, ..., 9 (related values: 48, 49, ..., 57, letters of the English alphabet: A, B, C, ..., Z (related values: 65, 66, 67, ..., 90).

In case the parameter value is: 0, there is no third axis name.

For example:

N0101	Designation	Value	Note
A1	**A**	**65**	A (channel 1 axis XMA...)
A2	**A**	**65**	A (channel 1 axis ZMA...)
A3	**L**	**76**	L (channel 2 axis XSL...)
A4	**L**	**76**	L (channel 2 axis ZSL...)
A5	**Not used**	**0**	Not used (channel 3 axis X3...)
A6	**Not used**	**0**	Not used (channel 3 axis Y3...)
A7	**Not used**	**0**	Not used (channel 3 axis Z3...)
A8	**Not used**	**0**	Not used (channel 3 axis B3...)
A9	**Not used**	**0**	Not used

☞ **Warning**: *The extended addresses shall not be used for passing arguments during the macro programming.*

☞ **Warning**: *In case the axis address ends with a number, e.g. Y2, in the part program a = sign shall be programmed after it: Y2=126.853.*

☞ **Warning**: *There shall not be any axes with the same names within the same channel, but it is allowed in different channels.*

N0103 Axis to Plane (A1...32, egész, axis)

From the aspect of circle interpolation (G2, G3), tool radius compensation calculation (G41, G42), plane selection (G17 ... G19), etc. one needs to know which axes are the basic axes (X, Y, Z) in the given system, which participate in plane selection; respectively, which are the axes parallel with them, which may participate in plane selection too. ***In each channel there can be only 1-1 basic axis (X, Y, Z)***, but there may be several parallel axes.

Setting	Meaning
0	it does not participate in plane selection, e.g. rotary axis: B
1	X basic axis (e.g.: X)
2	Y basic axis (e.g.: Y1)
3	Z basic axis (e.g.: ZM)
5	axis parallel with axis X (e.g.: U)
6	axis parallel with axis Y (e.g.: Y2)
7	axis parallel with axis Z (e.g.: ZS)

For example:

N0103	Designation	Value	Note
A1	**X base**	**1**	axis XMA basic axis X
A2	**Z base**	**3**	axis ZMA basic axis Z
A3	**X base**	**1**	axis XSL basic axis X
A4	**Z base**	**3**	axis ZSL basic axis Z
A5	**X base**	**1**	axis X3 basic axis X
A6	**Y base**	**2**	axis Y3 basic axis Y
A7	**Z base**	**3**	axis Z3 basic axis Z
A8	**Not used**	**0**	axis B3 does not participate in plane selection
A9	**Not used**	**0**	Not used

N0104 Unit of Measure (L1...8, bit, channel)

N0104 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
								IND

#0 **IND**: it indicates whether in the given channel the measurement is done in metric or inch unit along the linear axes:

=0: metric

=1: inch

N0105 Increment System (L1...8, integer, channel)

The parameter tells the number of decimal places by which the
– position display,
– measuring system signals (e.g. lag)
– position data (offsets, etc.)
have to be displayed on the given axis.
The internal position representation of the system, the output increment system is independent from the parameter value of the Increment System:

At linear axes, in case of metric measurement: 10^{-6} mm,
At linear axes in case of inches measurement: 10^{-7} inch,
In case of rotary axes: 10^{-6} degrees.

Its possible values: 1, ..., 5

Designation	Value	Axis	Display accuracy depending on the unit system	
			G21 metric	**G20 inch**
ISA	1	linear	0.01 mm	0.001 in
		rotary	0.01 degree	0.01 degree
ISB	2	linear	0.001 mm	0.0001 in
		rotary	0.001 degree	0.001 degree
ISC	3	linear	0.0001 mm	0.00001 in
		rotary	0.0001 degree	0.0001 degree
ISD	4	linear	0.00001 mm	0.000001 in
		rotary	0.00001 degree	0.00001 degree
ISE	5	linear	0.000001 mm	0.0000001 in
		rotary	0.000001 degree	0.000001 degree

N0106 Axis Properties (A1...32, bit, axis)

N0106 A1...32	#7	#6	#5	#4	#3	#2	#1	#0
	HYP	**CSA**	**MGD**	**VIR**	**IDX**	**PER**	**ROT**	**DIA**

#0 **DIA**: It indicates whether given axis is programmed in diameter or radius. If the AP_DIARAD PLC flag turns from 0 to 1, the handle will be contradictory with the parameter, and if it turns from 1 to 0, the handle will be the same as the parameter. E.g.: in case the parameter is DIA=1, the display and data input will be made in diameter on the given axis. If the PLC sets the AP_DIARAD flag, the display and data input will be made in radius on the nth axis, till the AP_DIARAD flag is not written back to 0 by the PLC.

#1 **ROT**: It indicates whether the given axis is a linear or rotary one.
=0: linear axis: an ***inch - metric conversion needs to be carried out for the axis***,
=1: rotary axis:an ***inch - metric conversion does not need to be carried out for the axis***.

#2 **PER**: It indicates whether there is a periodicity on the given axis or not. A linear (ROT=0) as well as a circular axis (ROT=1) may be periodic. The amount of periodicity is indicated by the Per Amount parameter. It means that it is possible to set a periodic pitch error compensation for the axis.

#3 **IDX**: if the parameter value is 1, the axis is indexed, i.e. it has to be moved to a discrete position. It becomes clampable in discrete positions. E.g.: Hirth-type rotary table, which may be clamped down by 5 degrees.

#4 **VIR**: If there is only measuring device on an axis, but it is not possible to close the position control loop on this given axis, the axis has to be assigned as virtual axis by writing the parameter into 1. The control never issues movement command to these axes.

The axes that are measured only are such ones whose position can be set only manually.

In the case of using dual control loop, the master set in the parameter N0534 Dual Position Axis Number always has to be assigned as virtual axis too!

#5 **MGD**: It controls switching over the programming in radius/in diameter. If the parameter value

=0: the switching over the programming in radius/in diameter occurs by the AP_DIARAD PLC flag (e.g. by an M code),

=1: the switching over the programming in radius/in diameter occurs by the instruction G10.9.

#6 **CSA**: Not used.

#7 **HYP**: If its value is 1, the given axis is a hypothetical axis.

The hypothetical axis has a name, is assigned to a given channel and every configuration parameters are related to it, except for the fact that no servo circuit belongs to it. The physical parameters of the hypothetic axis (servo, refpoint, limit position, etc.) are not filled in.

Normally we cannot refer to it from a part program.

The hypothetical axis revives when the PLC, another non-hypothetic (real) axis, with appropriate servo parameters and motors, switches to the axis under this address. (Axis exchange, Composit control). Then we can program the given axis on the hypothetical axis address in its own channel.

E.g.: In channel 1 axis Y is the 2nd axis and HYP=0. The belonging servo parameters, etc. are set.

In channel 2 axis Y is addressed as the 6th axis and it is a hypothetical axis HYP=1. For this there aren't any servo parameters filled out and therefor nor drives neither motors are belonging to it.

Upon a PLC command, axis Y of channel 1 becomes exchanged with axis Y of channel 2. In such a case we cannot refer to an Y address in channel 1; but we can refer to an Y address in channel 2 and we can move the given axis, as channel 2 has received a real, physically movable axis.

N0107 RollOver Control (A1...32, bit, axis)

N0107 A1...32	#7	#6	#5	#4	#3	#2	#1	#0
					RND	**ABS**	**ASH**	**REN**

#0 **REN**: (Rollover ENable) If the ROT=1 it enables the roll-over function of the rotary axis based on the RollAmount parameter.

REN=0: the roll-over function is disabled.

REN=1: the roll-over function is enabled. The absolute and relative position displays become displayed with the following conditions:

0≤ABSOLUTE POSITION<RollAmnt

#1 **ASH**: (Absolute SHort) If on the given axis the ROT=1, REN=1, ABS=0 and

ASH=0: it moves always in the direction according to the given sign, in case of an absolute programming. E.g.: If B=0 is the starting position, upon the instruction G90 B380 the movement will be +20 and the absolute position at the end of movement will be B=20. (Provided that ABS=0)

ASH=1: moves always on the shorter path in case of absolute programming. E.g.: If the starting position is B=0, upon the instruction of G90 B340 the movement will be -20, and the absolute position at the end of the movement will be B=340.

#2 **ABS**: If on the given axis ROT=1, REN=1 and ASH=0:

ABS=0: it moves always in the direction according to the given sign in case of absolute programming. E.g.: If B=0 is the starting position G90 B-30 upon the instruction the movement will be -30, and the absolute position at the end of the movement will be B=330. (See ASH)

ABS=1: it moves in the direction according to the given sign and the position will be the absolute value of the programmed position. E.g.: if the starting position is B=0, upon the instruction G90 B-30 the movement will be -330 and the absolute position at the end of the movement will be B=30.

#3 **RND**: If on the given axis the parameters ROT=1, REN=1 are valid, in case of incremental programming it will move always to the direction according to the given sign. If the parameter is:

=0: it will not apply the parameter Roll Amount for the programmed path. E.g.: if the starting position is B=0, upon the instruction G91 B-450 the movement will be -450 and the absolute position at the end of the movement will be B=270.

=1: it willl apply the parameter Roll Amount for the programmed path. E.g.: if the starting position is B=0, upon the instruction G91 B-450 the movement will be -90 and the absolute position at the end of the movement will be B=270.

N0108 RollAmount (A1...32, floating-point, axis)

If on the given axis #1 ROT=1, #3 REN=1, i.e. the axis is a rotary axis, and the roll-over function is enabled, it carries out the roll-over handling based on the Roll Amount parameter.

The number written on the parameter may contain integers and decimals, too; e.g.: 360.0

N0109 PerAmount (A1...32, floating-point, axis)

Distance data, the number written on the parameter, based on the parameter #0 IND is to be interpreted in inches or millimetres, if #1 ROT=0.

If #1 ROT=1 the data becomes interpreted in degrees.
If on the given axis a parameter setting of PER=1 is set, there is a periodicity on the given axis. We write the level of periodicity onto this parameter.
E.g.: a heavy linear axis is moved by a motor by using a gear. If the periodicity of the output axis of the gear, which moves the ball screw, is 25 mm related to the displacement of the axis, then we write 25 on this parameter. If the measuring is carried out by the rotary encoder fixed to the axis of the motor, the linear error of the gear, which is periodic along the axis length, shall be compensated. This data is necessary for the compensation. The same applies also to rotary axes, too.

N0110 Indexing Amount (A1...32, floating-point, axis)

Distance data, the number written on the parameter, based on the parameter #0 IND is to be interpreted in inches or millimetres, if #1 ROT=0.
If #1 ROT=1 the data becomes interpreted in degrees.
The control takes it into consideration if on the Axis Properties parameter value IDX=1 is indicated. If on the given axis the position measured on the measuring system is located in the InPos-radius environment of the integer multiple of the Indexing Amount value, the AN_INDP PLC flag becomes true. Then the given axis can be clamped.
E.g.: if the Hirth-type rotary table can be clamped in each 5 degrees, let's write 5 to this parameter. Then the AN_INDP flag becomes true per 0, 5, 10, ...355 degrees.

4 Coordinates Parameter Group

N0200 Reference Position1 (A1...32, floating-point, axis)

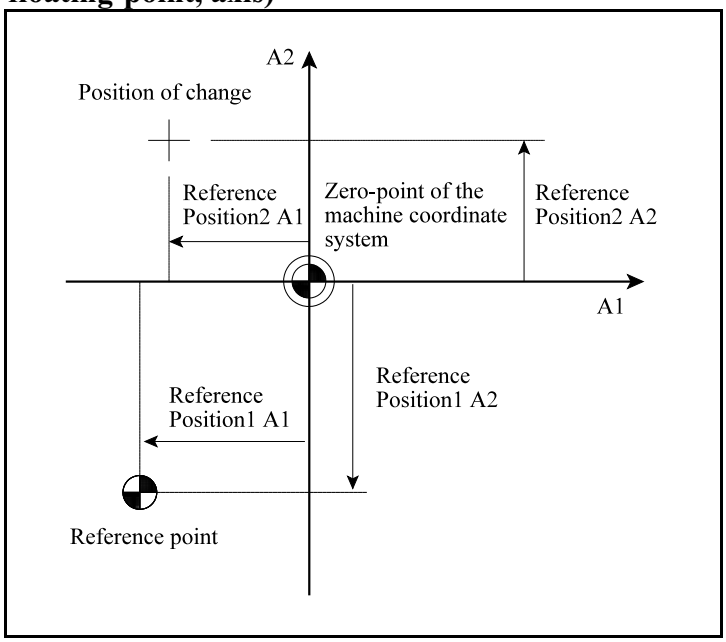
Distance data, the number written on the parameter, based on the parameter #0 IND is to be interpreted in inches or millimetres, if #1 ROT=0.
If #1 ROT=1 the data becomes interpreted in degrees.
It indicates the position of the reference point in the machine coordinate system. ***In case of an absolute, or distance-coded measuring system it is the location where the measuring system position is 0***.

☞ **Warning!**

On every machine tool, it is necessary to set the zero point of the machine coordinate system by the use of mechanical measuring devices, i.e. gauge blocks, dial gauges and measuring pointer. If, later, the measuring system of the machine tool will be disassembled, or the battery in the absolute measuring device will run down and the position will be lost because of it, this point will have to be measured, and the Reference Position1 parameter will have to be set again. With this, necessity to measure other position data (limit, change positions, compensations) again can be avoided.

N0201 Reference Position2 (A1...32, floating-point, axis)
N0202 Reference Position3 (A1...32, floating-point, axis)
N0203 Reference Position4 (A1...32, floating-point, axis)

Distance data, the number written on the parameter, based on the parameter #0 IND is to be interpreted in inches or millimeters, if #1 ROT=0.
If #1 ROT=1 the data becomes interpreted in degrees.
They are 3 arbitrary position sets, e.g. position of change - specified in the machine coordinate system. In a part program upon the command G30 P2, P3, P4 the control moves to this point.



N0204 Distance of DTP (L1...8, floating-point, channel)

Not used.

N0205 XN Contact Position S1 (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on the parameter #0 IND is to be interpreted in inches or millimetres.

The position of that manual tool sensor button, which is operated by the negative movement of axis X.

N0206 XP Contact Position S1 (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on the parameter #0 IND is to be interpreted in inches or millimetres.
The position of that manual tool sensor button, which is operated by the positive movement of axis X.

N0207 YN Contact Position S1 (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on the parameter #0 IND is to be interpreted in inches or millimetres.
The position of that manual tool sensor button, which is operated by the negative movement of axis Y.

N0208 YP Contact Position S1 (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on the parameter #0 IND is to be interpreted in inches or millimetres.
The position of that manual tool sensor button, which is operated by the positive movement of axis Y.

N0209 ZN Contact Position S1 (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on the parameter #0 IND is to be interpreted in inches or millimetres.
The position of that manual tool sensor button, which is operated by the negative movement of axis Z

N0210 ZP Contact Position S1 (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on the parameter #0 IND is to be interpreted in inches or millimetres.
The position of that manual tool sensor button, which is operated by the positive movement of axis Z.

N0211 XN Contact Position S2 (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on the parameter #0 IND is to be interpreted in inches or millimetres.
If on a single turret machine with subspindle we use two tool sensors, it is the position of the manual tool sensor button belonging to spindle S2, which is operated by the negative-direction movement of axis X.

N0212 XP Contact Position S2 (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on the parameter #0 IND is to be interpreted in inches or millimetres.
If on a single turret machine with subspindle we use two tool sensors, it is the position of the manual tool sensor button belonging to spindle S2, which is operated by the positive-direction movement of axis X.

N0213 YN Contact Position S2 (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on the parameter #0 IND is to be interpreted in inches or millimetres.
If on a single turret machine with subspindle we use two tool sensors, it is the position of the manual tool sensor button belonging to spindle S2, which is operated by the negative-direction movement of axis Y.

N0214 YP Contact Position S2 (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on the parameter #0 IND is to be interpreted in inches or millimetres.
If on a single turret machine with subspindle we use two tool sensors, it is the position of the manual tool sensor button belonging to spindle S2, which is operated by the positive-direction movement of axis Y.

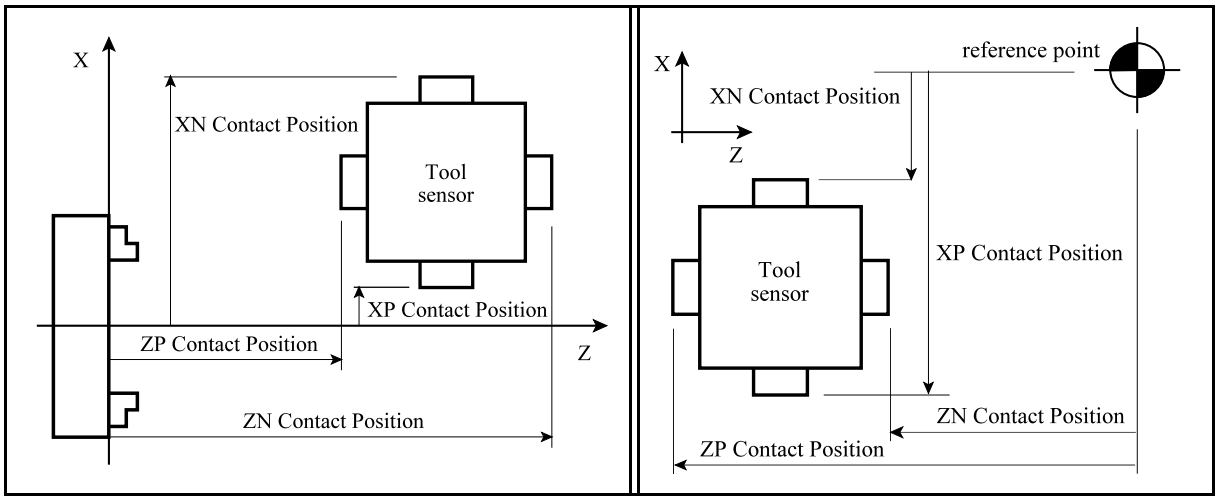
N0215 ZN Contact Position S2 (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on the parameter #0 IND is to be interpreted in inches or millimetres.
If on a single turret machine with subspindle we use two tool sensors, it is the position of the manual tool sensor button belonging to spindle S2, which is operated by the negative-direction movement of axis Z.

N0216 ZP Contact Position S2 (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on the parameter #0 IND is to be interpreted in inches or millimetres.
If on a single turret machine with subspindle we use two tool sensors, it is the position of the manual tool sensor button belonging to spindle S2, which is operated by the positive-direction movement of axis Z.

We can measure the positions of the tool sensor buttons for the coordinate system assigned to the chuck and also to the reference point.

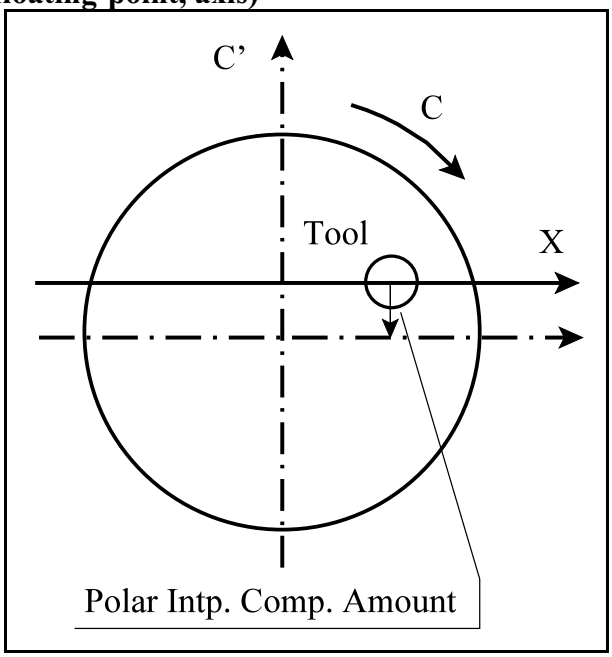


N0217 Polar Intp. Comp. Amount (A1...32, floating-point, axis)

Distance data, the number written on the parameter, based on the #0 IND parameter is to be interpreted in inch or millimeter.

During polar interpolation (G12.1), it indicates the distance between the center of rotation of the rotary axis (the second axis of the selected plane, virtual axis) and the first axis of the plane (linear axis). In other words, the distance between the center of rotation of the rotary axis and the axis of rotation of the tool in the direction perpendicular to the linear axis.

The value of compensation has to **always** be written **at the address of the rotary axis** in millimeter or in inch.



5 Feedrate Parameter Group

N0300 Default F G94 (L1...8, floating-point, channel)

Speed data; the number written on the parameter, based on parameter #0 IND is to be interpreted in inch/minute or mm/minute if #1 ROT=0.
If #1 ROT=1 the data is interpreted in degree/minute.
After power on, it is the default value of the feedrate in G94 state.

N0301 Default F G95 (L1...8, floating-point, channel)

Speed data; the number written on the parameter, based on parameter #0 IND is to be interpreted in inch/revolution or mm/revolution if #1 ROT=0.
If #1 ROT=1 the data is interpreted in degree/revolution.
After power on, it is the default value of the feedrate in G95 state.

N0302 Ref Rapid (A1...32, floating-point, axis)

Speed data; the number written on the parameter, based on parameter #0 IND is to be interpreted in inch/minute or mm/minute if #1 ROT=0.
If #1 ROT=1 the data is interpreted in degree/minute.
Before the reference position setting on the given axis it it's the highest allowed speed.
During the reference position setting it indicates the speed of searching for the deceleration switch.

N0303 Ref Feed (A1...32, floating-point, axis)

Speed data; the number written on the parameter, based on parameter #0 IND is to be interpreted in inch/minute or mm/minute if #1 ROT=0.
If #1 ROT=1 the data is interpreted in degree/minute.
During reference position setting:
if SWT=1 and GRI=0 (by using the deceleration switch): it is the speed of deceleration,
if SWT=1 and GRI=1 (on the first zero pulse after the deceleration switch): it indicates the speed of deceleration and searching for the zero pulse,
if SWT=0 and GRI=1 (reference position setting on the grid): it is the speed of zero pulse searching,
if DCD=1 (distance coded reference position setting): it is the speed of searching for zero pulses.

N0304 Rapid (A1...32, floating-point, axis)

Speed data; the number written on the parameter, based on parameter #0 IND is to be interpreted in inch/minute or mm/minute if #1 ROT=0.
If #1 ROT=1 the data is interpreted in degree/minute.
During the positioning in a part program it specifies the value of the rapid traverse rate per axis.

N0305 Max Feed (A1...32, floating-point, axis)

Speed data; the number written on the parameter, based on parameter #0 IND is to be interpreted in inch/minute or mm/minute if #1 ROT=0.
If #1 ROT=1 the data is interpreted in degree/minute.

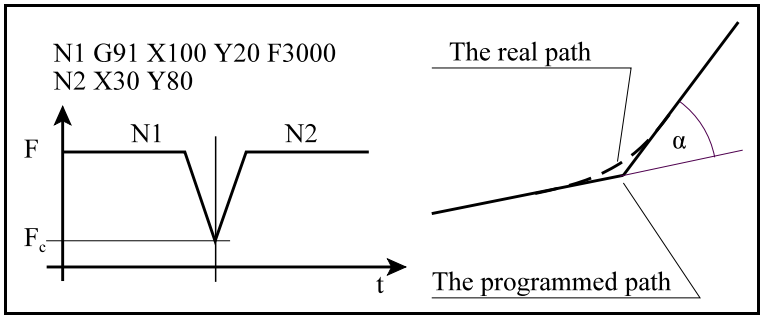
In case of a movement by feed from a part program it limits the feed components per axes. During a dry run it determines the rate of feed movements based on this parameter.

N0306 Feed Control (L1...8, bit, channel)

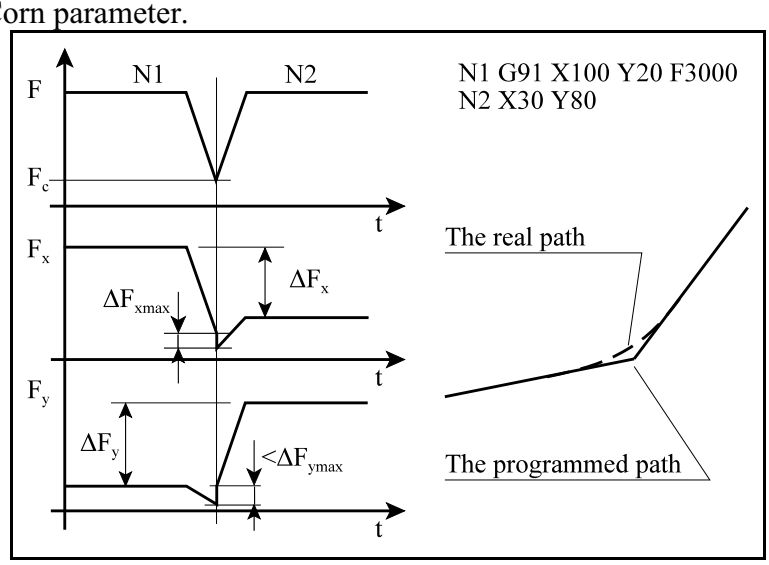
N0306 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
							GEO	**FDF**

#0 **FDF**: If the parameter

=0: at the transition of two consecutive blocks it determines the angle included by the two blocks and if it is greater than the Crit A Diff parameter it decelerates to the feed specified on Feed Corn parameter.



=1: at the transition of two consecutive blocks it determines the differencies in feedrate components per axis and if any of them greater than the value specified at Crit F Diff parameter it decelerates the feedrate in a way not to exceed the speed difference set by parameter.



#1 **GEO**: If the parameter

Not used.

N0307 Crit A Diff (L1...8, bit, channel)

Data of angle, its interpretation is ° (degree).

If the Feed Control #0 FDF=0, at the transition of two consecutive blocks the deceleration is made based on the critical angular difference set on this parameter. The deceleration is made to the feedrate determined on Feed Corn parameter.

N0308 Feed Corn (L1...8, bit, channel)

Speed data; the number written on the parameter, based on parameter #0 IND is to be interpreted in inch/minute or mm/minute if #1 ROT=0.

If #1 ROT=1 the data is interpreted in degree/minute.
If the Feed Control #0 FDF=0, and at the corners the value of the angle between the two blocks exceeds the value set on the Crit A Diff parameter, the control will decelerate to this feed.

N0309 Crit F Diff (A1...32, floating-point, axis)
Speed data; the number written on the parameter, based on parameter #0 IND is to be interpreted in inch/minute or mm/minute if #1 ROT=0.
If #1 ROT=1 the data is interpreted in degree/minute.
If the Feed Control #0 FDF=1, it specifies the allowable difference in feedrate components at the transition of two consecutive blocks per axis, in 1/min unit. It decelerates by the transition point between the two blocks to a feedrate at which the speed difference cannot exceed the value determined therein on neither of the axes.

N0310 Circ F Min (L1...8, floating-point, channel)
Speed data; the number written on the parameter, based on parameter #0 IND is to be interpreted in inch/minute or mm/minute.
By the machining of circular arcs the control limits the level of the feed F based on the function

$$F = \sqrt{a \times r}$$

where:
a: the lower one from among acceleration values determined for axes participating in circle interpolation,
r: is the radius of the circle.
In order to prevent the feed from decreasing through any levels, we can determine on the Circ F Min parameter a minimum feed value. If the calculated feed is lower than the value determined on the Circ F Min parameter (F<Circ F Min) the control will take the feedrate from the Circ F Min parameter.

N0311 G31 Feed (L1...8, floating-point, channel)
Speed data; the number written on the parameter, based on parameter #0 IND is to be interpreted in inch/minute or mm/minute if #1 ROT=0.
If #1 ROT=1 the data is interpreted in degree/minute.
If the control executes a G31 instruction, and the **SKF** parameter value 1, it takes the feedrate from this parameter.

N0312 G37 Feed (L1...8, floating-point, channel)
Speed data; the number written on the parameter, based on parameter #0 IND is to be interpreted in inch/minute or mm/minute if #1 ROT=0.
If #1 ROT=1 the data is interpreted in degree/minute.
If the control executes a G37 instruction, and the **TLF** parameter value is 1, it takes the feedrate from this parameter.

N0313 Feed Mult (L1...8, floating-point, channel)
A multiplier which specifies the number by which the programmed feedrate value is multiplied, if the operator keeps pushing the rapid traverse rate jog button during cutting.

N0314 Manual Rapid (A1...32, floating-point, axis)

Speed data; the number written on the parameter, based on parameter #0 IND is to be interpreted in inch/minute or mm/minute if #1 ROT=0.
If #1 ROT=1 the data is interpreted in degree/minute.
In jog mode, in case of pushing the rapid traverse rate button, the axes are going to move with this value which can be set per axis.
In incremental jog mode it moves with the selected increment by this speed.
In manual handle mode this value limits the speed of the movement.

N0315 Jog Max Feed (A1...32, floating-point, axis)

Speed data; the number written on the parameter, based on parameter #0 IND is to be interpreted in inch/minute or mm/minute if #1 ROT=0.
If #1 ROT=1 the data is interpreted in degree/minute.
In jog mode this parameter limits the speed per axes.

N0316 Jog F Contr (L1...8, integer, channel)

In Jog mode it determines the volume of the applied feed. Its value range: 1...3.

Designation	Value	Operation
JFT	1	The value of the jog feed is set based on an **CP_JOGFD** PLC register value, according to an exponential function. The feed is given out always in the unit of 1/min. It is advisable to link the value of CP_JOGFD register to the status of the override switch.
JFP	2	It takes the value of the jog feed: from parameter JOG F G94 in G94 state, and from parameter JOG F G95 in G95 state. In G95 state a spindle rotation is necessary for the movement, too.
JFM	3	It takes the value of the jog feed from the modal F value. It interprets the feed value in G94 state: in unit 1/min in G95 state: in unit 1/revolution In G95 state a spindle revolution is necessary for the movement, too.

The below table indicates the feed value depending on the CP_JOGFD register value, up to 15, if the JFT parameter is set.

CP_JOGFD	G21 mm/min	G20 in/min	rotary axis °/min
0	0	0	0
1	2	0.08	0.4
2	3.2	0.12	0.64
3	5	0.2	1
4	7.9	0.3	1.58
5	12.6	0.5	2.52
6	20	0.8	4
7	32	1.2	6.4
8	50	2	10
9	79	3	15.8
10	126	5	25.2
11	200	8	40
12	320	12	64
13	500	20	100
14	790	30	158
15	1260	50	252

N0317 Jog F G94 (L1...8, floating-point, channel)

Speed data; the number written on the parameter, based on parameter #0 IND is to be interpreted in inch/minute or mm/minute if #1 ROT=0.
If #1 ROT=1 the data is interpreted in degree/minute.
If the Jog F Contr parameter value is 2 (JFP), in G94 state it takes the jog feed value from this parameter.

N0318 Jog F G95 (L1...8, floating-point, channel)

Speed data; the number written on the parameter, based on parameter #0 IND is to be interpreted in inch/rev or mm/rev if #1 ROT=0.
If #1 ROT=1 the data is interpreted in degree/rev.
If the Jog F Contr parameter value is 2 (JFP), in G95 state it takes the jog feed value from this parameter. Spindle rotation is necessary for the movement, too.

N0319 T Meas Feed (L1...8, floating-point, channel)

Speed data; the number written on the parameter, based on parameter #0 IND is to be interpreted in inch/minute or mm/minute if #1 ROT=0.
In case of an automatic tool length measurement, when we approach the tool sensor by using the jog buttons, the slide is moving with this feed.

N0319+n Rn S Rapid n=1...8 (S1...S16, floating-point, spindle)

n=1...8: separate parameter for each element of the range between 1...8
Measurement unit: revolution/min.
After the command M19, it is the rapid traverse rate of spindle axes per range, in the unit of 1/min.

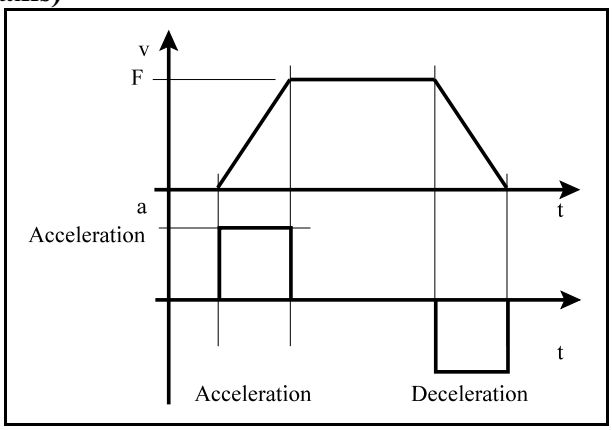
N0327+n Rn S F Max n=1...8 (S1...S16, floating-point, spindle)

n=1...8: separate parameter for each element of the range between 1...8
Measurement unit: revolution/min.
After the command M19, it is the allowed feed rate of spindle axes per range, in the unit of 1/min.

6 Acceleration Parameter Group

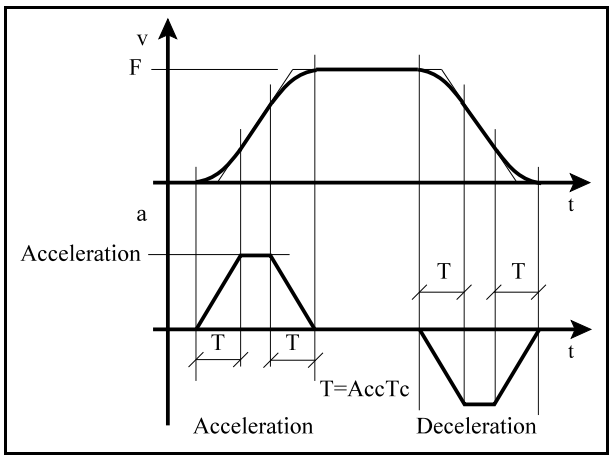
N0400 Acceleration (A1...32, floating-point, axis)

Acceleration data, the number written on the parameter is to be interpreted based on the #0 IND parameter in inch/sec^2, or mm/sec^2, if the #1 ROT=0.
If the #1 ROT=1 the data is to be interpreted in degree/sec^2.
It determines the level of acceleration per axes.



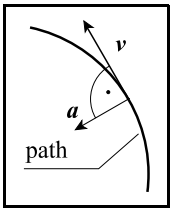
N0401 Acc Tc (A1...32, floating-point, axis)

Time data, it is set in seconds (sec).
It determines the time constant of acceleration per axis, i.e. the length of time during which the acceleration - value shall reach the value determined on the Acceleration parameter.



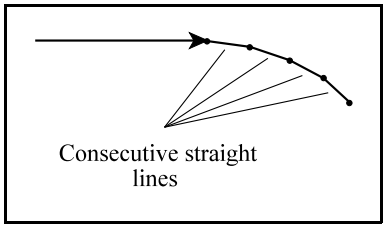
N0402 Normal Acc (A1...32, floating-point, axis)

Acceleration data, the number written on the parameter is to be interpreted based on the #0 **IND** parameter in inch/sec^2,or mm/sec^2, if the #1 ROT=0.
If the #1 ROT=1 the data is to be interpreted in degree/sec^2.
It limits the extent of (normal-direction) acceleration emerging in a direction perpendicular to the path, per axis.



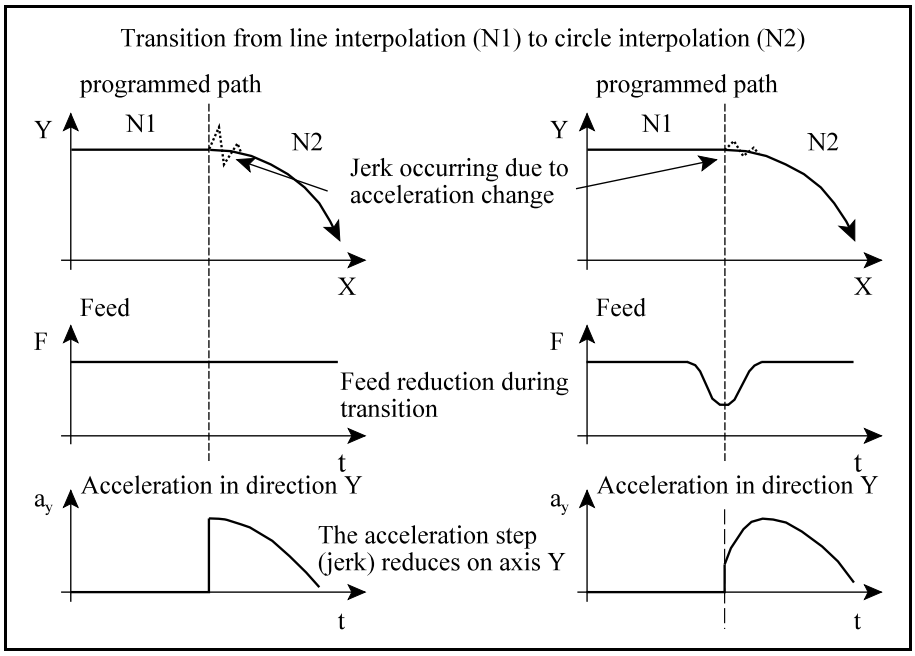
N0403 Acc Diff (A1...32, floating-point, axis)

Acceleration data, the number written on the parameter is to be interpreted based on the #0 **IND** parameter in inch/sec^2,or mm/sec^2, if the #1 ROT=0.
If the #1 ROT=1 the data is to be interpreted in degree/sec^2.
It determines the allowed acceleration step per axis, at block transitions, between two straight sections. Till the transition point between the two blocks it decelerates to a feed by which the acceleration change (jerk) will not exceed the value set there, on neither of the axes.



N0404 Acc Diff Circ (A1...32, floating-point, axis)

Acceleration data, the number written on the parameter is to be interpreted based on the #0 **IND** parameter in inch/sec^2,or mm/sec^2, if the #1 ROT=0.
If the #1 ROT=1 the data is to be interpreted in degree/sec^2.
At block transitions, if one of the blocks is a circle, it determines the extent of acceleration change (jerk) allowed per axis. Till the transition point between the two blocks it decelerates to a feed by which the acceleration step will not exceed the value set there, on neither of the axes.



N0404+n Rn S Acc n=1...8 (S1...S16, floating-point, spindle)

n=1...8: separate parameter for each element of the range between 1...8
Measurement unit: revolution/sec^2.
After the command M19, it determines the acceleration rate of spindle axes per range, in the unit of 1/sec^2.

N0412+n Rn S Acc Tc n=1...8 (S1...S16, floating-point, spindle)

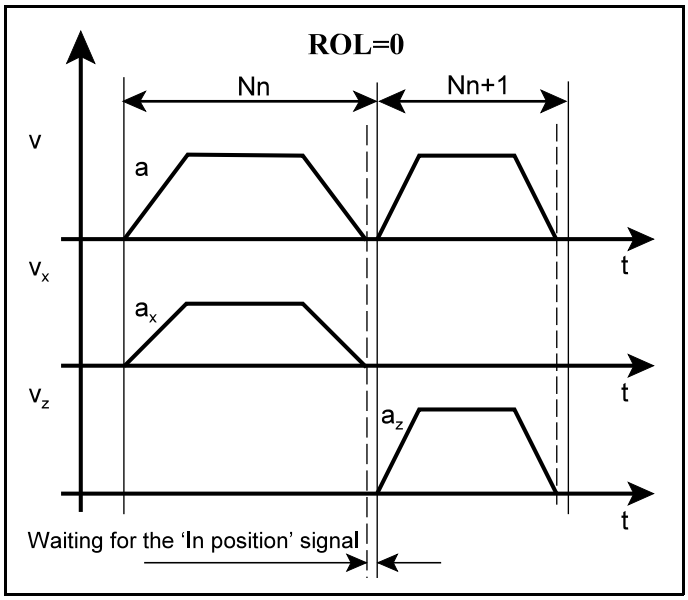
n=1...8: separate parameter for each element of the range between 1...8
Time data, determined in seconds (sec).
It determines the time constant of acceleration per spindle per range.

N0421 Acc Contr (L1...8, bit, channel)

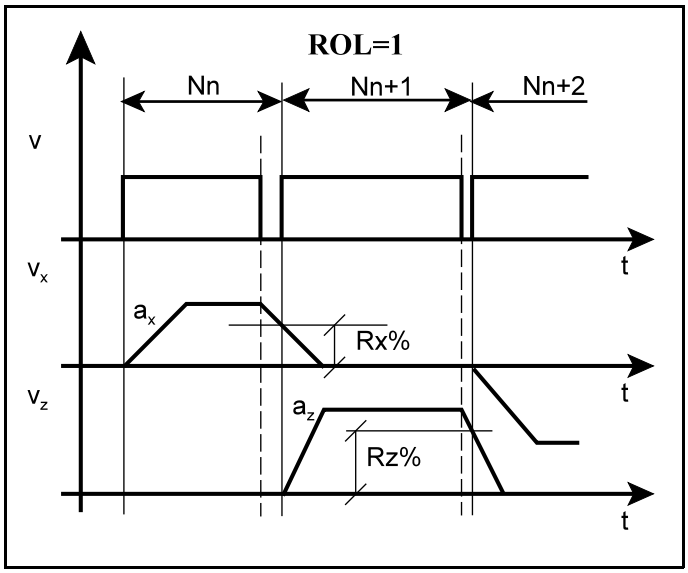
N0421 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
							JRK	**ROL**

#0 **ROL**: Allowance of overlapping between rapid traverse rate movements. During rapid traverse rate positioning, if the positioning block is followed by a positioning block and the parameter value is

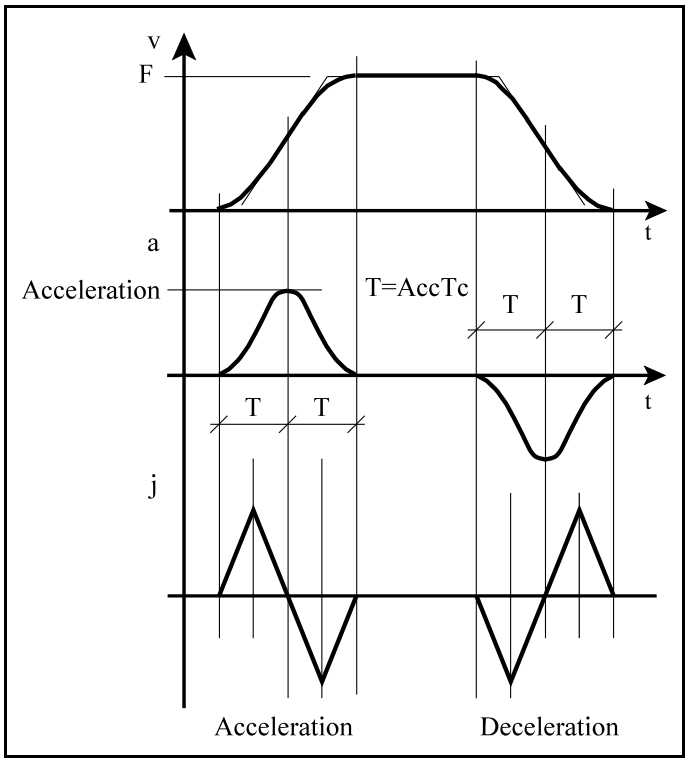
=0: the positioning will be made by acceleration before interpolation. The start of the next block will wait till the speed reduces to 0, and by PCH=1 parameter value it will wait for the "In Position" signal on axes participating in the movement.



=1: the positioning will be made by an acceleration after interpolation. The start of the next block will wait only for the reduction of the speed value on all axes - under the percentage value set on the Rapid Reduct. Ratio parameter. The PCH bit of parameter N1337 Execution Config shall be set to 0!



#1 **JRK**: Using it, the jerk in the first derivative of the acceleration function can be eliminated:
=0: there is jerk in the first derivative (j) of the acceleration function,
=1: there is no jerk in the first derivative (j) of the acceleration function.



N0422 Rapid Reduct. Ratio (A1...32, floating-point, axis)
The unit of the number written on the parameter is %.
The control will take it into consideration if the overlap between rapid traverse rate movements is allowed by the ROL=1 parameter setting. It will move on to the next positioning block or a block containing a movement if on all axes participating in the movement the level of feed components decreases under value (Rapid*Rapid Reduct. Ratio/100).

N0423 JerkRate (A1...32, floating-point, axis)
Not used.

7 Axis Servo Parameter Group

N0500 Axis Input Type (A1...32, integer, axis)

The hardware type belonging to the **encoder input** of the given axis. The selection can be made from a drop-down menu.

N0501 Axis Input Address (A1...32, integer, axis)

The periphery address of the unit (NCT drive, measuring system card) belonging to the **encoder input** of the given axis.

☞ ***Warning!*** *The selected address is not in connection with its number determined on the given axis parameter. E.g.: on the parameter axis Z is specified as axis A3, but the address of its encoder may be e.g. 5.*

If the parameter value**=0**: **there is no position measurement** on the given axis.
If the parameter value**>0**, **there is position measurement** on the given axis.
The output and input address belonging to the given axis may be the same but also different!

N0502 Axis Output Type (A1...32, integer, axis)

The hardware type belonging to the output of the given axis. The selection can be made from a drop-down menu.

N0503 Axis Output Address (A1...32, integer, axis)

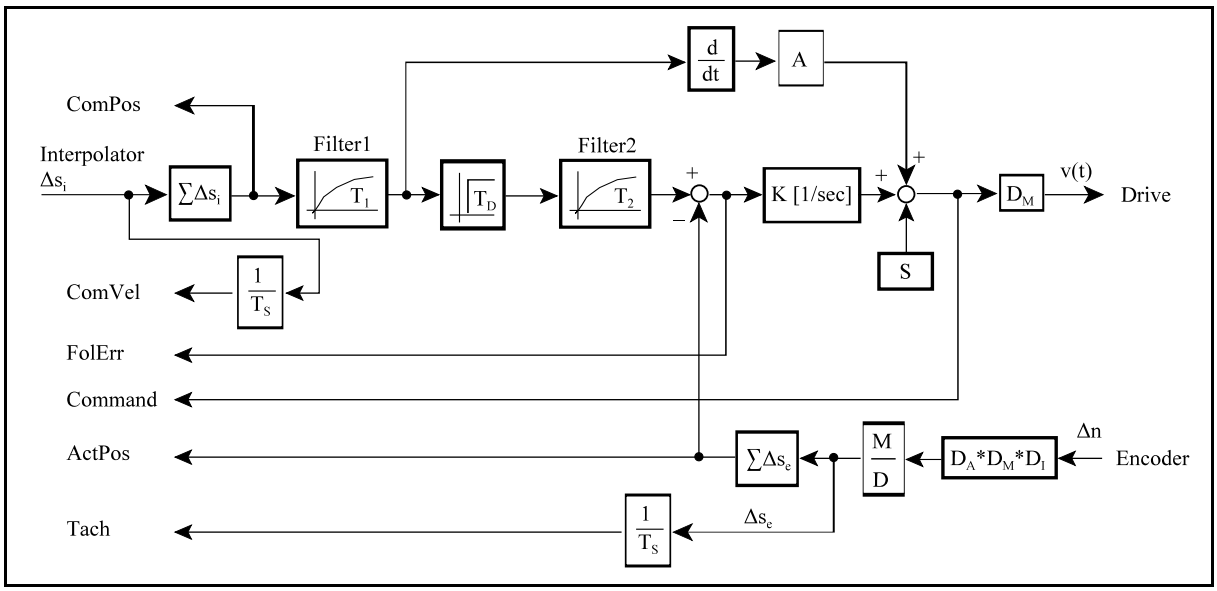
The periphery address of the unit (NCT drive, D-A converter card etc.) belonging to the output of the given axis, to which the measuring system outputs (e.g. command signal), respectively, the PLC control bits (e.g. drive enable, etc.) shall be issued.

☞ ***Warning!*** *The selected address is not in connection with its number determined on the given axis parameter. E.g.: on the parameter axis Z is specified as axis A3, but the address of its output may be e.g. 15.*

If the parameter value**=0**: **there is no command signal output** on the given axis.
If the parameter value**>0**, **there is command signal output** on the given axis.
The same command signal address may be assigned to a spindles or to an axis too:
– In case a spindle drive participates in polar interpolation as axis C, the same address shall be specified both to axis C and the spindle drive. The AP_DETCHR and SP_SDETCHR PLC flags will decide, the output of axis C or spindle S will be issued to the drive.

☞ ***Warning!*** *The input and output address belonging to the given axis may be the same but also different, depending on the applied peripheries! For example axis A3 Z gives command signals to drive with address No. 3, but it takes the signal of the linear scale from periphery with address No. 13.*

On the figure below we can see the diagram of the position control loop. The marks (K, A, etc.) on the diagram are in accordance with parameters. Signals displayable on the scope or measuring system test screen are indicated on it, too (ComPos, FolErr, etc). Measurement units of signals are: mm, inch, degree, mm/sec, inch/sec, deg/sec.



N0504 Position Gain (A1...32, floating-point, axis)

The position control loop gain.

Measurement unit: 1/sec.

The relationship between the loop gain value (K) and the position control lag (following error, FolErr), depending on feed (F) is the following:

$$FolErr = \frac{F}{60K}$$

If the above equation is not met, the Position Calibration parameter on the control, or the reference input gain of the drive shall be set.

N0505 Position Calibration (A1...32, floating-point, axis)

Speed data; the number written on the parameter, based on parameter #0 IND is to be interpreted in inch/min or mm/min if #1 ROT=0.

If #1 ROT=1 the data is interpreted in degree/min.

It is a value serving for the calibration of speed command output signal.

Its value is the value of rapid traverse rate (Rapid) parameter multiplied with a constant providing a regulation reserve of approximately 10 or 20 percent.

Position Calibration=1,1÷1,2*Rapid

N0506 Position Offset (A1...32, floating-point, axis)

In case of analog drives, in standstill, we have to write here the values (offset) read out from the lag (FolErr).

Position data, the number written on the parameter, based on parameter #0 IND is to be interpreted in inch or mm if #1 ROT=0.

If #1 ROT=1 the data is interpreted in degrees.

N0507 Position Drift Compensation (A1...32, floating-point, axis)
In case of analog drives it is the parameter serving for the automatic compensation of the drift.
Unit: sec
It is the time constant of the automatic drift compensation.

N0508 Time Constant1 (A1...32, floating-point, axis)
It is the time constant of the position command signal filter (T_1).
Unit: sec.

N0509 Time Constant2 (A1...32, floating-point, axis)
It is the time constant of the equalizer filter (T_2).
Unit: sec

N0510 Dead Time (A1...32, floating-point, axis)
The extent of the equalizer dead time (T_D).
Value range: 0...0.01.
Unit: sec.

N0511 Feedforward (A1...32, floating-point, axis)
The extent of the speed feedforward (A).
Unit: none
Value range: 0...1

N0512 Multiply (A1...32, integer, axis)
Multiplier of encoder pulses without a sign (M).
Allowable value range: 1≤Multiply≤4.294.967.296

N0513 Divide (A1...32, integer, axis)
Divisor of encoder impulses without a sign (D).
Allowable value range: 1≤Divide≤4.294.967.296

The Multiply and Divide parameter shall be set in a way to ensure that on the axis
for a movement of 1 mm, or 1 degree10^6
for a movement of 1 inch, 10^7
pulses arrive from the encoder.

Example 1: resolution of the rotary encoder
2^{37} pulses.
From this
2^{25} pulses belong to 1 revolution of the encoder and it is able to register
2^{12} encoder revolutions.
The slide moves 10 mm per 1 revolution of the encoder. In such cases

$$10 \times 10^6 = \frac{M}{D} \times 2^{25}$$

$$\frac{M}{D} = \frac{10^7}{2^{25}} = \frac{2^7 \times 5^7}{2^{25}} = \frac{5^7}{2^{18}} = \frac{78125}{262144}$$

Thus the Multiply parameter value is 78125, and the Divide value is 262144.

Example 2: resolution of the linear scale is 10^{-5} mm.
Then the Multiply parameter value is 10, and the Divide Value is 1.

N0514 Servo Control (A1...32, bit, axis)

N0514 A1...32	#7	#6	#5	#4	#3	#2	#1	#0
	OPN	**END**	**FUP**	**EFD**		**AD**	**ID**	**MD**

#0 **MD**: A parameter serving for turning the revolution direction of the motor (D_M). It shall be set per axes in a way to ensure that a positive slide movement belongs to positive programmed movements.
0: there is no direction change on the command signal output,
1: it reverses the direction of the command signal output.

#1 **ID**: The direction-change parameter of incremental movements issued by the encoder (D_I). If there is a positive feedback in the position control loop, by redirecting this bit it can be terminated (as if an A-notA exchange was taking place on the encoder).
0: there is no direction turning,
1: it turns the direction.

#2 **AD**: In case of using an absolute or absolutely coded encoder, it is the parameter serving for turning the absolute direction (D_A). E.g.: in case of a distance coded scale the positive direction of the scale is opposite to the positive direction of the axis.

#4 **EFD**: The encoder feedback disable parameter bit is only effective in the state 1 of the#7 OPN parameter or in the state 1 of the AP_OPNR PLC flag.
0: if the parameter value is 0 and the value of the AP_EFD PLC flag is 0 too, ***the pulses of the encoder*** assigned to the axis ***will be counted*** and taken into account in the position of the axis in the case, when the position loop is open.
1: if the parameter value is 1 or the value of the AP_EFD PLC flag is 1, ***the pulses of the encoder*** assigned to the axis ***will not be counted*** and taken into account in the position of the axis in the case, when the position loop is open. It is used in the case, when several axes use the same encoder for position feedback.

#5 **FUP**: (Follow UP) In an emergency state (Machine Off) the ***position loop is not closed***, the encoder pulses are collected in the position register (ActPos). After switching on the machine (Machine On):
0: If the parameter value 0, **and** the AP_FLWU PLC flag value is 0, before closing the position control loop the ***command position*** (ComPos) takes up the value of the position register (ActPos, updated from the encoder) and ***no movement happens*** on the axis. The machine will remain in that position till the time we progam an absolute movement.

1: If the parameter value is 1, **or** the AP_FLWU PLC flag value is 1, before closing the position control loop the ***command position*** (ComPos) does no take up the value of the absolute position (ActPos measured from the encoder) and ***a movement happens*** on the axis, it makes the movement accumulated in the switched off state of the servo. This setting is highly advisable e.g. in case of a synchronized movement of axes!

#6 **END**: A parameter serving for switching out the encoder monitoring:

0: If the parameter value is 0 **and** the AP_END PLC flag is 0, the encoder monitoring is switched on,

1: If the parameter value is 1 **or** the AP_END PLC flag is 1, the encoder monitoring is switched off.

#7 **OPN**: A parameter serving for opening the position control loop:

0: If the parameter value is 0 **and** the AP_OPNR PLC flag is 0, the position loop is closed,

1: If the parameter value is 1 **or** the AP_OPNA PLC flag is 1, the position loop is open.

N0515 Feed Forward Control (A1...32, bit, axis)

N0515 A1...32	#7	#6	#5	#4	#3	#2	#1	#0
							FFR	**FFE**

#0 **FFE**: enable of feedforward:

0: the speed feedforward is not enabled

1: the speed feedforward is enabled.

#1 **FFR**: If the feedforward is enabled (FFE=1), in case of rapid traverse, based on FFR parameter

0: there is no feedforward in case of a rapid traverse,

1: the feedforward is effective in case of a rapid traverse.

N0516 Inpos (A1...32, floating-point, axis)

Position data, the number written on the parameter, based on parameter #0 IND shall be interpreted in inch or mm, if #1 ROT=0.

If #1 ROT=1 the data shall be interpreted in degrees.

It is the size of the “In Position” window (the difference between position measured on the encoder and the command position ComPos-ActPos).

In case an axis does not reach a position within a time defined on N1340 Inpos Timeout parameter, the control will indicate an error.

N0518 Time Constant3 (A1...32, floating-point, axis)

It is the time constant of the simulation necessary for creating a servo error, in seconds (sec).

From cycle to cycle, the position control loop compares whether the difference between the position measured from the encoder and the forecasted position calculated from the command position of the interpolator is higher in absolute value or not than the Serrl1, or Serrl2 parameter value. It is the constant value necessary for the calculation.

Recommended value (if the position loop gain is increased up to the oscillation limit):

$$T_3 \approx \frac{1}{4K}$$

where K is the loop gain value.

N0519 Serrl1 (A1...32, floating-point, axis)

It is the servo error limit in when the axis is in standstill.
From cycle to cycle, the position control loop compares whether the difference between the position measured from the encoder and the forecasted position calculated from the command position of the interpolator is higher in absolute value or not than the Serrl1 parameter value.
Position data, the number written on the parameter, based on parameter #0 IND shall be interpreted in inch or mm, if #1 ROT=0.
If #1 ROT=1 the data shall be interpreted in degrees.

N0520 Serrl2 (A1...32, floating-point, axis)

It is the dynamic servo error limit.
From cycle to cycle, the position control loop compares whether the difference between the position measured from the encoder and the forecasted position calculated from the command position of the interpolator is higher in absolute value or not than the Serrl2 parameter value.
Position data, the number written on the parameter, based on parameter #0 IND shall be interpreted in inch or mm, if #1 ROT=0.
If #1 ROT=1 the data shall be interpreted in degrees.

N0521 Serrl3 (A1...32, floating-point, axis)

It is the absolute servo error limit. A lag (FolErr) value belonging to rapid traverse.
Position data, the number written on the parameter, based on parameter #0 IND shall be interpreted in inch or mm, if #1 ROT=0.
If #1 ROT=1 the data shall be interpreted in degrees.

N0522 Synchr K (A1...32, floating-point, axis)

If an axis has been synchronized to an other one, it is the gain factor controlling the relation of the two axis positions. It refers always to the slave axis. It is a number without unit.

N0523 Backlash (A1...32, floating-point, axis)

Distance data, the number written on the parameter, based on parameter #0 IND shall be interpreted in inch or mm, if #1 ROT=0.
If #1 ROT=1 the data shall be interpreted in degrees.
It is the extent of backlash compensation per axis.

N0524 EnDat Resolution (A1...32, integer, axis)

It is the bit number to be taken into consideration from the resolution of the EnDat encoder, on the given axis.
Be it for example the rotary EnDat encoder a 37-bit one. From this 25 bit falls on one revolution of the encoder and it counts the revolutions made by the encoder on 12 bits.
In case of a linear axis, let's write 37 on the parameter.

In case of a rotary axis, if the axis turns one per one revolution of the encoder, let's write 25 on the parameter as data above 25 bit are unnecessary.

N0525 Compensation Control (A1...32, bit, axis)

N0515 A1...32	#7	#6	#5	#4	#3	#2	#1	#0
					FSE	**FRE**	**SME**	**BLE**

#0 **BLE**: The backlash acceleration compensation can be applied in case of position feedbacks with totally closed loops, for example, if we use a linear scale on linear axes.
The backlash acceleration compensation is:
=0: is not allowed,
=1: is allowed.

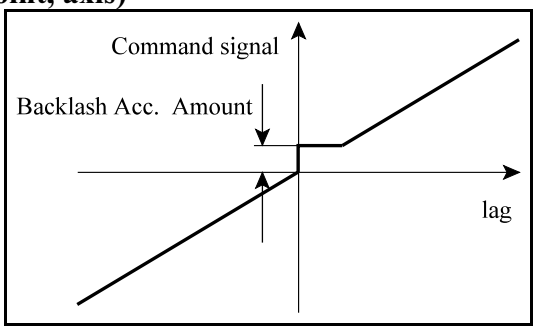
#1 **SME**: It is the parameter controlling the *stopping of the issuance of the backlash acceleration signal*. In case the backlash acceleration is allowed, and the parameter is:
=0: the compensation signal is issued
– it will terminate only after the expiry of time determined on N0527 Backlash Acc Period parameter,
=1: the stopping based on the distance done is allowed. The compensation signal is issued
– it will terminate either after the expiry of time determined on N0527 Backlash Acc Period parameter,
– or after doing the distance determined on N0528 Backlash Acc Distance parameter.

#2 **FRE**: The compensation of the static friction is used on axes at which the static friction comes into effect in a standstill status, after the expiry of a certain period of time.
The compensation of the static friction:
=0: is not allowed,
=1: is allowed.

#3 **FSE**: The parameter controlling the *stopping of the issuance of signals overcoming static friction*. If the compensation of the static friction is allowed, and the parameter is:
=0: the compensation signal is issued
– it will terminate only after the level of command signal emerging from the lag reaches the value determined on N0529 Friction Comp. Amount parameter,
=1: the stopping based on the distance done is allowed. The compensational signal is issued.
– it will terminate either after the level of command signal emerging from the lag reaches the value determined on N0529 Friction Comp. Amount parameter,
– or after having done the distance determined on N0531 Friction Comp. Distance parameter.

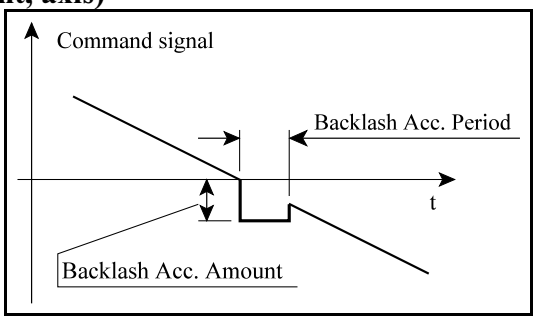
N0526 Backlash Acc. Amount (A1...32, floating-point, axis)

The amount of backlash acceleration, i.e. it is the amount of speed command signal during the start-up of the function, on the given axis, independently of the lag value.



N0527 Backlash Acc. Period (A1...32, floating-point, axis)

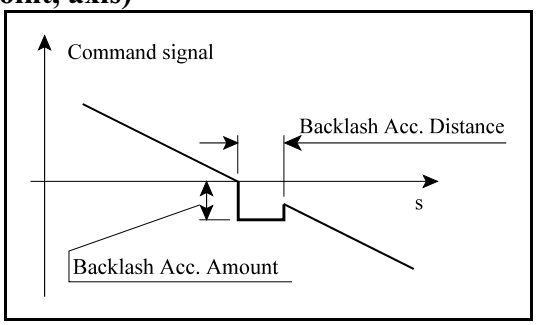
The period expressed in second unit (sec) during which the backlash acceleration is effective, on the given axis.



N0528 Backlash Acc. Distance (A1...32, floating-point, axis)

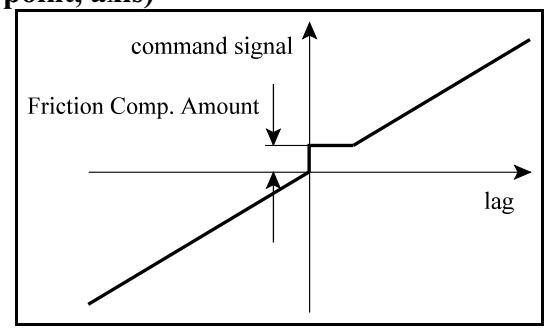
Position data, the number written on the parameter, based on parameter #0 IND shall be interpreted in inch or mm, if #1 ROT=0. If #1 ROT=1 the data shall be interpreted in degrees.

If during the backlash acceleration the ending based on the distance done is allowed with the parameter state N0525 Compensation Control parameter #1 SME=1, then after doing the distance defined here it will stop issuing the compensation signal.



N0529 Friction Comp. Amount (A1...32, floating-point, axis)

It is the amount of static friction compensation, i.e. it is the level of speed command signal during the start-up of the function, on the given axis, independently of the lag value.

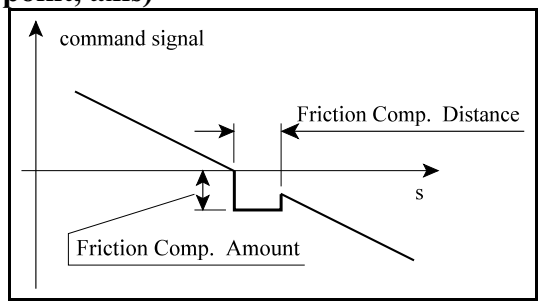


N0530 Stop Judgement Period (A1...32, floating-point, axis)

For the start-up of the compensation of the static friction it has to be judged whether the given axis has been in a standstill status or not and whether there has been sufficient time elapsed or not for the static friction becoming effective. If the time determined in seconds (sec) on the parameter elapses without a movement on the given axis, during the time of starting the axis the compensation of static friction will be activated.

☞ ***Warning!*** *If we set the parameter value to a too low level, in case of low feedrates the compensation may become active which may cause an unwanted effect.*

N0531 Friction Comp. Distance (A1...32, floating-point, axis)



Position data, the number written on the parameter, based on parameter #0 IND shall be interpreted in inch or mm if #1 ROT=0. If #1 ROT=1 the data will be interpreted in degrees.

If in case of the static friction compensation an ending based on the distance done is allowed, with a parameter state N0525 Compensation Control parameter #3 FSE=1, then after doing the distance defined here it will stop issuing the compensation signal.

N0532 Compensation A/x Multiplier (A1...32, floating-point, axis)

If the parameter value is different from 0, the ***switch-off of compensation signals*** will not happen immediately, but according to an A/x function where "A" is the parameter value.

N0533 Tandem Master (A1...32, integer, axis)

We call the case tandem control, if the same axis is moved by two drives and motors, but the axis has just one position encoder.

In this case the position control loop is closed on the master axis, but the speed command signal is to be output to the slave drive too.

Therefore an axis, the tandem slave, is to be assigned among the parameters. You must specify on parameters Axis Output Type and Axis Output Address which drive the tandem slave should output speed command signal to.

The Tandem Master parameter must be specified on the slave axis. The axis number of the master axis is to be written on this parameter. Command signal is only output to the slave axis, if on N0514 Servo Control parameter bit 7 OPN=1 is, that is the position control loop is open.

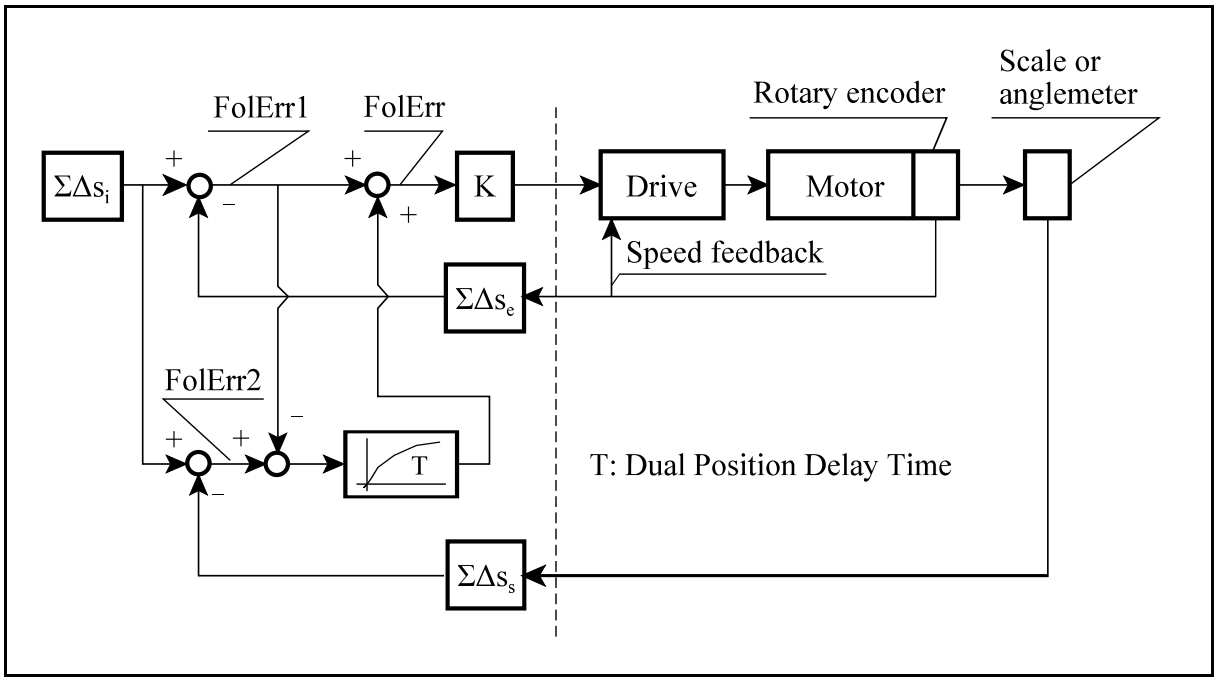
The tandem control, that is the synchronous turning of motors and the elimination of the backlash, is done by the drives.

Dual position control loop

If the position of a linear axis is measured by a scale or the position of a rotary axis is measured by an angle encoder (***direct measurement***), and the position control is carried out on these measuring devices, fluctuations may develop when axes stand because of the backlash between the motor and the measuring device.

If, on the motor, there is an encoder too ***(indirect measurement***), dual position control loop may also be set. ***By the use of dual position control loop, such a stability can be reached in position***

control as in the case of indirect measurement, in other words, control is executed on the encoder of the motor, at the same time accuracy derived from the direct measurement can be maintained.



Setting the position control loop is executed by the use of the indirect measuring device, i.e. the motor encoder. After setting, the direct measuring device (scale, anglemeter) has to be connected into the position control. The flowchart of the control is illustrated in the figure above.
When double loop is used, the value of the lag FolErr will be:

$$FolErr = FolErr1 + (FolErr2 - FolErr1)(1 - e^{-\frac{t}{T}})$$

The time constant T of delay can be specified in the Dual PositionDelay Time parameter. It follows from the equation above, that

if T= 0: FolErr=FolErr2, i.e. position control will be executed according to the direct measuring device,

if T= ∞: FolErr=FolErr1, i.e. position control will be executed according to the indirect measuring device.

N0534 Dual Position Axis Number (A1...32, integer, axis)

To the address of the axis designated for indirect measurement and position control, the number of that axis has to be written which receives the direct measuring device.

For example:

Let the axis X be the axis 1:

N0100 Axis Name1 A1=X.

Let the motor encoder of the axis 1 (X) be received by the input 1:

N0501 Axis Input Address A1=1.

Position control is executed according to the motor encoder. If the X drive is at the address 1:

N0503 Axis Output Address A1=1,

Let the signals of the scale be received by the axis 11. The axis 11 may be named too, for example XM:

N0100 Axis Name1 A11=X,

N0101 Axis Name2 A11=M.

The encoder of the scale X will be received by the input 6:

N0501 Axis Input Address A11=6.

Multiply/Divide parameters of the encoder of the axis 11 has to be set. No output belongs to the axis 11:

N0503 Axis Output Address A11=0.

After these,

N0534 Dual Position Axis Number A1=11,

i.e. the parameter indicates that the direct measuring device (the scale) belonging to the X axis 1is on the input of the XM 11.

☞ ***Warning!***

The axis receiving the direct measuring device (the master) has to be assigned as virtual one by parameter setting of #4 VIR=1 in the parameter N0106 Axis Properties!

N0535 Dual Position Delay Time (A1...32, floating-point, axis)

It is the time constant of the dual position control in second: sec.

It has to be written at the address of the axis designated for indirect measurement, i.e. position control.

N0536 Dual Position Difference Error (A1...32, floating-point, axis)

It will be compared by the position control loop cycle by cycle whether the difference between the positions measured by the direct and indirect measuring devices greater than the value of the Dual Position Difference Error parameter or not.

A position data, a number written in the parameter will have to be interpreted in inch or mm #0 IND if #1 ROT=0.

If #1 ROT=1, the data will be interpreted in degree.

N0537 Dual Position Maximum Value (A1...32, floating-point, axis)

It is the maximum value of the signal

$$(FolErr2 - FolErr1)(1 - e^{-\frac{t}{T}})$$

correcting the FolErr1signal. If the signal increased over this value, the correction degree would be the value of the Dual Position Maximum Value parameter. If the parameter is 0, there will not be high cut-off in the correcting signal.
Based on the #0 IND parameter , a position data, a number written in the parameter will have to be interpreted in inch or mm if #1 ROT=0.
If #1 ROT=1, the data will be interpreted in degree.

N0538 Dual Position Minimum Value (A1...32, floating-point, axis)

If the absolute value of the signal difference FolErr2–FolErr1is smaller than the value of the Dual Position Minimum Value parameter, the value of the signal

$$(FolErr2 - FolErr1)(1 - e^{-\frac{t}{T}})$$

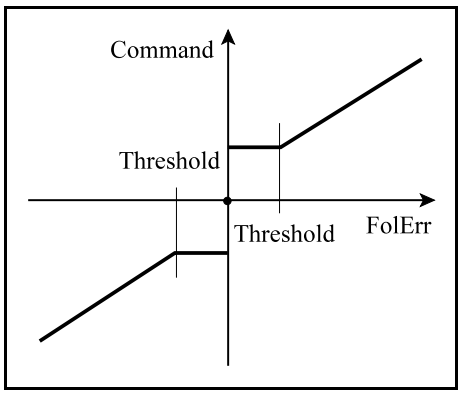
correcting the FolErr1signal will be 0.
Based on the #0 IND parameter , a position data, a number written in the parameter will have to be interpreted in inch or mm if #1 ROT=0.
If #1 ROT=1, the data will be interpreted in degree.

N0539 FeedForward2 (A1...32, floating-point, axis)

It is not used.

N0540 Threshold (A1...32, floating-point, axis)

It is the limit of the nonlinearity range of the reference output signals.
If the lag is zero, the output signal will be 0.
If the absolute value of the lag is greater than the value of the Threshold parameter, the output signal will be calculated according to the linear characteristic.
If the absolute value of the lag is smaller than the value of the Threshold parameter, the output signal will be equal to the value belonging to lag of Threshold value.



8 Spindle Control Parameter Group

N0600 Spindle Input Type (S1...16, integer, spindle)

The hardware type belonging to the **encoder input** of the given spindle. The selection is made from a drop-down menu.

N0601 Spindle Input Address (S1...16, integer, spindle)

The periphery address of the unit (NCT drive, measuring system card) belonging to the **encoder input** of the given spindle.

☞ ***Warning!*** *The selected address is not in connection with the number determined on the given spindle parameter. E.g.: on the parameter S2 is spindle 2, but the address of its encoder may be e.g. 5.*

If the parameter value**=0**: **there is no position measurement** on the given spindle.
If the parameter value**>0**, **there is position measurement** on the given spindle.
The output and input address belonging to the given spindle may be the same but also different!

N0602 Spindle Output Type (S1...16, integer, spindle)

The hardware type belonging to the output of the given spindle. The selection is made from a drop-down menu.

N0603 Spindle Output Address (S1...16, integer, spindle)

The periphery address of the unit (NCT drive, D-A converter card etc.) belonging to the output of the given spindle, to which the revolution reference signal, respectively, the PLC control bits (e.g. drive enable, etc.) shall be issued..

☞ ***Warning!*** *The selected address is not in connection with the number determined on the given spindle parameter. E.g.: on the parameter the S2 is spindle 2, but the address of its output can be e.g. 7.*

If the parameter value**=0**: **there is no issuance of command signals** on the given spindle.
If the parameter value**>0**, **there is issuance of command signals** on the given spindle.
The same command signal address may be assigned to a spindles or to an axis too:

– In case a spindle drive participates in polar interpolation as axis C, the same address shall be specified both to axis C and the spindle drive. The AP_DETCHR and SP_SDETCHR PLC flags will decide, the output of axis C or spindle S will be issued to the drive.

☞ ***Warning!*** *The output and input address belonging to the given spindle may be the same but also different! For example, spindle S1 spindle gives reference signals to drive with address 4, but it takes the signal of the thread cutting encoder from periphery with address 11.*

N0604 Default Spindle (L1...8, integer, channel)

To each channel a spindle can be assigned specified with its number, which will be the default spindle of the given channel. To this spindle it is possible to refer from the part program with a one-character S address, without specifying any P address, or if we have given the spindle a multi-character name. In case of reference to address S the content of this parameter will be moved to CN_SSEL spindle selection PLC register.

N0605 Spindle Name2 (S1...S16, integer, spindle)

It is mandatory that the names of all spindles begin with character S. This parameter tells what should be the second character of the name of the n-th spindle. Possible names: numbers 0, 1, ..., 9 (belonging values: 48, 49, ..., 57, letters of the English alphabet: A, B, C, ..., Z (belonging values: 65, 66, 67, ..., 90).

If the parameter value: 0 there is no second and third spindle name.

N0606 Spindle Name3 (S1...S16, integer, spindle)

It is mandatory that the names of all spindles begin with character S. This parameter tells what should be the third character of the name of the n-th spindle. Possible names: numbers 0, 1, ..., 9 (belonging values: 48, 49, ..., 57, letters of the English alphabet: A, B, C, ..., Z (belonging values: 65, 66, 67, ..., 90).

If the parameter value: 0 there is no third spindle name.

☞ **Warning!** *The extended addresses cannot be used for passing arguments during the macro programming.*

☞ **Warning!** *If the spindle address ends with a number, e.g. S2, in a part program a = sign shall be programmed after it: S2=3200.*

N0607 Spindle Config (S1...S16, bit, spindle)

N0607 S1...16	#7	#6	#5	#4	#3	#2	#1	#0
	IDS	**EAB**	**TOE**	**ZOR**	**IXC**	**INX**	**ORI**	**SEX**

#0 **SEX**: If the parameter

0: the given spindle does not exist,

1: the given spindle does exist. If the given spindle does exist, at the PLC flag state SN_SDETCHA=0 the spindle handler will operate, and at the PLC flag state SN_SDETCHA=1 the spindle handler will not operate.

#1 **ORI**: If the parameter

0: the given spindle cannot be oriented,

1: the given spindle is oriented by M19.

The orientation may be carried out mechanically, by running to a switch and by the closure of the position control loop. The parameter shall be written into 1 in all cases, as this parameter enables those drilling cycles (G76, G87), which require orientation.

#2 **INX**: If the parameter

0: the given spindle cannot be indexed,

1: the given spindle can be indexed, i.e. a position control loop can be closed.

#3 **IXC**: If the parameter

Not used.

#4 **ZOR**: If the parameter
0: after power on, the first orientation (M19) happens by searching for the zero pulse, and the other happen by positioning to the zero pulse,
1: every orientation happens by searching for the zero pulse.
It makes sense only at 0 state of EAB parameter.
#5 **TOE**: If the parameter
0: during thread cutting the spindle override is ineffective, the value is clamped to 100%,
1: during thread cutting the spindle override is effective.
#6 **EAB**: If the parameter
0: an incremental encoder is mounted onto the spindle
1: an absolute encoder is mounted onto the spindle.
#7 **IDS**: If the parameter
0: the spindle indexing by function M is made in a positive direction
1: the spindle indexing by function M is made in a negative direction

N0608 Spindle Encoder Counts (S1...S16, integer, spindle)
It is the pulse count of the spindle encoder. On the parameter,
– in case of an incremental encoder (EAB=0) the fourfold of the value (4*data) indicated on the data sheet shall be written here,
– in case of an absolute encoder (EAB=1) the value indicated on the data sheet.
In case there is no encoder mounted on the spindle, the value is 0.

N0609 Spindle Encoder Config (S1...16, bit, spindle)

N0609 A1...32	#7	#6	#5	#4	#3	#2	#1	#0
		END	**UNI**	**OSW**	**OTR**	**AD**	**ID**	**MD**

#0 **MD**: It is the parameter serving for reversing the revolution direction of the spindle.
It has to be set in a way that upon M3 command and PLC flag state SP_NEG=0 to belong a revolution direction according to the right hand rule.
In the indexed state (position loop closed) of spindle, a revolution direction according to the right hand rule belongs to the positive programmed movement.
0: there is no direction change on the outgoing command signal,
1: it turns the direction of the outgoing command signal.
#1 **ID**: It is the parameter serving for turning the direction of incremental movements issued by the encoder. In case in the position control loop there is a positive feedback, by over-writing this bit it can be terminated (as if we carried out an A-notA exchange on the encoder).
0: there is no direction change,
1: it turns the direction.
#2 **AD**: In case of using an absolute, or distance coded encoder, it is the parameter serving for turning the absolute direction.

A spindle may be orientated in the following three ways:
– ***Orientation based on the zero pulse of the spindle encoder (basic case, TR=OSW=0)*** In this case, due to the NC PLC flags , the control will search for the zero pulse of the spindle encoder and will be joined with it. This is the most accurate and the best solution.

If the encoder is mounted not on the spindle but on the motor, and there is a transmission or a speed-change gear between the spindle and the motor, the position of the zero pulse relative to the spindle may change depending on the motor rotation. In this case, it is necessary to mount a proximity switch on the the spindle, and orientation will have to be executed ***on the basis of the signal of the switch***. ***Orientation accuracy*** is determined by the ***switching speed, the sampling time*** and the ***spindle speed***. In these cases, the spindle handling device will search for the signal of the switch in the case only, when the spindle has taken the value set in the N080n Rn S OrientSpeed parameter. ***The signal of the switch can be received in*** the following ***two ways*** that can be selected in parameters:

– ***Orientation based on the probe input (OTR=1, OSW=0)***
– ***Orientation based on the interface input (OTR=0, OSW=1)***

#3 **OTR**: If orientation is carried out on the basis of the probe input, this bit will have to be set into 1. If the ith probe input is to be assigned to the jth spindle, the address **Sj_OrientPoi** will have to be selected to the row ***Probe I*** in the EtherCAT configuration window.

☞ ***Warning!***

– *The probe card can detect the change of the inductive signal up to the speed of 700 rev/min.*
– *When the speed is the same, the position will be found with the deviation of approx. 0.1°.*

#4 **OSW**: If orientation is carried out on the basis of the interface input, this bit will have to be set into 1. Because of the sampling frequency, the signal of the switch has to be received in the fast module Int0 of the PLC program. The state of the switch has to be copied on the **SP_OSW** flag in the PLC program. Orientation will be started by the spindle handling device when the state of the SP_OSW flag is 1.
Accuracy of this solution is occasional, it is to be measured.

#5 **UNI**: If the parameter value
0: the spindle is bipolar, i.e. the revolution direction is determined by the sign of command signal. The spindle speed measured from the encoder, and the monitoring signals (N-Ns, etc.) are calculated by the spindle handler with a sign.
1: the spindle is unipolar, i.e. the issued command signal does not have a sign, and the revolution direction is set by the PLC. Neither the spindle speed measured from the encoder, nor the monitoring signals (N-Ns, etc.) are calculated by the spindle handler with a sign.

#6 **END**: It is the parameter serving for switching off the spindle encoder monitoring:
0: If the parameter value is 0 **and** the SP_SEND PLC flag is 0, the encoder monitoring is on,
1: If the parameter value 1 **or** the SP_SEND PLC flag is 1, the encoder monitoring is switched off.

N0610 Spindle N Time Constant (S1...S16, floating-point, spindle)
Measurement unit: sec
It is the time constant of the filter of spindle speed measured from the encoder.

N0610+n Rn S Multipy n=1...8 (S1...S16, integer, spindle)
It is the multiplier of the spindle encoder pulses without a sign (M).
n=1...8: for each element of the range 1...8 a separate parameter value can be set.

Specifiable value range: 1≤Multiply≤4.294.967.296

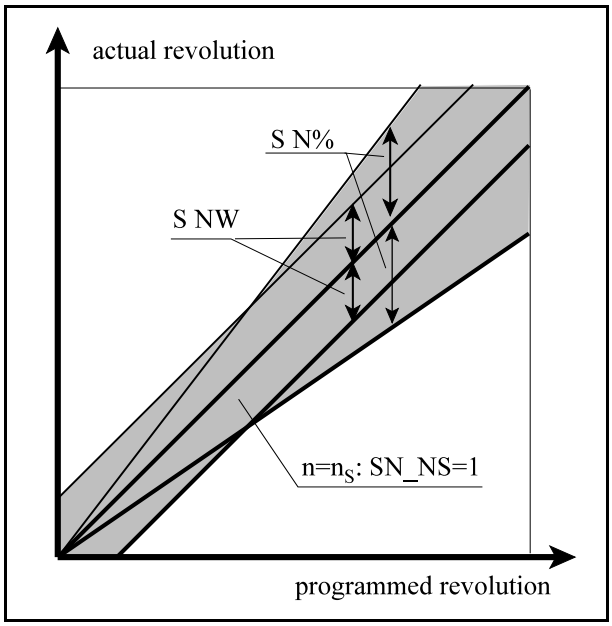
N0618+n Rn S Divide n=1...8 (S1...S16, integer, spindle)

It is the divisor of the spindle encoder pulses without a sign (D).
n=1...8: for each element of the range 1...8 a separate parameter value can be set.
Specifiable value range: 1≤Divide≤4.294.967.296

The above two parameters shall be used if the encoder is not mounted directly to the spindle, but, for example, to the spindle motor. Then in the various ranges the multiplication/division parameter shall be set in a way to ensure that the pulse number per spindle is the same with the value set on the N0608 Spindle Encoder Counts parameter, in each range.

N0627 S N% (S1...S16, integer, spindle)

The number limiting the percentage difference of the actual spindle speed and the programmed spindle speed (S) of the given spindle for issuing the $n=n_S$ (the spindle has taken up the revolution) signal.
Measurement unit: %.
If the difference between the programmed (S) and the actual spindle speed gets within the range between parameters S N% **or** S NW, the control will set the SN_NS ($n=n_S$) PLC flag.



N0628 S NW (S1...S16, integer, spindle)

The number limiting the absolute difference of the programmed spindle speed (S) and the actual one for issuing the $n=n_S$ (the spindle has taken up the revolution) signal.
Measurement unit: rev/min.
If the difference between the programmed (S) and the actual spindle speed gets within the range specified by parameters S N% **or** S NW, the control will set the SN_NS ($n=n_S$) PLC flag.

N0629 S N0 (S1...S16, integer, spindle)

The spindle speed serving for the issuance of n=0 (spindle is in standstill) signal.
Measurement unit: rev/min.
If the spindle speed value gets within the range determined by S N0 parameter, the control issues the n=0 signal, i.e. it sets the SN_N0 PLC flag. Simultaneously with this, it sets the SN_NS PLC flag (n=nS the spindle has taken up the revolution).

☞ ***Notes:*** *Functions $n=n_S$ and n=0 are effective only if an encoder is mounted to the spindle (Spindle Encoder Counts>0 parameter state).*

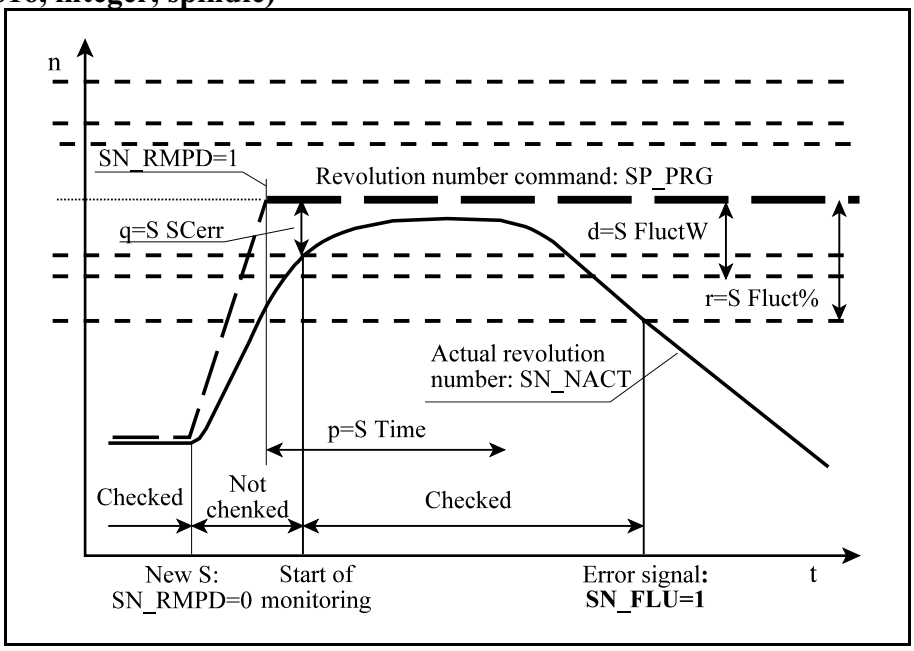
The spindle speed command, compared to which the given spindle speed is monitored, becomes calculated by taking into consideration the override, the spindle speed limits of ranges and in a G96 constant surface speed calculation state also by taking into consideration the programmed maximum spindle speed (G92 S_).
Both the $n=n_S$ and n=0 (S NS and S N0) signals together are 1 in the standstill state of the spindle.

N0630 S Time (S1...S16, floating-point, spindle)

The NC starts the monitoring of the spindle fluctuation, after the time, set here, elapsed.
Measurement unit: sec.
If the difference between the spindle speed commanded and the actual is higher than the difference specified on S Scerr parameter the monitoring of fluctuation will start **after** the time set on S Time has elapsed.

.

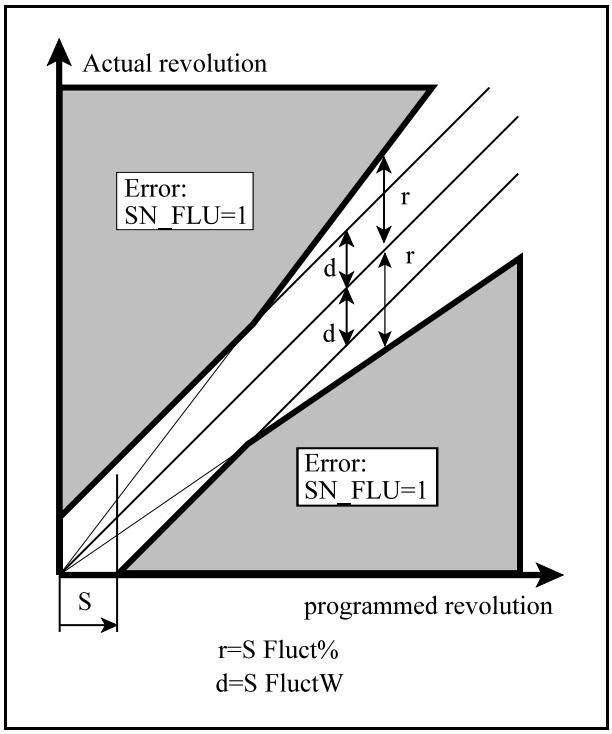
N0631 S SCErr (S1...S16, integer, spindle)



It is the parameter starting monitoring of spindle fluctuation upon a percentage difference. Measurement unit: %.
If the difference between the commanded and the actual revolution number gets smaller than the difference determined on S Scerr parameter **before** the time set on S Time parameter elapses, the monitoring of fluctuation will start.

N0632 S Fluct% (S1...S16, integer, spindle)

The parameter specifies the relative amount of spindle fluctuation. Exceeding the value set here an error signal emerges.
Measurement unit: %.
It is the allowable level of revolution number fluctuation in the percentage of the issued revolution number.
In case after the start of fluctuation monitoring the given revolution number of the spindle falls out of both revolution number ranges determined by the parameters S Fluct% and S FluctW, the control will set the PLC flag SN_FLU.



N0633 S FluctW (S1...S16, integer, spindle)

It is the parameter determining the absolute amount of the spindle fluctuation. Exceeding the value set here an error signal emerges.
Measurement unit: rev/min.
It is the allowable level of revolution number fluctuation in absolute value.
In case after the start of fluctuation monitoring the given revolution number of the spindle falls out of both revolution number ranges determined by the parameters S Fluct% and S FluctW, the control will set the PLC flag SN_FLU.

N0633+n Rn S Calibration n=1...8 (S1...S16, floating-point, spindle)

n=1...8: different parameter for every item of the range 1...8
It is the calibration of the spindle output. It is the spindle speed belonging to 2^{31}, or 10V in the eight ranges of the 1...16th spindles.
Measurement unit: rev/min.
In the different ranges that spindle speed shall be given on parameters in rev/min unit by which the spindle would turn if on the analog output the maximum 10V or on digital output 2^{31} was given out.

N0641+n Rn S Min n=1...8 (S1...S16, integer, spindle)

n=1...8: different parameter for every item of the range 1...8
It is the minimum spindle speed in the eight ranges of the 1...16th spindles.
Measurement unit: rev/min.
If in the given range a revolution smaller than this is programmed, the spindle is going to take up this revolution.

N0649+n Rn S Max n=1...8 (S1...S16, integer, spindle)

n=1...8: different parameter for every item of the range 1...8
It is the maximum spindle speed in the eight ranges of the 1...16 spindles.
Measurement unit: rev/min.
The parameter value cannot be higher than that specified on Rn S Calibration parameter.
If in the given range a speed higher than this is programmed, the spindle is going to take up this speed.

N0657+n Rn S Jog Speed n=1...8 (S1...S16, integer, spindle)

n=1...8: different parameter for every item of the range 1...8
A spindle speed with a parameter in in the eight ranges of the 1...16. spindles.
Measurement unit: rev/min.
In case the **SP_PAR** PLC flag is set, the spindle speed set on the parameter will be issued. It can be used at gear range changes or for jogging the spindle. This speed can be lower than the speed minimum valid for the range:

Rn S Jog Speed<Rn S Min

N0665+n Rn S Ramp Up n=1...8 (S1...S16, integer, spindle)

n=1...8: different parameter for every item of the range 1...8
It is the ramping up parameter of the command signal in the n-th gear range of the 1...16. spindles.
Ramping up: when the spindle speed (energy take-up) is increasing in an absolute value.
Its meaning: the spindle speed command signal in absolute value shall be *increased* specified by the parameter, per second.
Measurement unit: rev/min/sec
If the parameter value is 0, there is no ramping up, and the command signal becomes issued immediately.

N0673+n Rn S Ramp Down n=1...8 (S1...S16, integer, spindle)

n=1...8: different parameter for every item of the range 1...8

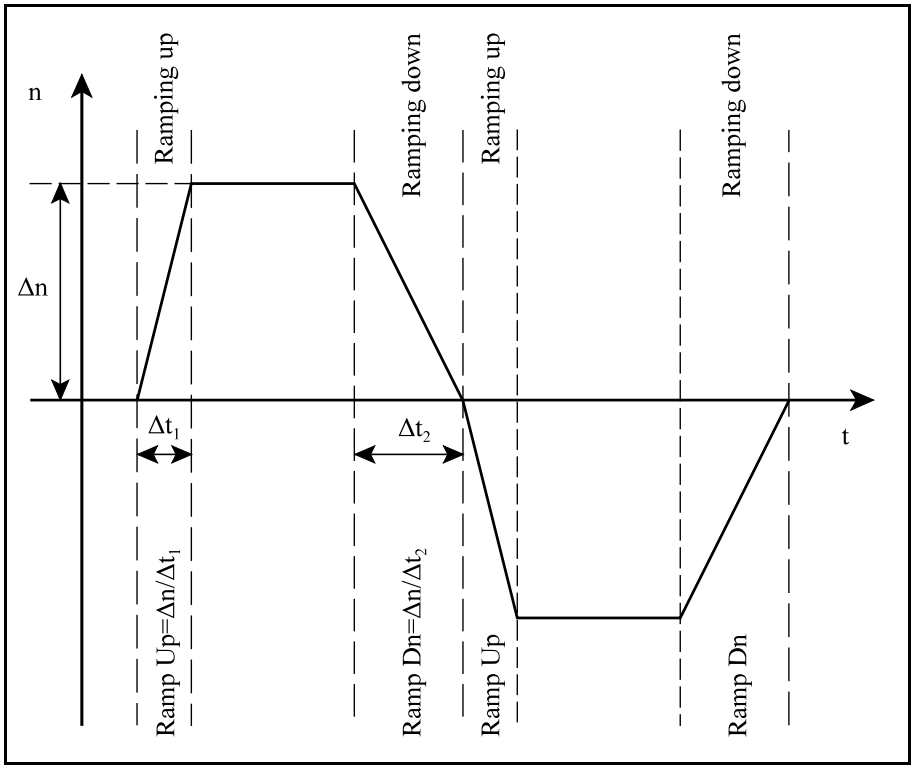
It is the ramping down parameter of the command signal of the n-th spindle speed range of the 1...16. spindles.

Ramping down: when the spindle speed decreases in absolute value (energy dissipation).

Its meaning: the spindle speed command signal in absolute value shall be *decreased* specified by the parameter, per second.

Measurement unit: rev/min/sec

If the parameter value is 0, there is no ramping down, and the command signal becomes issued immediately.



N0682 Spindle Analog Adjust (S1...S16, floating-point, spindle)

In case of an analog output it is the parameter serving for the calibration of the D-A converter.

Its setting: let's write a zero to the parameter, let's issue in the given range the spindle speed belonging to the 10V command signal, and then let's measure the value of the spindle command signal. Let's write the value of the voltage measured this way in V to this parameter. This parameter shall be higher than or equal to 10.

N0683 Spindle Speed Offset (S1...S16, floating-point, spindle)

It is the value compensating the offset of the spindle command signal.

Unit: V.

N0684 Spindle Grid Shift (S1...S16, integer, spindle)

Measurement unit: encoder pulse

During an orientation made by the closure of position loop:

in case of an incremental encoder it moves from the zero pulse this number of endocer pulses with a sign, and this point is regarded as the M19 position,

in case of an absolute encoder it takes up this position upon the command M19.

A number with a sign.

There is no point in determining this parameter per ranges because if the encoder is not mounted to the spindle on a 1:1 basis, no absolute position can be got. It may be oriented but the position of the zero pulse is uncertain.

N0685 Spindle Phase Shift (S1...S16, integer, spindle)

A number with a sign.

Measurement unit: encoder pulse.

In a synchronized state the zero pulse of the slave spindle is moved by this number of pulses compared to the zero pulse of the master. The shift takes into consideration the resolution of the encoder of the slave spindle!

A number with a sign.

N0686 Default Surf Speed (L1...8, integer, channel)

Speed data, the number written on the parameter, based on parameter #0 IND shall be interpreted in feet/min or m/min.

To every channel a default surface speed may be selected, which can be determined on this parameter. In case after power on, upon a G96 command we do not determine a value for address S, it will calculate the cutting speed based on this parameter.

N0687 Default G96 Axis (L1...8, integer, channel)

The number of the axis shall be written on this parameter for which in the given channel the constant suface speed shall be calculated.

N0688 Min Spindle Speed G96 (L1...8, integer, channel)

Spindle speed data, Measurement unit: rev/min.

A minimum spindle speed may be appointed in every channel, below which the NC does not allow the spindle in G96 constant surface speed calculation state.

N0689 Spindle M Low (S1...S16, integer, spindle)

N0690 Spindle M High (S1...S16, integer, spindle)

Standard M codes relevant to the operation of spindles:

M3: spindle on CW,

M4: spindle on CCW,

M5: spindle stop,

M19: orientation

From among these codes the active one is written by the PLC program to the SP_ROT PLC register. E.g.: the meaning of SP_ROT=3 is that the spindle is rotating clockwise. These are M codes and spindle states excluding each other.

If the PLC program uses also M codes different from the above ones, and these M codes are excluding the above states, and written into the SP_ROT register, those shall be

written to this pair of parameters. We write the smallest value of the M code block to the Spindle M Low parameter and the highest value of the block to the Spindle M high parameter. For example, if apart from the above, we use for spindle S1 also the following M codes:

M20: transformation of the spindle into axis C
M21: synchronization of the spindle to a master
M22: synchronization of the spindle to a master by phase shift
M23: preparation of the spindle for polygonal turning
M24: spindle position loop closure without orientation (M Code for Closing S Loop)

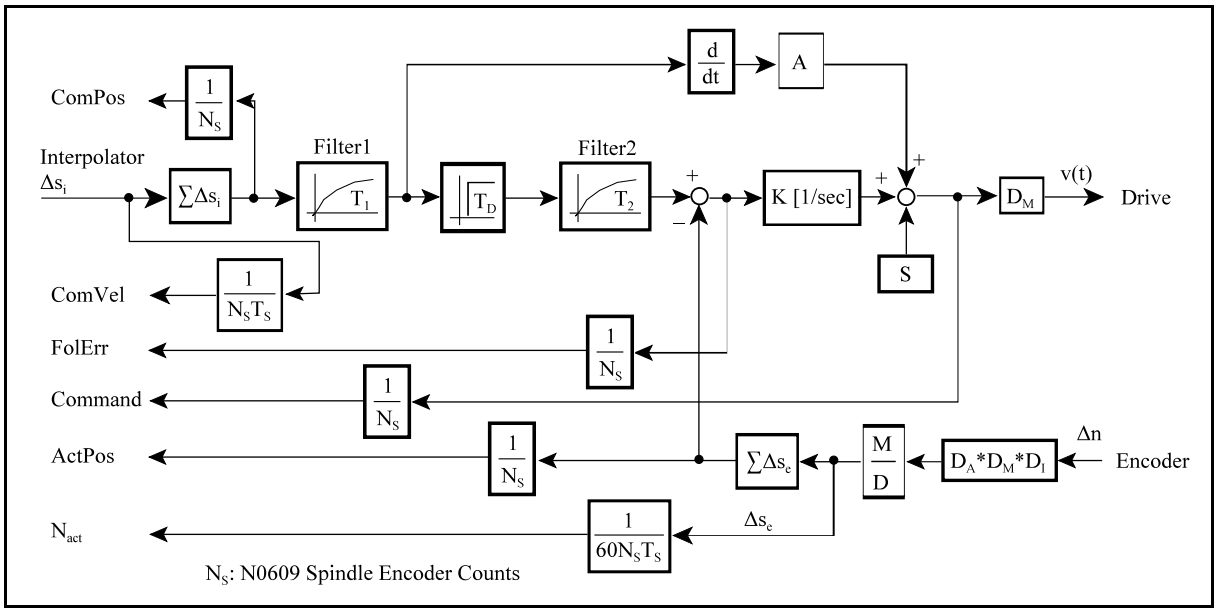
the Spindle M Low S1=20, the Spindle M High S1=24.
During searching for a block, besides the collection of M3, M4, M5, M19 codes it will collect the above codes, too and it will enter only the last one into codes to be executed.

N0691 EnDat Resolution (S1...S16, integer, spindle)

It is the bit number per one revolution, from the resolution of EnDat encoder, per spindle. Let's say that the rotary EnDat encoder a 37-bit one. From this 25 bit is covered per a rotation of the encoder and it counts the revolutions made by the encoder on 12 bits.
The bit number per one revolution of the encoder is 25, therefore the parameter shall be 25.
If between the encoder and the spindle the gear-ratio is not 1:1, the gear-ratio shall be taken into consideration on parameters Rn S Multiply, Rn S Divide.
Value range: 0...64.

9 Spindle Servo Parameter Group

On the figure below we can see the diagram of the position control loop. The marks (K, A, etc.) on the diagram are in accordance with parameters. Signals displayable on the scope or measuring system test screen are indicated on it, too (ComPos, FolErr, etc). Measurement units of signals are: rev, rev/sec, rev/min.



N0700+n Rn Spindle Gain n=1...8 (S1...S16, floating-point, spindle)

n=1...8: different parameters for every element of the range 1...8

Measurement unit: 1/sec.

It is the spindle position control loop gain (K), per range, in a closed state of the position control loop.

Between the loop gain (K) and the lag of the position control loop (FolErr), depending on the revolution number [n]=rev/min the equation is the following:

$$FolErr = \frac{n}{60K}$$

If the above equation is not fulfilled, the Rn S Calibration parameter, respectively, the reference input gain of the drive shall be set.

N0709 S Position Drift Compensation (S1...S16, floating-point, spindle)

In case of analog drives it is the parameter serving for the automatic compensation of the drift.

Unit: sec

It is the time constant of the automatic elimination of the drift.

N0709+n Rn S Time Constant1 n=1...8 (S1...S16, floating-point, spindle)

n=1...8: different parameters for every element of the range 1...8

It is the time constant of the spindle position command signal filter, per range (T_1).

Unit: sec

N0717+n Rn S Time Constant2 n=1...8 (S1...S16, floating-point, spindle)

n=1...8: different parameters for every element of the range az 1...8
It is the time constant of the equalizer filter per range (T_2).
Unit: sec

N0725+n Rn S Dead Time n=1...8 (S1...S16, floating-point, spindle)

n=1...8: different parameters for every element of the range az 1...8
It is the extent of the equalizer dead time per range (T_D).
Value range: 0...0.01.
Unit: sec

N0733+n Rn S Feedforward n=1...8 (S1...S16, floating-point, spindle)

n=1...8: different parameters for every element of the range az 1...8
It is the level of speed feedforward per range (A).
Unit: none
Value range: 0...1

N0742 Spindle Servo Control (S1...16, bit, spindle)

N0706 A1...32	#7	#6	#5	#4	#3	#2	#1	#0
	OPN							

#7 **OPN**: It is the parameter serving for the opening of the position control loop:

0: If the parameter value is 0 the position loop is closed in the state 1 of the SP_OREQ PLC flag,

1: If the parameter value is 1, the position loop is not closed in the state 1 of the SP_OREQ PLC flag.

N0743 Spindle Feedforward Control (S1...16, bit, spindle)

N0707 A1...32	#7	#6	#5	#4	#3	#2	#1	#0
							FFR	**FFE**

#0 **FFE**: allowance of the speed feedforward:

0: there is no speed feedforward

1: speed feedforward enabled

#1 **FFR**: If the speed feedforward is enabled (FFE=1), in case of a rapid traverse, based on FFR parameter

0: the speed feedforward is cancelled in case of a rapid traverse

1: the speed feedforward is effective also in case of a rapid traverse.

N0743+n Rn S Inpos n=1...8 (S1...S16, floating-point, spindle)

n=1...8: different parameters for every element of the range 1...8.
It is the size of the "spindle in position" window in spindle revolutions.

N0752 S Inpos TimeOut (S1...S16, integer, spindle)

The limit of waiting for the signal “spindle in position” in seconds (sec).

N0752+n Rn S Time Constant3 n=1...8 (S1...S16, floating-point, spindle)

n=1...8: different parameters for every element of the range 1...8

It is the time constant of simulation necessary for the creation of the servo error, in seconds (sec).

The position control loop compares from cycle to cycle, whether the difference between the position measured from the encoder and the forecasted position calculated from the command position of the interpolator, in an absolute value, is higher than the Rn S Serrl1, or Rn S Serrl2 parameter value, or not. It is the time constant value necessary for the calculation.

Its recommended value (if the loop gain is increased till the oscillation limit):

$$T_3 \approx \frac{1}{4K}$$

where K is the value of loop gain.

N0760+n Rn S Serrl1 n=1...8 (S1...S16, floating-point, spindle)

n=1...8: different parameters for every element of the range 1...8

It is the limit of the servo error in a standstill state of the spindle, in spindle revolutions.

The position control loop compares from cycle to cycle, whether the difference between the position measured from the encoder and the forecasted position calculated from the command position of the interpolator, in an absolute value, is higher than the Rn S Serrl1 parameter value, or not.

N0768+n Rn S Serrl2 n=1...8 (S1...S16, floating-point, spindle)

n=1...8: different parameters for every element of the range 1...8

It is the limit of the dynamic servo error in spindle revolutions.

The position control loop compares from cycle to cycle, whether the difference between the position measured from the encoder and the forecasted position calculated from the command position of the interpolator, in an absolute value, is higher than the Rn S Serrl2 parameter value, or not.

N0776+n Rn S Serrl3 n=1...8 (S1...S16, floating-point, spindle)

n=1...8: different parameters for every element of the range 1...8.

It is the lag value belonging to the rapid traverse.

It is the limit of the absolute servo error in spindle revolutions.

N0784+n Rn S Synchr K n=1...8 (S1...S16, floating-point, spindle)

n=1...8: different parameters for every element of the range 1...8.

In case a spindle has synchronized to an other one, it is the gaining factor controlling the joint operation of the two spindle positions. It refers always to the slave spindle.

It is a number without unit.

N0792+n Rn S Time Constant4 n=1...8 (S1...S16, floating-point, spindle)

Not used.

N0800+n Rn S OrientSpeed n=1...8 (S1...S16, floating-point, spindle)

n=1...8: different parameters for every element of the range 1...8.

During the orientation of the spindle, it is the level of orientation speed in a unit of 1/min.

N0808+n Rn S BackLash n=1...8 (S1...S16, integer, spindle)

n=1...8: different parameters for every element of the range 1...8

It is the backlash error of the spindle, in encoder pulses. It is taken into consideration in case of a closed position control loop.

N0817 Spindle Axis Name1 (S1...S16, integer, spindle)

If we would like to index the spindle after the closure of the position control loop (M19) from the part program like an axis, according to the resolution of the spindle encoder, we have to specify the reference address of the spindle axis. The reference address of the spindle axis may be up to 3 characters long. Its **first character** shall be mandatorily **A, B, or C.**

It tells the first character of the name of the given spindle axis.

If the parameter value =0, it is a spindle axis without any address, which can be indexed from a part program just by M codes. (See parameters Start M of Spnd. Pos., No. of M Code for Spnd. Pos., Basic Angle of Spnd. Pos.) However, such kind of a spindle axis may take part in rigid tapping.

☞ ***Warning!***

A spindle axis may be only positioned (indexed), respectively, it may take part in rigid tapping. A spindle axis neither in polar interpolation command nor in other interpolation operation like a rotary axis, cannot take part.

N0818 Spindle Axis Name2 (S1...S16, integer, spindle)

It tells the second character of the name of the given spindle axis. Possible names: numbers 0, 1, ..., 9 (related values: 48, 49, ..., 57), letters of the English alphabet: A, B, C, ..., Z (related values: 65, 66, 67, ..., 90).

If the parameter value is 0, there is no second and further (third) spindle axis name.

N0819 Spindle Axis Name3 (S1...S16, integer, spindle)

It tells the third character of the name of the given spindle axis. Possible names: numbers 0, 1, ..., 9 (related values: 48, 49, ..., 57), letters of the English alphabet: A, B, C, ..., Z (related values: 65, 66, 67, ..., 90).

If the parameter value is 0, there is no third spindle axis name..

N0820 Start M of Spnd. Pos. (S1...S16, integer, spindle)

It specifies the starting M code of the M functions which index the spindle by the angle determined on Basic Angle of Spnd. Pos. parameter, incrementally, in the spindle position control mode. The direction of the movement is determined by the IDS bit of the Spindle Config. parameter.

If e.g. the parameter value 201 and the angle of indexing is 18°, upon the M201 command it steps incrementally 18° in a positive or negative direction, based on the IDS parameter bit.

If the parameter value is 0, it does not carry out any positioning.

N0821 No. of M Code for Spnd. Pos. (S1...S16, egész, spindle)

It tells the number of those M codes which are related to indexing the spindle, starting from the code specified on Start M of Spnd. Pos. parameter.

N0822 Basic Angle of Spnd. Pos. (S1...S16, floating-point, spindle)

Data of angle, it shall be interpreted in ° (degrees).
It specifies the angle if indexing that is how many degrees should be stepped by the M code determined on Start M of Spnd. Pos. parameter.
If the position of the spindle is in the Rn S Inpos-radius environment of an integer multiple of the Basic Angle of Spnd. Pos. value, the SN_SINDP PLC flag will be set. Then the given spindle may be clamped.
0° is the position of the zero pulse by taking into consideration the N0684 Spindle Grid Shift offset.
If the parameter value is 0, it does not carry out any indexing.

☞ **Explanation:**

We have a spindle, which can be clamped at 20 points (per every 18°). We are going to carry out the indexing for an M code.
Let's set the parameters in the following way:
Spindle Config:
#7 IDS=0: it indexes in a positive direction
Start M of Spnd. Pos.=201 (m=201)
No. of M Code for Spnd. Pos.=19 (n=20)
Basic Angle of Spnd. Pos.=18 (φ=18)
The below table indicates the meaning of the different M codes:

M code	Rotated angle
Mm (M201)	α=φ=18°
M(m+1) (M202)	α=2φ=36°
M(m+2) (M203)	α=3φ=54°
....	
M(m+n) (M220)	α=nφ=360°

Let's take the below example:

M19	(orientation)
M205	(rotation with 90° degrees)
...	(drilling)
M210	(rotation with 180°)
...	(drilling)

Before issuing an indexing M code, the spindle have to be oriented with M19. Then the spindle gets into position α=0.
A hole has to be drilled at 90°, and upon the command M205 the spindle takes up position α=5φ=5*18=90°.

The next hole has to be drilled in a position of 270°. As by an M code we can move the spindle only incrementally, we program M210, as in that case the movement will be 10*18°=180°.

N0823 M Code for Closing S Loop (S1...S16, integer, spindle)

It is the number of the M code upon which the position control loop closes. It differs from the M19 code (orientation) in the fact that upon M19 the spindle positions to the zero pulse, while upon this code it only stops and closes the position control loop.

10 Reference Point Parameter Group

N0900 Reference Type (A1...32, bit, axis)

N0900 A1...32	#7	#6	#5	#4	#3	#2	#1	#0
	GOS	**CIR**	**DIR**	**FLO**	**ABS**	**DCD**	**GRI**	**SWT**

#0 **SWT**: If the parameter value is 1, the reference position setting happens with running to switch. If #1 GRI=0, it takes up the reference point at the moment when it leaves the switch. If #1 GRI=1, after it leaves the switch it will also look for a zero pulse.

#1 **GRI**: If the parameter value =1 and #0 SWT=0, it will search for the first zero pulse and it will regard it as the reference point. If #0 SWT=1 it will search for a zero pulse leaving the switch and it will record it as the reference point.

#2 **DCD**: Distance coded reference position setting.

The #2 DCD=1 and the #3 ABS=1 settings mean that the distance-coded measuring device outputs not pulses but it sends the position data to the NC via EnDAT protocol.

#3 **ABS:** Absolute measuring system, after power on it immediately records the reference point.

The #2 DCD=1 and the #3 ABS=1 settings mean that the distance-coded measuring device outputs not pulses but it sends the position data to the NC via EnDAT protocol.

The #3 ABS=1 and the #4 FLO=1 settings mean that the reference point setting is carried out by the drive, and the existence of the reference point is indicated by the PLC program to the NC by setting the flag AP_RPE.

#4 **FLO**: floating reference point.

#5 **DIR**: The direction of reference position setting:

=0: positive

=1: negative

If SWT=1 and GRI=0: the direction of leaving the switch,

If SWT=1 and GRI=1: the direction of leaving the switch and zero pulse searching,

If SWT=0 and GRI=1: the direction of zero pulse searching,

If DCD=1: the direction of searching for zero pulses.

#5 **CIR**: *It can be used only in case of applying an **EnDat** encoder!*

We use it in case of ***rotary axes*** when a non-integer number encoder ***revolution*** belongs to a ***revolution*** of the ***rotary axis***, i.e. after power on the absolute position is not identifiable!

During set-up, when we set the CIR bit to 1, we nullify the N0908 EnDat Round Number parameter and restart the control.

After that, every time the axis position over-, or underflows the resolution determined on N0524 EnDat Resolution parameter during the movement, the control will calculate them with a sign and save them (the circles made) on the EnDat Round Number parameter. After power on, it multiplies the number of circles made (EnDat Round Number parameter value) with the EnDat Resolution parameter, and afterwards it adds the position read from the encoder to it. This way the absolute position emerges.

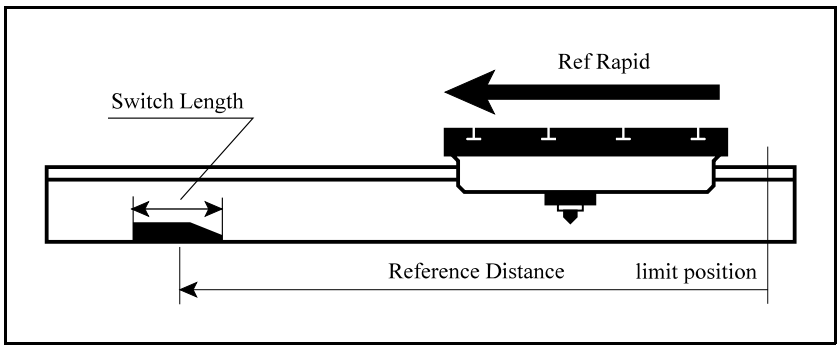
#7 **GOS**: The parameter is effective in the states SWT=1 and GRI=1, i.e. if it is necessary to run onto the switch and then to search for zero pulse during reference position setting. If the value of the parameter is:

=0: in the state AP_DECSW=1 (***it ran onto the reference point switch***) the axis will decelerate, and then, in the state AP_DECSW=0 (***it left the switch***) ***it will search for the zero pulse***.

=1: in the state AP_DECSW=1 (***it ran onto the reference point switch***) the axis will decelerate, and then, ***it will begin to search for the zero pulse*** immediately.
In this case, the only thing the control will monitor is whether the zero pulse arrives within the distance specified in the N0902 Switch Length parameter; it will not check the distance between the position of running onto the switch and the position of zero pulse arriving.

N0901 Reference Distance (A1...32, floating-point, axis)

Distance data, the number written on the parameter, based on #0 IND parameter, shall be interpreted in inch or mm, if #1 ROT=0.
If #1 ROT=1 the data will be interpreted in degrees.
It is used only in #0 SWT=1 case, its sign indicates the direction of search for the deceleration switch and its absolute value indicates the maximum path to be done.
The control searches for the deceleration switch in the direction determined by the sign of the parameter, by the speed determined on the Ref Rapid parameter, and it waits for the running on the switch within the distance determined by the parameter. If we have to be able to initiate the the reference position setting from all points of the axis, we have to specify the stroke of the axis. In case the switch signal (AP_DECSW=1 PLC flag) does not arrive within the given distance, the control will issue an error "*Has not found any ref-switches on axis "t" till the RefDis distance*", where "t" is the axis name.



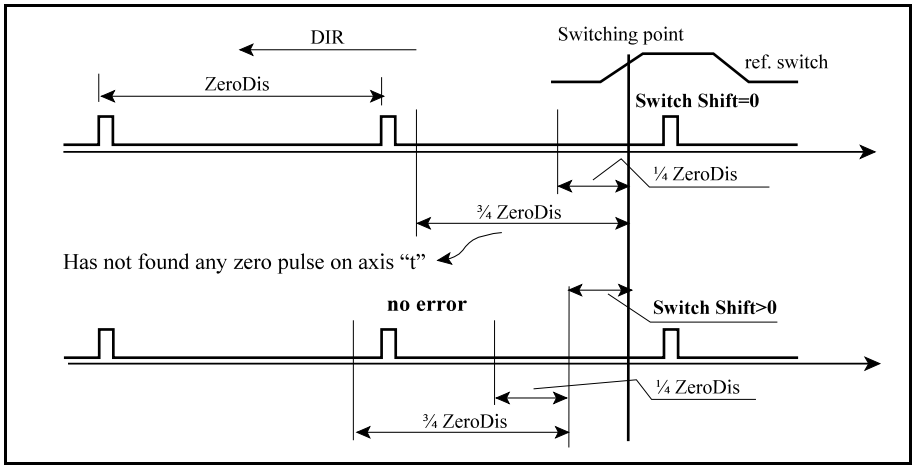
N0902 Switch Length (A1...32, floating-point, axis)

Distance data, the number written on the parameter, based on #0 IND parameter, shall be interpreted in inch or mm, if #1 ROT=0.
If #1 ROT=1 the data will be interpreted in degrees.
It is used in a #0 SWT=1 case and it is the length of the deceleration switch.
After running to the deceleration switch, it will decelerate and leave the switch in the direction determined on the DIR parameter, by the speed determined on the Ref Feed parameter. If it does not leave the deceleration switch (AP_DECSW PLC flag is not reset) within the distance determined on the parameter, a *"Has not decelerated from the ref-*

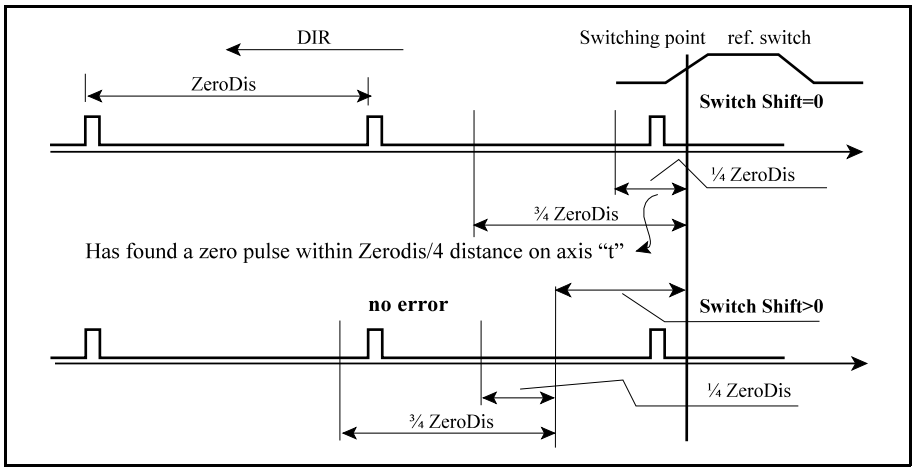
switch of axis "t" till SwLength distance error message will appear where "t"is the axis name.

N0903 Switch Shift (A1...32, floating-point, axis)

Distance data, the number written on the parameter, based on #0 IND parameter, shall be interpreted in inch or mm, if #1 ROT=0.
If #1 ROT=1 the data will be interpreted in degrees.
It is used only in #0 SWT=1 case, and it indicates the shift amount of the deceleration switch.
After leaving the switch, the zero pulse has to arrive within the 1/4 and 3/4 of the distance set on the ZeroDis parameter.
If the zero pulse arrives after the 3/4 ZeroDis distance after leaving the switch, the control will issue a "*Has not found any Zero pulse on axis "t"* error message where "t" is the axis name.



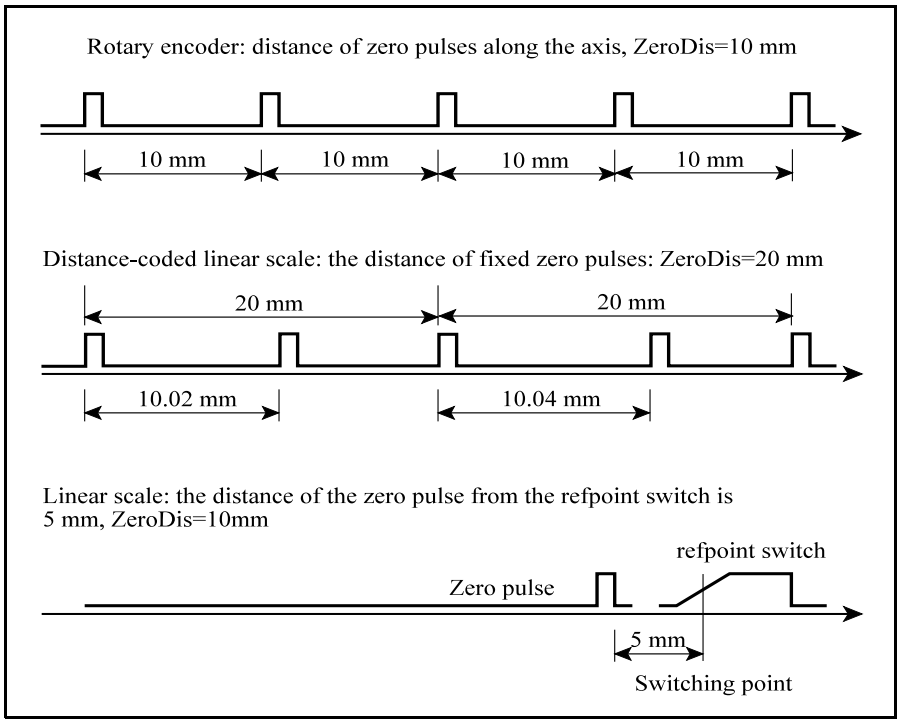
If the zero pulse arrives in a distance of 1/4 ZeroDis after leaving the switch, the control will issue a message *'Found a Zero pulse within a distance of Zerodis/4 on axis "t"'*, where "t" is the name of the axis.



The parameter shall be set in a way to ensure that the control does not sign any errors. The parameter is a positive number and it takes the shift into consideration in the direction of leaving the switch (DIR). Its effect is the same as if we physically shifted the switch to the direction of DIR.

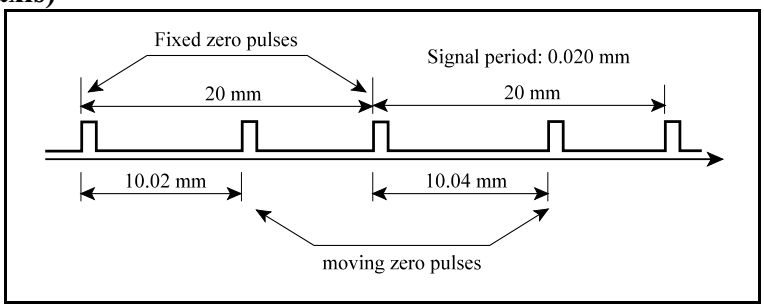
N0904 ZeroDis (A1...32, floating-point, axis)

Distance data, the number written on the parameter, based on #0 IND parameter, shall be interpreted in inch or mm, if #1 ROT=0.
If #1 ROT=1 the data will be interpreted in degrees.
In case of #0 GRI=1 it tells the distance of zero pulses along the axis.
In case of #2 DCD=1 (distance coded scale or encoder) it means the distance of fixed zero pulses.



N0905 BasDist (A1...32, integer, axis)

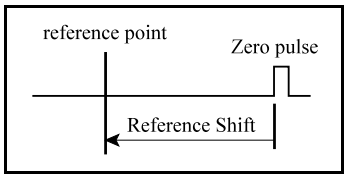
It has to be determined in case of distance coded devices. Meaning of the parameter: distance of fixed zero pulses, depending on the signal period.



Heidenhain type	Distance of fixed zero pulses in mm	Signal period of the linear scale in mm	**BASDISTn**
LF	20 mm	0.004 mm	20/0.004=5000
LS	20 mm	0.020 mm	20/0.020=1000
LB	80 mm	0.040 mm	80/0.040=2000

N0906 Reference Shift (A1...32, floating-point, axis)

Distance data, the number written on the parameter, based on #0 IND parameter, shall be interpreted in inch or mm, if #1 ROT=0.
If #1 ROT=1, the data will be interpreted in degrees.
It will step this distance after having found the zero pulse and it will regard the position recorded thereafter, as the reference point.
If the absolute position of the machine reference point is fixed due to some reason, then it may set this point without physically shifting the zero pulse, by using this parameter. After having found the zero pulse, the axis will do also the signed distance specified on this parameter by Ref Feed speed.



In the case of distance-coded scales, it has to be set in such a way, so that the reference point will be inside the positive limit, close to it, because the G28 instruction will send the axis to this point.

N0907 EnDat Machine Reference (A1...32, floating-point, axis)

It is the offset of the position measured from the EnDat encoder, per axis. After turning on, it will shift the absolute position data arriving from the encoder by the value set here.
Its setting: let's move with the axis to any of the stroke limit and in the parameter editor we have to push the "Position request" button. Save the parameter and restart the control.
Its effect:

– It will eliminate the jump of the axis if the absolute encoder rolls over inside the stroke. E.g. the resolution of the encoder is 37-bits and it rolls over inside the stroke from 2^{37}-1 to 0,
– The positions displayed fall into the range of small numbers,
– By the modification of the encoder multiplication/division a linear compensation may be applied.
– ***Due to the G28 reference point setting instruction, the axis will move to this position.***

N0908 EnDat Round Number (A1...32, integer, axis)

*It can be used only in case of applying an **EnDat** encoder!*
We use it in case of ***rotary axes***, if a non-intenger number encoder revolution belongs to a revolution of the ***rotary axis***, i.e. the absolute position cannot be identified after turning on!
Any time the position of the axis over- or underflows the resolution determined on N0524 EnDat Resolution parameter during the movement, the control will count them with a sign and save them (the circles done) on the EnDat Round Number parameter. After power on, the number of circles done (EnDat Round Number parameter value) will be multiplied by

the EnDat Resolution parameter, and then it will add the position read from the encoder to it. The absolute position emerges this way.
The counting is enabled by the CIR=1 state of the N0900 Reference Type parameter.

N0909 ABS position alarm distance (A1...32, floating-point, axis)

Distance data, the number written on the parameter, based on the#0 IND parameter is to be interpreted in inches or millimeters, if #1 ROT=0.
If #1 ROT=1 the data is to be interpreted in degrees.
The parameter will be taken into account only in the case, if there is #3 ABS=1 in the N0900 Reference Type parameter, i.e. there is an absolute measuring device on the machine. It is used for monitoring whether ***in the switch-off state of the machine***:

– mechanical linkage between the measuring device and the motor or the machine was disassembled,
– the slides were displaced,
– the battery in the measuring device run down.

During motion of the axis, if displacement of the axis is greater than the value written in the parameter, the control will save the absolute position of the axis. Following the switch-on, if the difference between the position provided by the measuring device and the saved position is greater than the value written in the parameter, the control will not enter the status of existing the reference point for the given axis, and it will send the error message ‘The encoder position has changed on the axis. REF error!’.
In such case, first, the reference point has to be set according to the way mentioned at the *N0907 EnDat Machine Reference parameter. Then, according to the way mentioned at the N0200 Reference Position1 parameter , it is necessary to measure the position of the reference point in the machine coordinate system.*
If it is not necessary to carry out the settings mentioned above, choose the mode of reference poin setting and select the given axis by pushing its jog button. With this, the error will be deleted and the reference point will be entered.
If the value of the parameter is 0, the control does not carry out monitoring.

11 Axis Limits Parameter Group

N1000 Range Enable (A1...32, bit, axis)

N1000 A1...32	#7	#6	#5	#4	#3	#2	#1	#0
						RE3	**RE2**	**RE1**

#0 **RE1**: It enables to check the first range on the given axis. The first range relates always to an area **outside forbidden** and it defines the **stroke limits** of the given axis.

#1 **RE2**: It enables to check the second range on the given axis. The area may be **forbidden either inside or outside**.

#2 **RE3**: It enables to check the third range on the given axis. The third range refers always to an area **forbidden inside**.

N1001 StrkCont (L1...8, bit, channel)

N1001 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
				ABA	**SKD**	**CBM**	**STE**	**EXT**

#0 **EXT**: It determines whether range 2 is forbidden outside or inside in the given channel. If
#0 EXT=0, the area is forbidden inside,
#0 EXT=1, it is forbidden outside.

#1 **STE**: It determines whether the checking of range 2 is enabled or not.
Checking of range 2 is applied to axes which are enabled on parameter #1 RE2.

#2 **CBM**: If the parameter value
=0, the control does not carry out,
=1 the control carries out
checking the limit position ranges, before movement in each block.

#3 **SKD**: If the CBM (Check Before Move) bit is 1 and SKD bit is 1, it will not carry out checking the limit position ranges in blocks G31 and G37 before starting movement.

#4 **ABA**: If the
If the parameter
=0,
in the given channel, PLC flags enabling parameter selection per axis and per direction will be effective: AP_LIMSELP in a positive direction, AP_LIMSELN in a negative direction, selects from among ranges 1A and 1B.
E.g.: If ABA=0, AP_LIMSELP=1 and AP_LIMSELN=0 on axis 3, in positive direction parameter “B”, in negative direction parameter “A”will be effective.
If the parameter
=1,
in the given channel, the CP_LIMSEL PLC flag will be effective. The CP_LIMSEL flag will decide which stroke limit parameter group (CP_LIMSEL=0: group 1A, CP_LIMSEL=1: group 1B) shall be valid in range 1 for all axes of the channel, in both directions.

N1002 Range1A Positive (A1...32, floating-point, axis)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm if #1 ROT=0.
If #1 ROT=1, the data will be interpreted in degrees.
It is the positive stroke limit of axis, specified in the machine coordinate system. It is effective when:
parameter #4 ABA=0, and in the corresponding channel CP_LIMSEL=0, or
parameter #4 ABA=1, and on the corresponding axis AP_LIMSELP=0, and
on the given axis #0 RE1=1.

N1003 Range1A Negative (A1...32, floating-point, axis)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm if #1 ROT=0.
If #1 ROT=1, the data will be interpreted in degrees.
It is the negative stroke limit of axis, specified in the machine coordinate system. It is effective when:
parameter #4 ABA=0, and in the corresponding channel CP_LIMSEL=0, or
parameter #4 ABA=1, and on the corresponding axis AP_LIMSELN=0, and
on the given axis #0 RE1=1.

N1004 Range1B Positive (A1...32, floating-point, axis)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm if #1 ROT=0.
If #1 ROT=1, the data will be interpreted in degrees.
It is the positive stroke limit of axis, specified in the machine coordinate system. It is effective when:
parameter #4 ABA=0, and in the corresponding channel CP_LIMSEL=1, or
parameter #4 ABA=1, and on the corresponding axis AP_LIMSELP=1, and
on the given axis #0 RE1=1.

N1005 Range1B Negative (A1...32, floating-point, axis)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm if #1 ROT=0.
If #1 ROT=1, the data will be interpreted in degrees.
It is the negative stroke limit of axis, specified in the machine coordinate system. It is effective when:
parameter #4 ABA=0, and in the corresponding channel CP_LIMSEL=1, or
parameter #4 ABA=1, and on the corresponding axis az AP_LSELN=1, and
on the given axis #0 RE1=1.

☞ ***Warning***: *The checking of the first range in the corresponding channel will be blocked by the CP_LIM1DIS=1 PLC flag state.*

N1006 Range2 Positive (A1...32 floating-point, axis)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm if #1 ROT=0.
If #1 ROT=1, the data will be interpreted in degrees.
It is specified in the machine coordinate system. It is effective when #1 STE=1, and on the given axis #1 RE2=1.

N1007 Range2 Negative (A1...32, floating-point, axis)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm if #1 ROT=0.
If #1 ROT=1, the data will be interpreted in degrees.
It is specified in the machine coordinate system. It is effective when #1 STE=1, and on the given axis #1 RE2=1.

☞ ***Warning***: *The checking of the second range in the corresponding channel will be disabled by the CP_LIM2DIS=1 PLC flag state.*
The range may be forbidden both inside and outside, depending on #0 EXT parameter.

N1008 Range3 Positive (A1...32, floating-point, axis)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm if #1 ROT=0.
If #1 ROT=1, the data will be interpreted in degrees.
It is the positive limit of an area forbidden inside, specified in the machine coordinate system. It is effective when on the given axis #1 RE3=1.

N1009 Range3 Negative (A1...32, floating-point, axis)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm if #1 ROT=0.
If #1 ROT=1, the data will be interpreted in degrees.
It is the negative limit of an area forbidden inside, specified in the machine coordinate system. It is effective when on the given axis #1 RE3=1.

☞ ***Warning***: *The checking of the third range in the corresponding channel will be blocked by the CP_LIM3DIS=1 PLC flag state.*

☞ ***Warning***: *In case an axis reaches the stroke limit of any of the ranges, the NC will set the corresponding positive-direction AN_OTP, or the corresponding negative-direction AN_OTN flag.*

12 Position Switches Parameter Group

N1100+n SWn Axis Number n=1...32 (integer)

n=1...32: different parameters for all position switches from the range 1...32
It tells, to which axis shall the n-th software switch be mounted. Value: 1, ..., 32.
If the parameter value 0, the n-tch switch does not exist.

N1132+n SWn Min Pos n=1...32 (floating-point)

n=1...32: different parameters for all position switches from the range 1...32
Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm if #1 ROT=0.
If #1 ROT=1, the data will be interpreted in degrees.
It is the machine position of negative end of the n-th switch, on the given axis.

N1164+n SWn Max Pos n=1...32 (floating-point)

n=1...32: different parameters for all position switches from the range 1...32
Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm if #1 ROT=0.
If #1 ROT=1, the data will be interpreted in degrees.
It is the machine position of positive end of the n-th switch, on the given axis.

Before each PLC cycle, the control checks whether the n-th switch does exist or not. If yes, according to the machine position measured from the encoder of the axis, it will set the PLC flags

N_SW,
N_SWN,
N_SWP

in the following way:

– If the position is lower than SWn Min Pos, then
  N_SW,(n-1)=0
  N_SWN,(n-1)=1: the axis is in the negative direction from the switch
  N_SWP,(n-1)=0
– If the position is higher than SWn MIN POS, but lower than SWn Max Pos
  N_SW,(n-1)=1: the axis is on the switch
  N_SWN,(n-1)=0
  N_SWP,(n-1)=0
– If the position is higher than SWn Max Pos
  N_SW,(n-1)=0
  N_SWN,(n-1)=0
  N_SWP,(n-1)=1: the axis is in the positive direction from the switch.

(PLC flag indexed by n-1 belongs to switch n.)

13 PLC Constants Parameter Group

N1200 PLC Bits0 (bit)

N1200	#7	#6	#5	#4	#3	#2	#1	#0
	P07	**P06**	**P05**	**P04**	**P03**	**P02**	**P01**	**P00**

N1201 PLC Bits1 (bit)

N1201	#7	#6	#5	#4	#3	#2	#1	#0
	P17	**P16**	**P15**	**P14**	**P13**	**P12**	**P11**	**P10**

N1202 PLC Bits2 (bit)

N1202	#7	#6	#5	#4	#3	#2	#1	#0
	P27	**P26**	**P25**	**P24**	**P23**	**P22**	**P21**	**P20**

N1203 PLC Bits3 (bit)

N1203	#7	#6	#5	#4	#3	#2	#1	#0
	P37	**P36**	**P35**	**P34**	**P33**	**P32**	**P31**	**P30**

N1204 PLC Bits4 (bit)

N1204	#7	#6	#5	#4	#3	#2	#1	#0
	P47	**P46**	**P45**	**P44**	**P43**	**P42**	**P41**	**P40**

N1205 PLC Bits5 (bit)

N1205	#7	#6	#5	#4	#3	#2	#1	#6
	P57	**P56**	**P55**	**P54**	**P53**	**P52**	**P51**	**P56**

N1206 PLC Bits6 (bit)

N1206	#7	#6	#5	#4	#3	#2	#1	#0
	P67	**P66**	**P65**	**P64**	**P63**	**P62**	**P61**	**P60**

N1207 PLC Bits7 (bit)

N1207	#7	#6	#5	#4	#3	#2	#1	#0
	P77	**P76**	**P75**	**P74**	**P73**	**P72**	**P71**	**P70**

P00, ..., P77: 64 pcs., free purpose, bit-type, PLC parameters. They serve for the configuration of the PLC program.
They are directly accessible parameters, can be used as contacts in PLC program through PLC flags N_P00, ..., NP_77.

N1208+n PLC DWordn n=1...32 (integer)

32 pcs., free purpose, double-word PLC parameters. They serve for the configuration of PLC program.

The symbol of the first PLC DWord1 parameter in the PLC is N_PDW1, which is directly accessible, and the other parameters may be read with an indexed access.

N1240+n PLC Doublen n=1...32 (floating-point)

32 db., free purpose, 64-bit, floating-point, PLC parameter. They serve for the configuration of PLC program.

The symbol of the first PLC Double1 parameter in the PLC is N_PDB1, which is directly accessible, and the other parameters may be read with an indexed access. (They are in a 2 DWORD distance from each other!)

N1273 Lubrication Distance (A1...32, floating-point, axis)

It is the parameter of lubrication, per axis. If the given axis has moved the distance written on the parameter, NC will set the AN_LUBR PLC flag for one cycle.

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm if #1 ROT=0.

If #1 ROT=1, the data will be interpreted in degrees.

N1274 PLC PrgConfig (bit)

N1207	#7	#6	#5	#4	#3	#2	#1	#0
							NVF	**MRW**

#0 **MRW**: This parameter controls execution of the MR101, MR104, MR107 instructions (reading the memory) used in the PLC program. If the value of the parameter:

=0: the instruction will be executed immediately. If execution of more than one MR101, MR104 or MR107instruction in a PLC cycle at the same time is enabled by the PLC program, the control may easy run into the error of PLC Timeout. (Simultaneously, only one of its inputs enabling such MR instruction can be TRUE in the PLC.)

=1: the instruction will not be executed immediately. More than one MR101, MR104 or MR107instruction in a PLC cycle at the same time can be enabled in the PLC program. (Simultaneously, more than one of its inputs enabling such MR instruction can be TRUE in the PLC.)

☞ ***Warning!*** *After rewriting the parameter, the control must be turned off!*

#1 **NVF**: If a software with 1.10.1 or higher version number is to be loaded in a battery-driven control, ESYS card will have to be connected to the control so that the built-in options can be enabled.

In the case of battery-driven controls without ESYS card, the non-volatile PLC variables will be saved in file automatically on turning off, and they will be read out from the file on turning on, and it is executed by the system automatically.

When ESYS card is used, the non-volatile variables of the PLC program will have to be saved to the ESYS card by the PLC program using the MW instruction, and they will have to be read out using MR instruction after turning on.

If the value of the parameter is

=0: saving the PLC variables to be retained to the ESYS has to be written in the PLC program and they have to be read out on turning on, by the use of MW and MR instructions.

=1: the system will automatically save the PLC variables to be retained in file on turning off and will also read out them from file after turning on. Thus, the PLC program will not have to be modified.

☞ ***Warning!***

The case of NVF=1 must not be used for controls without battery*, because stopping the control occurs during function execution and, for example in the case of power failure, it results in losing the data!*

In the case of controls with battery, stopping the control in case of power failure is solved.

14 Program Parameter Group

N1300 DefaultG1 (L1...8, bit, channel)

N1300 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
	G91	**G44**	**G43**	**G23**	**G20**	**G19**	**G18**	**G01**

After power on or reset the various G-code groups will be placed in a pre-set default state, except for the G codes below:

#0 **G01**: interpolation code:
=0: G0 rapid traverse rate positioning,
=1: G1 linear interpolation.
#1 **G18**: plane selection 1:
=0: G17 XY plane, provided that G19=0
=1: G18 ZX plane.
#2 **G19**: plane selection 2:
=0: G18 parameter will decide
=1: G19 YZ plane, provided that G18=0.

	#1 **G18**	#2 **G19**
G17	0	0
G18	1	0
G19	0	1

#3 **G20**: input unit of length and speed data:
=0: G21 metric,
=1: G20 inch.
#4 **G23**: stored stroke check:
Not used.
#5 **G43**: tool length compensation:
=0: it will be cancelled, it will take up G49 state after power on, provided that G44=0,
=1: it is taken into consideration with G43 code.
#6 **G44**: length compensation:
=0: it will be cancelled, it will take up G49 state after power on, provided that G43=0,
=1: it is taken into consideration with G44 code if G43=0.
#7 **G91**: position data:
=0: they will be interpreted as absolute commands (G90),
=1: they will be interpreted as incremental commands (G91).

N1301 DefaultG2 (L1...8, bit, channel)

N1301 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
	CLR	**MBM**						**G95**

After power on or reset the various G-code groups will be placed in a pre-set default state, except for the G codes below:

#0 **G95**: unit of feedrate:
=0: feed per minute (G94),
=1: feed per revolution (G95).

#6 **MBM**: (Multiple Buffer Mode) If the parameter
=0: the control will keep in the puffer only a minimum number of blocks (in G40 state 1, in G41, G42 state 3 blocks),
=1: the control will process the highest possible number of blocks in advance and place them into the puffer.
#7 **CLR**: upon a reset:
=0: it will set the G code groups to the power on default values,
=1: based on the CLR G Table1, 2, 3, 4, 5 parameters, it will set the default value per G code groups, or it will leave the modal values unchanged.

N1302 CLR G Table1 (L1...8, bit, channel)

N1302 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
	C07	**C06**	**C05**	**C04**	**C03**	**C02**	**C01**	

N1303 CLR G Table2 (L1...8, bit, channel)

N1303 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
	C15	**C14**	**C13**	**C12**	**C11**	**C10**	**C09**	**C08**

N1304 CLR G Table3 (L1...8, bit, channel)

N1304 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
	C23	**C22**	**C21**	**C20**	**C19**	**C18**	**C17**	**C16**

N1305 CLR G Table4 (L1...8, bit, channel)

N1305 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
	C31	**C30**	**C29**	**C28**	**C27**	**C26**	**C25**	**C24**

N1306 CLR G Table5 (L1...8, bit, channel)

N1306 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
	C39	**C38**	**C37**	**C36**	**C35**	**C34**	**C33**	**C32**

In case #7 CLR=1 on the DefaultG2 parameter upon reset
If the **C01...C39**: parameter:
=0: the corresponding G code group assumes the default value,
=1: the corresponding G code group is left in the state assumed earlier.

Parameter	G code group	G code	Function
C01	1	G0, G1, G2, G3, G33, G34	interpolation
C02	2	G17, G18, G19	plane selection
C03	3	G90, G91	absolute/incremental programming
C04	4	G22, G23	stored stroke check on/off
C05	5	G94, G95	feed per minute/revolution
C06	6	G20, G21	inch/metric data input
C07	7	G40, G41, G42	radius compensation on/off
C08	8	G43, G44, G43.1, G43.4, G43.5, G43.7, G43.8, G43.9 G49	length compensation on/off
C09	9	G73, G74, G76, G80, G81, G82, G83, G84, G84.2, G84.3, G85, G86, G87, G88, G89	drilling cycles
C10	10	G98, G99	drilling cycles: return to initial/R point level
C11	11	G50, G51	scaling on/off
C12	12	G66, G66.1, G67	modal macro call on/off
C13	13	G96, G97	constant surface speed control on/off
C14	14	G54, G54.1, G54.2, G55, G56, G57, G58, G59	workpiece coordinate system selection
C15	15	G61, G62, G63, G64	feed control functions
C16	16		
C17	17	G15, G16	polar cordinate data input on/off
C18	18	G68.1, G68.2, G69.1	3D rotations on/off
C19	19		
C20	20		
C21	21	G12.1, G13.1	polar interpolation on/off
C22	22	G50.1, G51.1	programmable mirror image on/off
C23	23		

Parameter	G code group	G code	Function
C24	24		
C25	25		
C26	26		
C27	27		
C28	28		
C29	29		
C30	30		
C31	31	G50.2, G51.2	polygonal turning on/of
C32	32		
C33	33		
C34	34	G80.8, G81.8	electronic gear box on/off
C35	35		
C36	36		
C37	37		
C38	38		
C39	39		

N1306+n M Suppr Bn n=1...10 (L1...10, integer, channel)

10 different M code values can be specified on the parameters.
The codes set here suppress buffering the blocks. Buffering restarts again only after the execution of the function (after returning the CP_FIN PLC flag).

N1316+n M GR Low Suppr Bn n=1...8 (L1...8, integer, channel)
N1324+n M GR High Suppr Bn n=1...8 (L1...8, integer, channel)

On the 8 parameter pairs 8 different M code groups can be specified. On the parameter marked as Low the lowest number of group shall be written, and on the parameter marked as High the highest number of group shall be written.
The codes set here suppress buffering the blocks. Buffering restarts again only after the execution of the function (after returning the CP_FIN PLC flag).

☞ ***Warning***

*The codes **M00, M01, M02, M30 always suppress** the buffering; these codes do not have to be written on these parameters.*
The function in which the PLC an axis or a spindle attaches or detaches, in which takes over or returns the control of an NC axis, shall be always specified on the parameter to suppress the buffering!

N1332+n Aux Fu Addrn n=1..3 (L1...8, integer, channel)

The address of the first, second and third auxiliary function. Specifiable values:

Address	**none**	**A**	**B**	**C**	**U**	**V**	**W**
Value	0	65	66	67	85	86	87

The value of addresses specified here as auxiliary functions, will be passed to PLC program in registers CN_AUX1C, CN_AUX2C, CN_AUX3C to execute them.

N1336 Funct Suppr B (L1...8, bit, channel)

N1336 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
				AU3	**AU2**	**AU1**	**TSB**	**SSB**

The following functions can be appointed individually to suppress buffering of blocks:

#0 **SSB**: If the parameter
=0: the spindle function (code S) does not suppress buffering,
=1: the spindle function (code S) suppresses buffering.

#1 **TSB**: If the parameter
=0: the tool function (code T) does not suppress buffering,
=1: the tool function (code T) suppresses buffering.

#2 **AU1**: If the parameter
=0: the first auxiliary function (the code set on Aux Fu Addr1 parameter) does not suppress buffering,
=1: suppresses buffering.

#3 **AU2**: If the parameter
=0: the second auxiliary function (the code set on Aux Fu Addr2 parameter) does not suppress buffering,
=1: suppresses buffering.

#4 **AU3**: If the parameter
=0: the third auxiliary function (the code set on Aux Fu Addr3 parameter) does not suppress buffering,
=1: suppresses buffering.

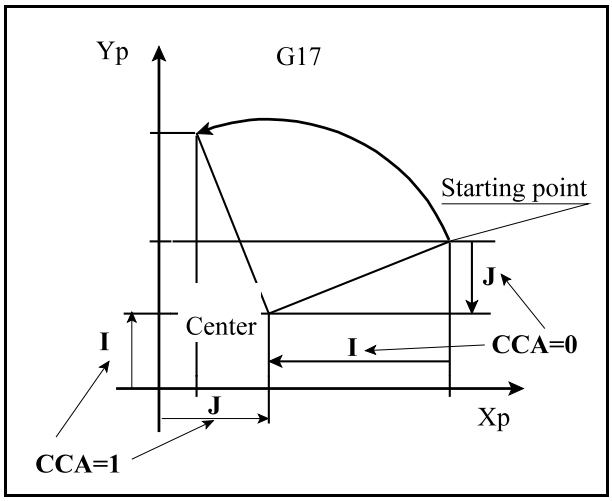
N1337 Execution Config (L1...8, bit, channel)

N1337 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
			REL	**CBB**	**HEF**	**CCA**	**SEC**	**PCH**

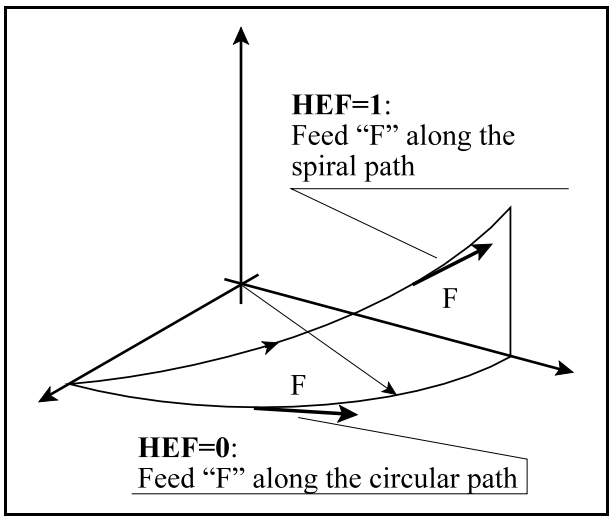
#0 **PCH**: in G00 positioning blocks at the end of the block, if the parameter
=0: it does not wait for the “In position” signal,
=1: it waits for the signal.
The width of the “In position” signal can be set on the N0516 Inpos parameter.
If the function of overlapping of rapid traverse movements is enabled on the N0407 Acc Contr parameter, the setting PCH=0 shall be applied!

#1 **SEC**: address P specified in G04 dwell command is considered to be if the parameter
=0: in unit second/spindle revolution according to G94/G95 state
=1: always in seconds, independently of G94/G95 state.

#2 **CCA**: during the programming of the centre of the circle on addresses I, J, K, if the parameter
=0: I, J, K will always be the incremental distance from the start point of the arc,
=1: I, J, K will always be the absolute coordinate of the center.



#3 **HEF**: in case of programming a spiral interpolation, the feed, if the parameter
=0: will be taken into consideration in the plane of the circle, along the circle,
=1: along the spiral path.

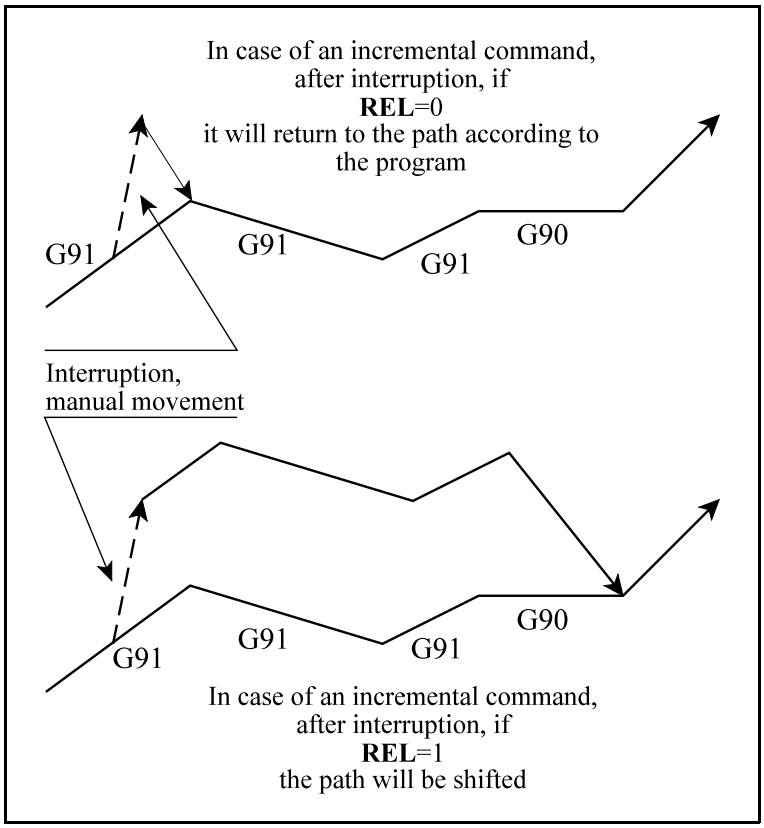


#4 **CBB**: the conditional block skip instruction "/" if the parameter
=0: it will suppress buffering, (in states G41, G42 the path is becoming distorted)
=1: it will not suppress it, (in states G41, G42 the path is not becoming distorted, however, the conditional block switches shall be set before starting the program).

#5 **REL**: after an unconditional return from interrupted state (neither the Block Return, nor the Block Restart switch is on), if the block to be executed contains an incremental movement, and the parameter is:

=0: it will move on the path calculated from the beginning of the program,

=1: it will move on the shifted path, untill it receives an absolute command (G90).



N1338 Block No Search (L1...8, bit, channel)

N1338 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
							DSM	**M06**

#0 **M06**: The tool change will happen - if the parameter is

=0: upon code T,

=1: upon code M06.

The parameter will be used by the control during searches for a block, to regulate the collection of functions. If the parameter value is 0, after searching for the block it will collect the tool number (T) to be executed.

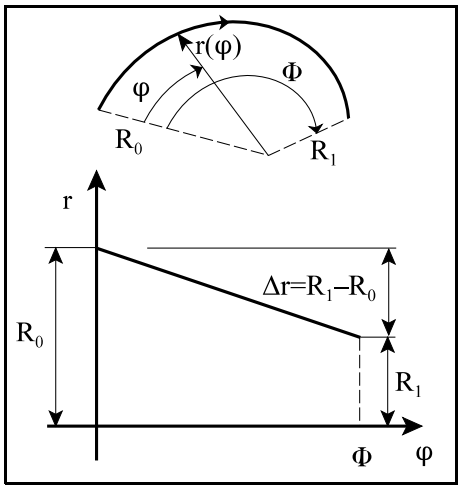
If the parameter value is 1, during the search for a block it will collect two tool numbers (T) and the code M06: T(to be changed), M06 and T(to be prepared).

#1 **DSM**: during searching for a block, it will - if the parameter is

=0: not read into to functions and G codes appointed for macro call, and it will not collect any information in them,

=1: it will read into them and collect all information.

N1339 Radius Diff (L1...8, floating-point, channel)



Distance data, the number written on the parameter, based on #0 IND parameter, shall be interpreted in inch or mm.
It is the tolerance of radii in the start point and end point of the circle. If the difference is lower than the value given, it will interpolate a circle with a varying radius, and if it is higher, an alarm is generated.

N1340 Inpos Timeout (L1...8, floating-point, channel)

It is the timeout of the signal "In position", expressed in seconds (sec).
It will wait for the signal "In position" (N0516 Inpos window)
in G61 state after each movement block,
in G9 at the end of the given block,
in G64 state after each positioning (G0, G53, G30) (if PCH=1).
If the signal "In position" does not arrive within the time determined on the parameter, "*Position error"* message is generated and the machining will stop.

N1340+n M GR Low n n=1...16 (L1...8, integer, channel)
N1356+n M GR High n n=1...16 (L1...8, integer, channel)

On the 16 parameter pairs 16 different M code groups can be specified. The code of the group with the lowest number shall be written on the parameter marked as Low, and the code of the group with the highest number shall be written on the parameter marked as High.
The values of these M codes are written into registers CP_MGR1, ..., CP_MGR16 by the PLC program, and the values of these registers are displayed in the M codes window.
The parameter pair M GR Low n and M GR High n is in accordance with the CP_MGRn register.
The parameters shall be specified in a way to ensure that the M codes in each groups are codes representing each other excluding machine states.
During the execution of part program the M codes are filtered in a way that only one M code belonging to the same group can be in the given block, otherwise Contradictory M codes error message will be issued.
The control will take into consideration the values set on the parameter also during the search for a block, at the collection of M codes. Among M codes belonging to the same group specified last will be collected
Let's have a look on the following M codes:

M51: clamp chuck on spindle S1
M52: unclamp chuck on spindle S1
M53: unclamp chuck on spindle S1 when rotating

Let's set the parameter in the following way:

N1341 M GR Low 1=51
N1357 M GR High 1=53

Clamping state:

M54: internal clamping S1
M55: external clamping S1

The parameters:

N1342 M GR Low 2=54
N1358 M GR High 2=55

During the search for a block, from among codes M51, M52, M53 only the one programmed as the last one will be collected, and executed. The same applies also to the M54, M55 group. From among the appropriate machine states only one will be written out in CP_MGR1, 2 registers, e.g. after clamping of the chuck, the content of registers will be as follows:

CP_MGR1=51 – chuck clamped on spindle S1 and
CP_MGR2=54 – internal clamping on spindle S1.

15 Tool Compensation Parameter Group

N1400 No. of Tool Offsets (L1...8, integer, channel)

It shall be determined per channel, how many tool compensation cells shall be available in the given channel.
In the whole system, the total amount of compensation cells shall not exceed the available space: max. 999 cells.
In each channel the reference to a compensation cell either made from a program (T, H, D), or viewed on the offset table, shall start from 1 and will last till the value of the parameter set here.

N1401 No. of Common Tool Offsets M (integer)

A common parameter which is valid for all milling channels.
It indicates the number of common compensation cells which can be referenced in all milling channels.
In each channel the reference to a compensation cell ranging from 1 to the value set here either made from a program on addresses H or D, or viewed on the offset table, will show the common offset values.

Example:

Let's have 3 channels. Let's have
30 compensation cells in channel 1 (No. of Tool Offsets L1=30)
40 compensation cells in channel 2 (No. of Tool Offsets L2=40),
60 compensation cells in channel 3 (No. of Tool Offsets L3=60).
The number of common compensation cells is 10 (No. of Common Tool Offsets=10).

<table>
<tr><td>Cell No.</td><td>1. channel</td><td>2. channel</td><td>3. channel</td></tr>
<tr><td>N001</td><td colspan="3" rowspan="3">From N001 to N010 all three channels see the same common compensation values</td></tr>
<tr><td>...</td></tr>
<tr><td>N010</td></tr>
<tr><td>...</td><td rowspan="2">From N011 to N030 it sees its own offsets</td><td rowspan="4">From N011 to N040 it sees its own offsets</td><td rowspan="5">From N011 to N060 it sees its own offsets</td></tr>
<tr><td>N030</td></tr>
<tr><td>...</td><td></td></tr>
<tr><td>N040</td><td></td></tr>
<tr><td>...</td><td></td><td></td></tr>
<tr><td>N060</td><td></td><td></td></tr>
</table>

N1402 Tool Meas (L1...8, bit, channel)

N1402 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
						ONL	**LSG**	**DIA**

#0 **DIA**: The value of cutter offset in the offset table in milling channel is:
if the parameter
=0: in radius,
=1: in diameter

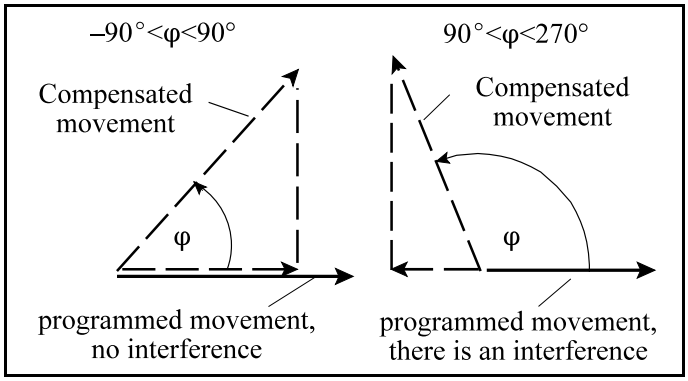
#1 **LSG**: it sets the sign of the tool length offset in milling channel, in case of data input during tool length measurement. If the parameter
=0: for a call by G43,
=1: for a call by G44.

#2 **ONL**: On the tool measurement screen, if the parameter
=0: it will not adjust the cursor to the offset cell, equal to the tool number,
=1: it will adjust the cursor to the offset cell, equal to the tool number.

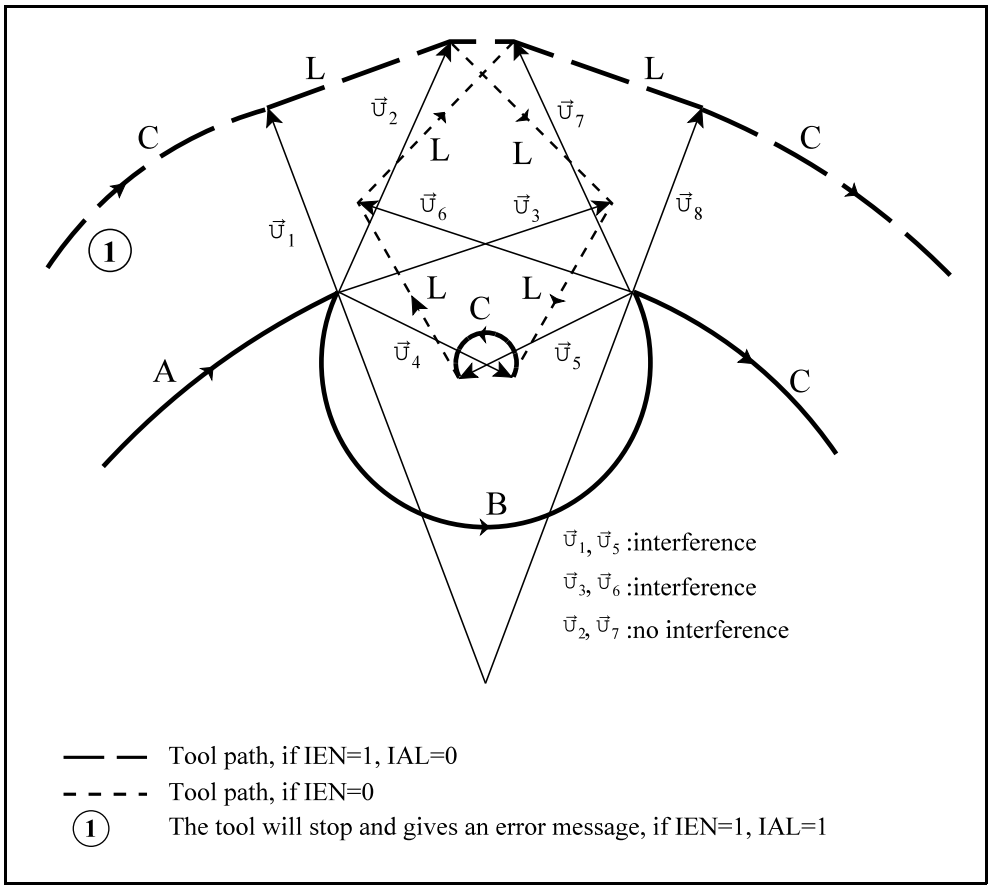
N1403 Interference (L1...8, bit, channel)

N1403 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
						GAP	**AAL**	**IEN**

#0 **IEN**: in G41, G42 state the interference check, if the parameter
=0: is not enabled,
=1: is enabled.
Interference error will be generated if the angle between the programmed and compensated path falls between 90° and 270° In other words, if the component of the compensated movement falling on the programmed movement have a different sign.



#1 **AAL**: in case of G41, G42, in the enabled state of the interference check, (#0 IEN=1), if the parameter
=0: the interference error will be automatically corrected.
=1:it will not be corrected but an *Interference error* message will be sent.



During tool radius compensation (G41, G42 state), if parameters IEN=1 and IAL=1, after the angle check an alarm *"Interference error"* will arise, one block earlier than the error occurs.

In cases parameters are set the way IAL=0, but IEN=1, no alarm arising, but the control will try to correct the contour automatically, for the purpose of preventing cut-ins. The correction process is as follows:

Radius compensation is effective on blocks A, B, and C. Between blocks A and B the calculated compensation vectors are: $\vec{v}_1$, $\vec{v}_2$, $\vec{v}_3$, $\vec{v}_4$. Between blocks B and C the compensation vectors are: $\vec{v}_5$, $\vec{v}_6$, $\vec{v}_7$, $\vec{v}_8$.

If there is an interference between $\vec{v}_4$ and $\vec{v}_5$ they are omitted,
If there is an interference between $\vec{v}_3$ and $\vec{v}_6$, they are omitted,
If there is an interference between $\vec{v}_2$ and $\vec{v}_7$ they are omitted
If there is an interference between $\vec{v}_1$ and $\vec{v}_8$ they cannot be omitted and alarm is arised.

From the above it can be seen that the offset vectors are paired from start point and end point of block B, and they are omitted per pairs. If on one of the sides the number of offset vectors is one or decreases to one, it will leave only those being on the other side. The abandoning is continued until the interference is in effect. The first compensation vector of the start point and the last one of the end point of block B cannot be abandoned. If due to the omissions the interference ceases no error is sent, if not, an alarm *"Interference*

error" is generated. The rest of compensation vectors left from the abandonments are always connected with straight lines even if block B is a circle.

#2 **GAP**: Not used.

N1404 BK No. Interf (L1...8, integer, channel)

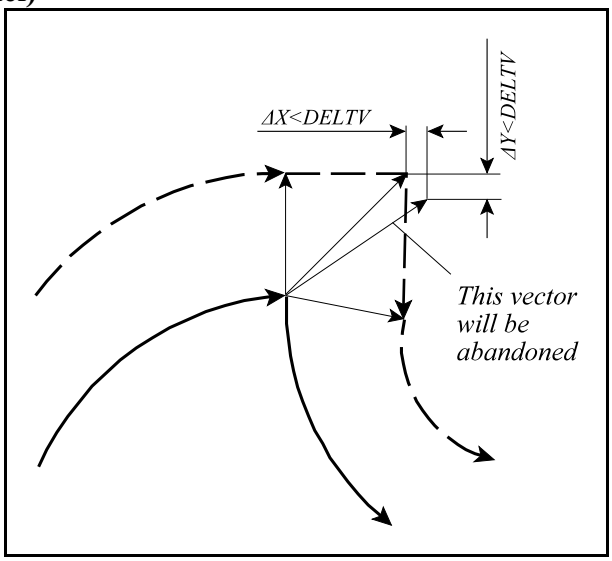
Interpretation: integer: 0....8

In G41, G42 state it is the number of blocks read in advance for the purpose of interference check. In a default case, if the parameter value is 0, it will read always the current N, the following blocks N+1 and N+2 and it will carry out the interference check to the start and end point of block N+1. If the parameter is higher than 0, it will read so many blocks in advance, as the parameter value is and it will carry out an interference check between the current block N and the following N+1, N and N+2, ... N and N+ BK No. Interf+2. *The higher this number is, the longer time will be necessary to process a block, which may slow down the operation of the machine.*

N1405 DELTV (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm.

During tool radius compensation (G41, G42), when bypassing sharp corners, if the movement is smaller on both axes than the value set on DELTV parameter, the movement judged to be small will be contracted with the following one.



N1406 CircOver (L1...8, floating-point, channel)

Proportional factor. Value limit: 0...1.

It automatically overrides the programmed feed (F), during tool radius compensation (G41, G42), in case of internal machining of arcs, to the center of the tool radius:

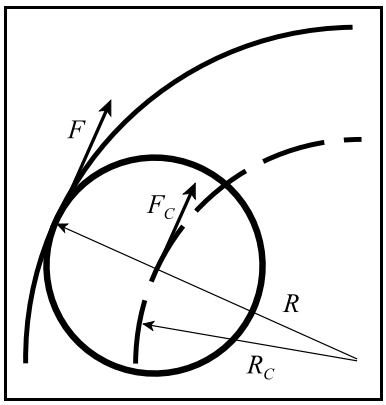
$$F_c = \frac{R_c}{R} F$$

where F_c: is the feed of the tool radius center (overridden feed)

R: is the programmed radius of arc

R_c: is the compensated radius of arc

F: is the programmed feed.

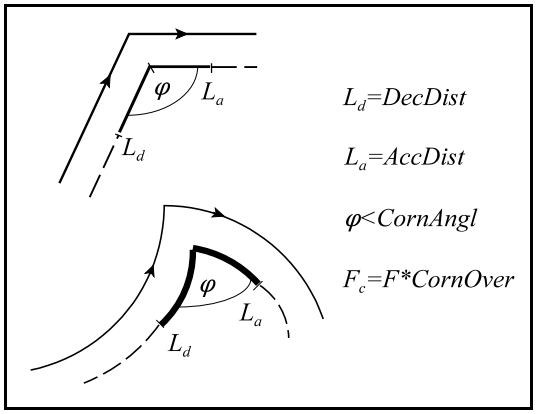


The automatic feedrate reduction has a lower limit specified by the *CircOver* parameter, where one can set by percentages the minimum of the feed reduction, i.e. for the current feed the condition $F_c \geq F^*CircOver$ will apply.

N1407 DecDist (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm.

If the corner override function is enabled, before the corner within the distance specified by DecDist paramter the corner override will be effective.



N1408 AccDist (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm.

If the corner override function is enabled, after the corner up to the distance specified by AccDist paramter the corner override will be effective.

N1409 CornAngle (L1...8, floating-point, channel)

Data of angle, unit ° (degrees).

During machining of internal corners, in state **G62** and in tool radius compensation mode (**G41, G42**) the corner overide function is **enabled** if the angle created by the corner is lower than the value set on CornAngl parameter. The value range of the parameter is 0–180°.

N1410 CornOver (L1...8, floating-point, channel)

Proportional factor. Value limit: 0...1.

The amount of corner override. The feedrate is F*CornOver, where F is the programmed feed.

N1411 Dominator Const (L1...8, floating-point, channel)

Not used.

N1412 No. of Common Tool Offsets T (egész)

A common parameter which is valid for all lathe channels.

It indicates the number of common compensation cells which can be referenced in all lathe channels.

In each channel the reference to a compensation cell ranging from 1 to the value set here either made from a program on address T or viewed on the offset table, will show the common offset values.

N1413 No. of Digits of Offs. No. in T Code (L1...8, egész, channel)

A parameter valid in lathe channels. Value set: 0, 1, 2, 3.

It specifies the number of digits in T code ending portion that refers to the offset number (number of a compensation cell). If e.g. the parameter value is 2, the interpretation of T code is:

Txxxxxxyy

where: xxxxxx is the tool number
yy is the offset number.

N1414 Comp. Config on Lathes (L1...8, bit, channel)

N1414 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
	RTL	**TOM**	**TCR**	**SGC**	**ZCG**	**QGW**	**GTN**	**TCM**

#0 **TCM**: In a lathe channel, it determines the way how to select the offset number (length and radius):
=0: selection of the offset number is made by T code,
=1: selection of the offset number is made by G code: Gxx Dnn

#1 **GTN**: In a lathe channel, if the selection of the offset number is made under address T (TCM=0), and the parameter value is:
=0: the offset number will select both the geometry and wear compensation value,
=1: the tool number will select the geometry and the offset number the wear compensation value.

☞ ***Example***:

Let's assume the last two digits of T code refer to the offset number: No. of Digits of Offs. No. in T Code=2. The T code is:

T0315

Tool number 3 is passed to the PLC in register CN_TC.
In case of GTN=0 the offset value will be the sum of the geometry value No. 15 and the wear value No. 15.
In case of GTN=1 the offset value will be the sum of the geometry value No. 03 and the wear value No. 15.

#2 **QGW**: In a lathe channel, if the selection of offset number is made on address T (TCM=0), and the tool number selects the geometry offset number (GTN=1) if the parameter value is:
=0: the imaginary tool nose number (Q) is specified by the wear offset number,
=1: the imaginary tool nose number (Q) is specified by the geometry offset number.

☞ ***Example***:

Let's assume the last two digits of code T refer to the offset number: No. of Digits of Offs. No. in T Code parameter value=2, and the tool number selects the geometry offset value: GTN=1. The T code is:

T0315

In case of QGW=0 the imaginary tool nose number Q is selected by the wear offset number No. 15, while
In case of QGW=1 the imaginary tool nose number Q is selected by the geometry offset number No. 03.

#3 **ZCG**: In a lathe channel, if the selection of offset number is made on address T (TCM=0), and the tool number selects the geometry offset number (GTN=1) if the parameter value is:
=0: the offset number 0 does not delete the geometry, but only the wear compensation,
=1: the offset number 0 will delete both the geometry and wear compensation.

#4 **SGC**: In a lathe channel, if the parameter value is:
=0: the second geometry offset table is not used, and will not be displayed,
=1: the second geometry offset table is used, will be displayed and the use of offset values is controlled by PLC flags.
If the CP_SGOEN (Second Geometry Offset ENable) flag, is
=0 only the first geometry offset is used,
=1 both the first and the second geometry offset is used and the compensation value is obtained by adding the first and second offset values.
CP_SGOX, CP_SGOY, CP_SGOZ PLC flags select whether to add the second geometry offset value to the first one on the appropriate axis or not:
=0 does not add the second geometry offset,
=1 adds the second geometry offset
on the given axis to the first geometry offset value if CP_SGOEN=1.

#5 **TCR**: on lathe controls, the tool length offset selected on address T:
=0: will not be deleted upon reset,
=1: it will be deleted.

#6 **TOM**: In a lathe control (channel), if TCM=1, that is selection of the offset number is made by G code and the value of parameter bit TOM:
=0: it is impossible to use the milling machine compensation table and to call it from a part program,
=1: it is possible to use the milling machine compensation table and to call it from a part program.

#7 **RTL**: In the lathe channel (control), and
in the Milling channel (control) when lathe compensation was called using G43.7 function,
if the value of the parameter is:
=0: the misalignment compensation applied in the workpiece coordinate systems will also rotate the axial components of the length compensations in the respective planes,
=1: the axial components of the length compensation will not be rotated but will be taken into account along the given axes.

N1415 Max. Amount of Wear Comp. (L1...8, floating-point, channel)
Positive number.
If the value of the parameter is 0, the maximum absolute value of the wear compensation will be 1 mm or 0.394 inch.
If the value of the parameter is greater than 0, the maximum absolute value of the wear compensation will be specified by the parameter.

N1416 Comp. Config on Mills (L1...8, bit, channel)

N1416 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
					RTM	**TOL**	**PLN**	**ZAX**

In the milling channel, compensation will be taken into account in accordance with the bits ZAX and PLN:

#0 **ZAX**:

=1: when the compensation is called (G43, G44), the value of the compensation (Hh) will always be registered for the axis Z, independently of the axis addresses programmed in the block.

#1 **PLN**:

=1: when the compensation is called (G43, G44), the value of the compensation (Hh) will always be registered for the axis perpendicular to the selected plane, independently of the axis addresses programmed in the block.

PLN	ZAX	Taking the length compensation into account in the G43, G44 block
0	0	for all the axes given in the block
0	1	always for the axis Z
1	0	in the case of G17: for the axis Z in the case of G18: for the axis Y in the case of G19: for the axis X
1	1	for all the axes given in the block

#2 **TOL**: In a milling machine control (channel), if the value of parameter bit TOL:

=0: it is impossible to use the lathe compensation table and to call it from a part program,

=1: it is possible to use the lathe compensation table and to call it from a part program.

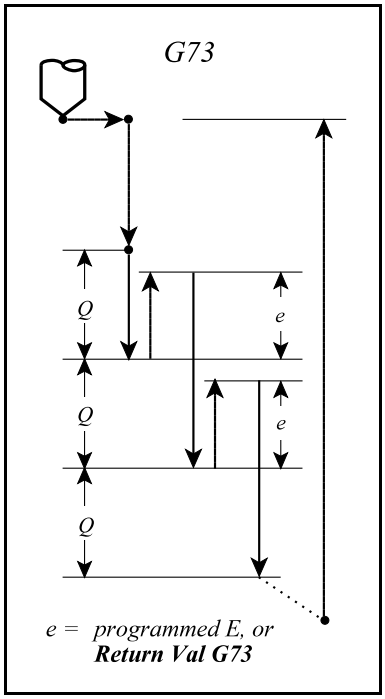
#3 **RTM**: it is not used.

16 Drilling Cycles Parameter Group

N1500 Return Val G73 (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm.

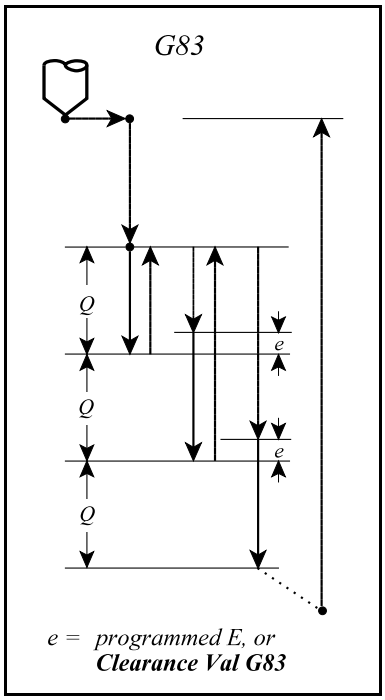
If address E is not specified in G73 (G83.1) high-speed peck drilling cycle, the control will take the return value of “e” from this parameter.



N1501 Clearance Val G83 (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm.

If address E is not specified in G83 peck drilling cycle, the control will take the clearance value of “e” from this parameter.



N1502 Extraction Override in G85, G89 (L1...8, floating-point, channel)

A ratio, the number written on the parameter shall be interpreted in %. Interpretation range: 0...200%.

In boring cycles G85, G89, it is the feedrate override value applied at tool extraction.

If no override values is specified on address “I” in G85/G89 boring cycles for the tool extraction feed, the control will take this parameter into consideration.

If the parameter value is 100, the feedrate of the extraction is value F applied in the cycle.

$F_{extraction} = F_{programmed}$ * Feed Override * Extraction Override in G85, G89/100

If the parameter value is 0, the control will take 100%.

N1503 Drilling Cycles Config. (L1...8, bit, channel)

N1503 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
		EAE	**ROE**	**EOE**	**PTC**	**RFF**	**TSC**	**CSM**

#0 **CSM**: The parameter is valid in milling channel. It determines the code of the high-speed peck drilling cycle, the left-handed tapping cycle and the fine boring cycle. If the value is:

=0: the above three cycles are specified by G73, G74, and G76 codes,

=1: the above three cycles are specified by G83.1, G84.1, and G86.1 codes.

In a lathe channel the control automatically uses codes G83.1, G84.1 and G86.1.

#1 **TSC**: The parameter will configure the rigid tapping cycles (G84.2, G84.3). If the parameter value is:

=0: before start of tapping, the spindle is not oriented, only the position control loop is closed. The code of the loop closure is determined by N0819 M Code for Closing S Loop parameter. This code will be passed to PLC.

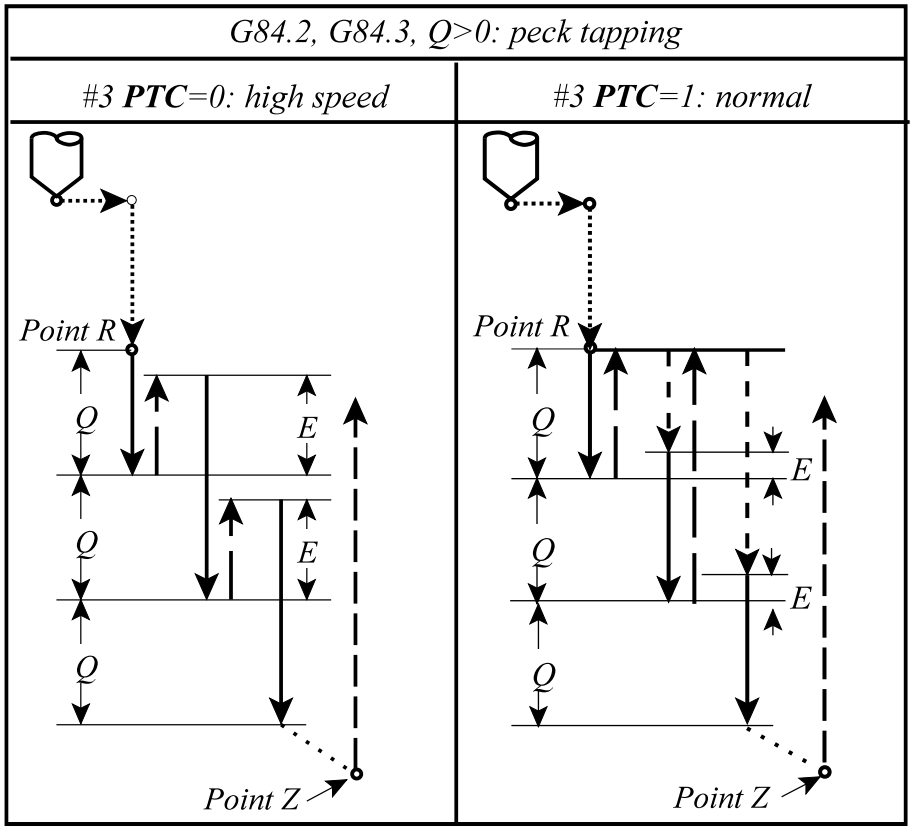
=1: before start of tapping the spindle will be oriented and command M19 will be passed to the PLC.

#2 **RFF**: The parameter will be valid in rigid tapping cycles (G84.2, G84.3). If the parameter value is:

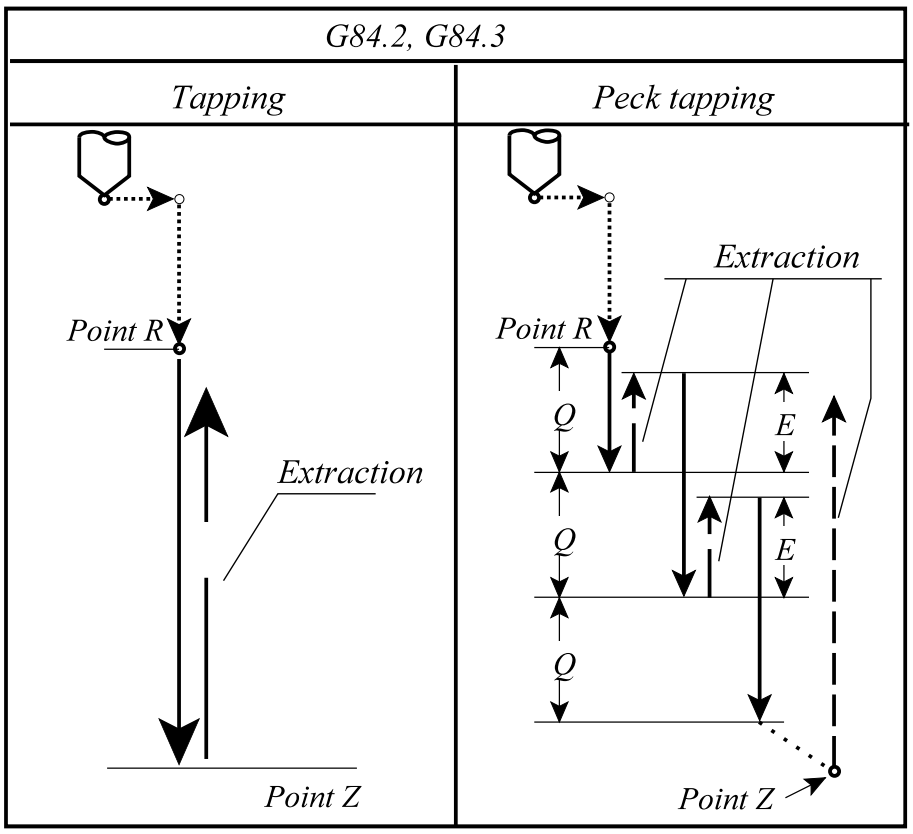
=0: in rigid tapping cycle the feedforward will be disabled even if the feedforward is enabled on the FFE bit of parameter N0515 Feed Forward Control.

=1: in rigid tapping cycle, the feedforward is enabled. The option of whether the feed-forward is operating or not, is determined by the FFE bit of parameter N0515 Feed Forward Control.

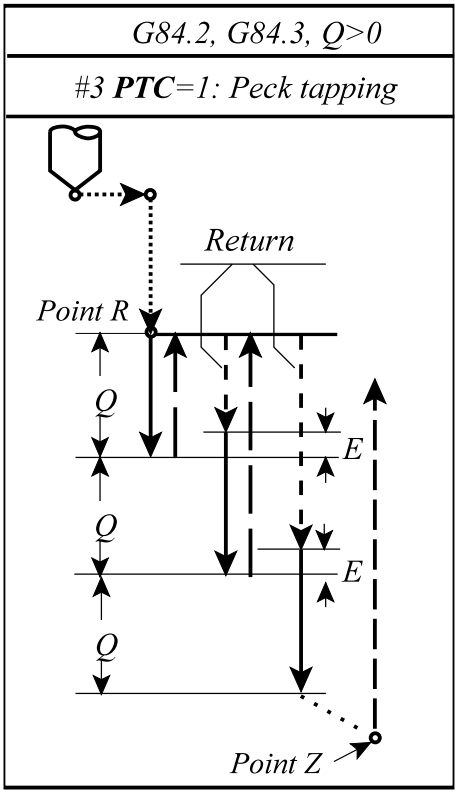
#3 **PTC**: The parameter will be valid in rigid tapping cycles (G84.2, G84.3) if peck tapping has been programmed on address **Q**. If the parameter value is:
=0: it applies high-speed peck tapping.
=1: it applies peck tapping.



#4 **EOE**: The parameter will be valid in rigid tapping cycles (G84.2, G84.3), when the tool is extracted. If the parameter value is:
=0: during extraction, the Extraction Override parameter will be ineffective for feed F,
=1: during extraction, the Extraction Override parameter will be effective.

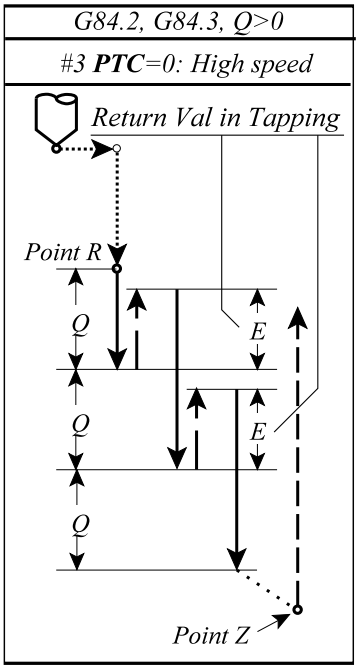


#5 **ROE**: The parameter will be valid in case of rigid peck tapping cycles (a value has been given to address Q) and with PTC=1 parameter.
It controls the return feedrate. If the parameter value is:
=0: during return, the Return Override parameter will be ineffective for feed F,
=1: during return, the Return Override parameter will be effective.



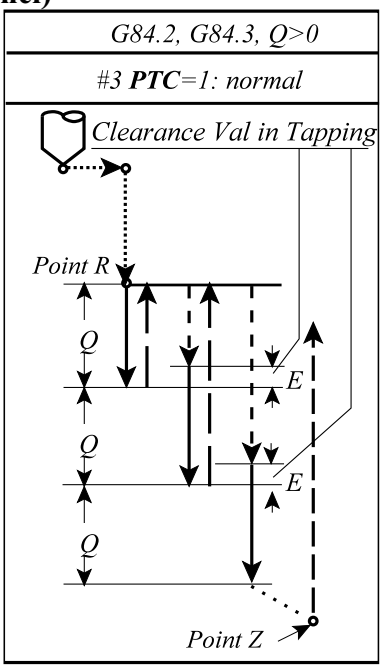
#6 **EAE**: The parameter will be valid in rigid tapping cycles (G84.2, G84.3), when the tool is extracted. If the parameter value:
=0: during extraction, the Extraction Acc parameter will be ineffective, and the acceleration will be made based on the Tapping Acc parameter,
=1: during extraction the acceleration will be made based on the Extraction Acc parameter.

N1504 Return Val in Tapping (L1...8, floating-point, channel)
Distance data, the number written on the parameter, based on parameter #0 IND shall be interpreted in inch or mm.
The parameter will be valid in case of high speed peck tapping cycles (a value has been given to address Q and with PTC=0 parameter).
If address E is not specified in the cycle, the control will take the return value of "e" from this parameter.



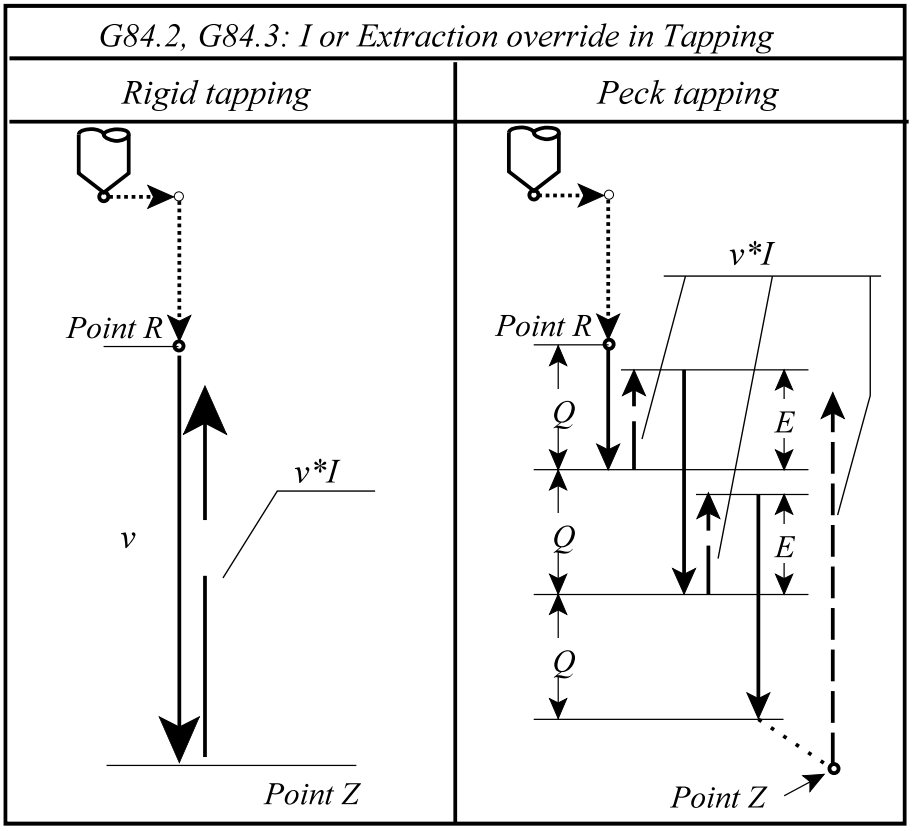
N1505 Clearance Val in Tapping (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on parameter #0 IND shall be interpreted in inch or mm.
The parameter will be valid in case of peck tapping cycles (a value has been given to address Q and with PTC=1 parameter).
If address E is not specified in the cycle, the control will take the clearance value of “e” from this parameter.



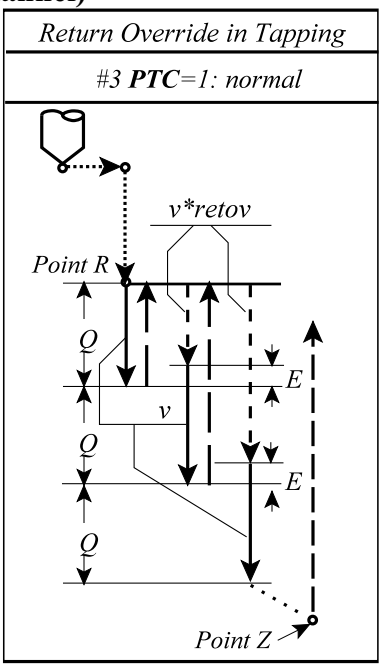
N1506 Extraction Override in Tapping (L1...8, floating-point, channel)

A ratio, the number written on the parameter shall be interpreted in %. Value range: 0...200%.
If no extraction override is pecified on address “I” in G84.2, G84.3 cycles, the override value will be taken from this parameter.
The override will be valid only in case of an EOE=1 parameter state, either peck tapping is applied or not and is applied to the feedrate specified in the cycle.
If the parameter value is 0, the control will take 100%.



N1507 Return Override in Tapping (L1...8, floating-point, channel)

A ratio, the number written on the parameter shall be interpreted in %.
Interpretation range: 0...200%.
The parameter will be valid in case of peck tapping cycles (a value has been given to address Q and with PTC=1 parameter).
It is the return override value in case if it is enabled with ROE=1 parameter state.
If the parameter value is 0, the control will take 100%.

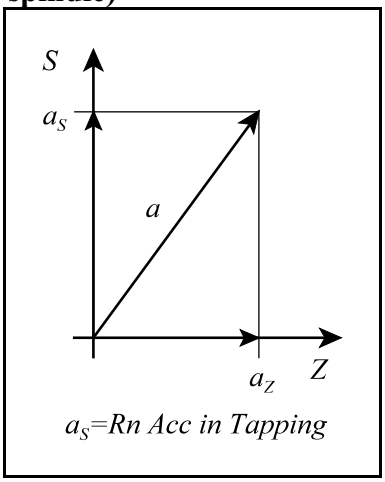


N1507+n Rn Acc in Tapping n=1...8 (S1...S16, floating-point, spindle)

n=1...8: different parameter values for each the range 1...8 of the spindle.

During rigid tapping, it is the acceleration component value falling on the spindle. The acceleration happens by an acceleration before interpolation. The control will take it into consideration according to the range set on the given spindle.

Unit: 1/sec^2.



N1515+n Rn Acc Tc in Tapping n=1...8 (S1...S16, floating-point, spindle)

n=1...8: different parameter values for each the range 1...8 of the spindle

During rigid tapping it is the value of the acceleration time constant falling on the spindle, per range.

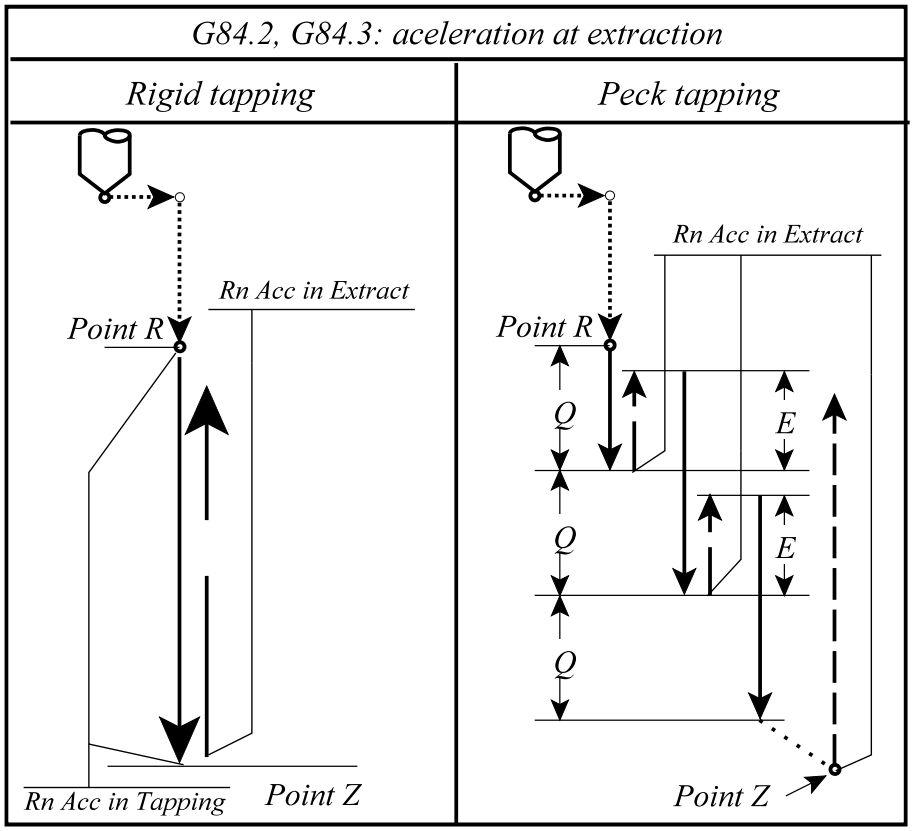
Time data, specified in seconds (sec).

N1523+n Rn Acc in Extract n=1...8 (S1...S16, floating-point, spindle)

n=1...8: different parameters for every item of the range 1...8

During rigid tapping it is the value of the acceleration component falling on the spindle, when starting the extraction. The control will take it into consideration if it is enabled with EAE=1 parameter state. Advisably it is lower than the Rn Acc in Tapping Value, in order to let more torque for the motor for breaking the chip. The control will take it into consideration according to the range exchanged on the given spindle.

Unit: $1/sec^2$.

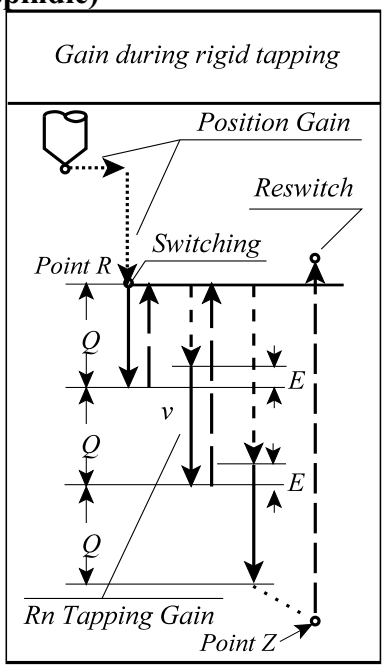


N1531+n Rn Tapping Gain n=1...8 (S1...S16, floating-point, spindle)

n=1...8: different parameters for every item of the range 1...8

During rigid tapping it is the gain value valid for the spindle and the axis participating in tapping (in case of a tilted tool, for axes). The control will take it into consideration according to the range set on the given spindle.

Measurement unit: 1/sec.

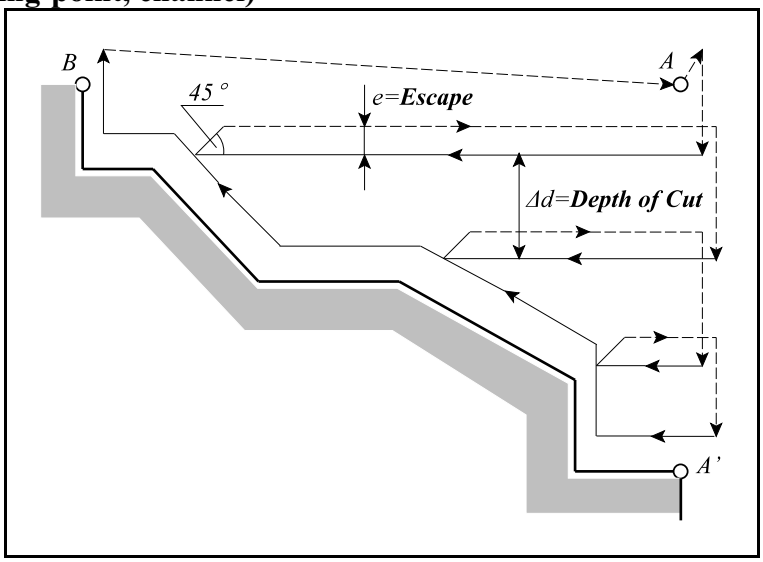


☞ ***Warning!***

The axis- and spindle handler will automatically recalculate the Inpos, Rn S Inpos, Serrl3 and Rn S Serrl3 parameters, based on the Rn Tapping Gain parameter.

17 Turning Cycles Parameter Group

N1600 Depth of Cut (L1...8, floating-point, channel)



Distance data, the number written on the parameter, based on parameter #0 IND shall be interpreted in inch or mm.

In cycles stock removing in turning G71 and stock removal in facing G72 it is the depth of cut (Δd). It is always interpreted in radius. The parameter value will be used if if in program block

G71 U(Δd) R,

or the

G72 W(Δd) R

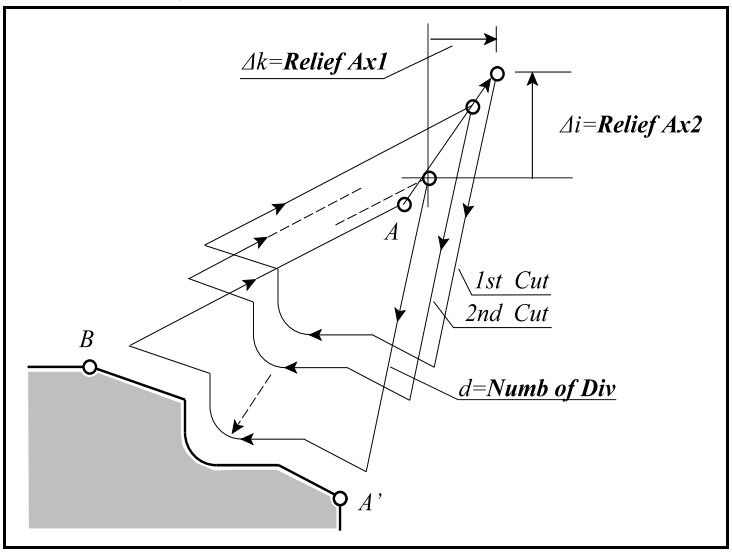
Δd value is not specified on U, or W address.

N1601 Escape (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on parameter #0 IND shall be interpreted in inch or mm.

In cycles stock removing in turning G71 and stock removal in facing G72 it is the escaping amount (e). It is always interpreted in radius. The parameter value will be used if in program block G71 U R(e), or G72 W R(e) value e is not specified on address R.

N1602 Relief Ax1 (L1...8, floating-point, channel)



Distance data, the number written on the parameter, based on parameter #0 IND shall be interpreted in inch or mm.

It is the distance and direction of relief on ***axis 1 of the selected plane*** in pattern repeating cycle G73. The parameter value will be used if in program block

G73 U W(Δk) R

relief value is not specified on address W in case of G18 plane.

It is always interpreted in radius.

N1603 Relief Ax2 (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on parameter #0 IND shall be interpreted in inch or mm.

It is the distance and direction of relief on ***axis 2 of the selected plane*** in pattern repeating cycle G73.

The parameter value will be used if in program block

G73 U(Δi) W R

relief value is not specified on address U in case of G18 plane.

It is always interpreted in radius.

N1604 Numb of Div (L1...8, integer, channel)

A number.

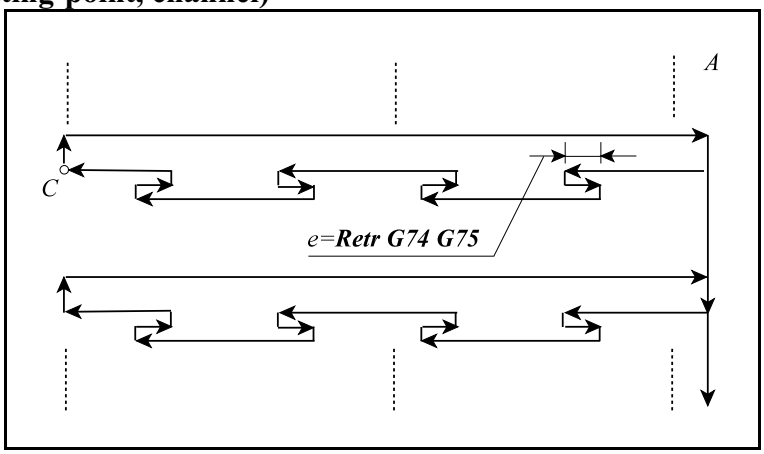
It is the number of division in pattern repeating cycle G73.

The parameter value will be used if in program block

G73 U W R(d)

division value d is not specified on address R.

N1605 Retr G74 G75 (L1...8, floating-point, channel)



Distance data, the number written on the parameter, based on parameter #0 IND shall be interpreted in inch or mm.

It is the return amount in end face and outer/internal diameter drilling cycles of types G74 and G75.

The parameter value will be used if in program block

G74 R(e), or

G75 R(e)

return amount is not specified on address R.

It is always interpreted in radius.

N1606 ThrdChmfr (L1...8, integer, channel)

A ratio.

In G76 and G78 thread cutting cycles it is the amount of chamfer at the end of the thread. The amount of chamfer is:

ThrdChmfr·L/10

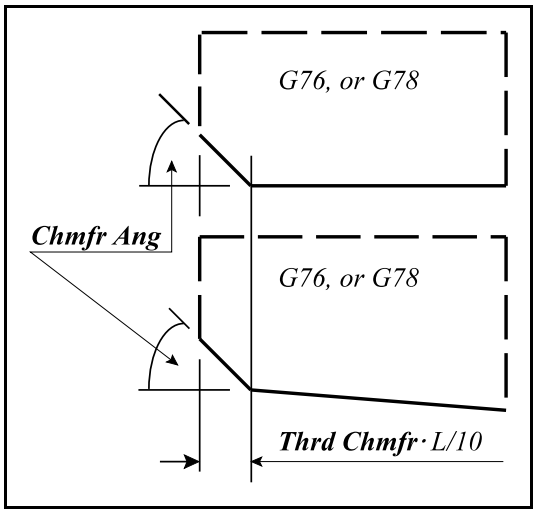
where: L is the programmed thread lead.

If ThrdChmfr=15 and the programmed thread lead is 1 mm, then the amount of chamfer will be 1.5 mm.

If in program block

G76 P(n)(r)(α) Q R

value r is not specified on address P_r_ the parameter value will be applied.



N1607 Chmfr Ang (L1...8, floating-point, channel)

Data of angle, Unit ° (degree). Value range: 0°...89°.

It sets the angle of the chamfer at the end of G76 and G78 thread cutting cycles. If the parameter value is 0, the chamfering angle will be 45°.

N1608 Count Fin (L1...8, integer, channel)

A number.

It is the repetitive count of finishing in G76 thread cutting cycle.

If in program block

G76 P(n)(r)(α) Q R

value n is not specified on address Pn__ the parameter value will be applied.

N1609 Min Thrd Cut (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on parameter #0 IND shall be interpreted in inch or mm.

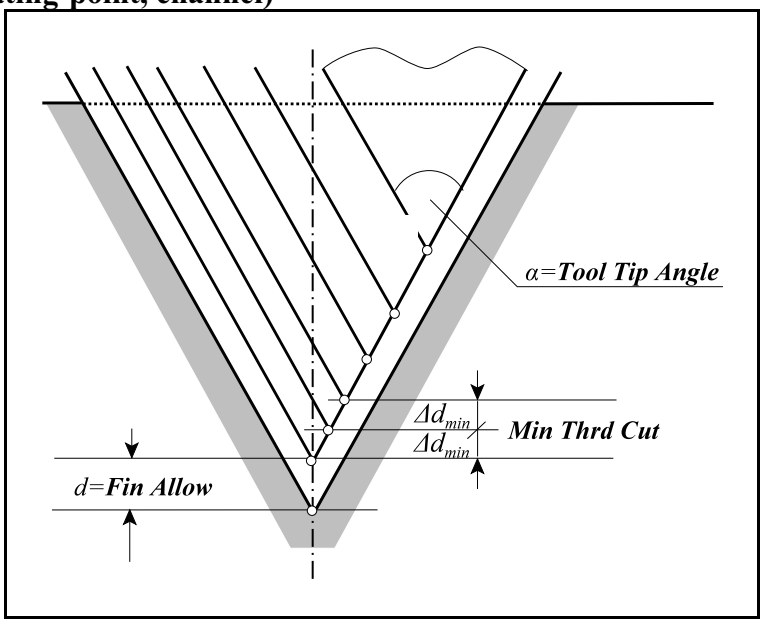
In G76 thread cutting cycle it is the amount of minimum cutting depth.

If in program block

G76 P Q(Δd_{min}) R

value Δd_{min} is not specified on address Q the parameter value will be applied.

It is always interpreted in radius.



N1610 Fin Allow (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on parameter #0 IND shall be interpreted in inch or mm.

In G76 thread cutting cycle it is the amount of finishing allowance.

If in program block

G76 P Q R(d)

value d is not specified on address R the parameter value will be applied.

It is always interpreted in radius.

N1611 Turning Cyc. Config. (L1...8, bit, channel)

N1611 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
					FCK	**FPT**	**SKP**	**TMI**

#0 **TMI**: The parameter will be effective only in a milling channel. If it

=0: no turning cycles are used,

=1: turning cycles are used in the milling channel.

If turning cycles are used, the codes of the below drilling cycles will be the following:

Instead of G73: G83.1 (G183)

Instead of G74: G84.1 (G184)

Instead of G76: G86.1 (G186)

#1 **SKP**: During the execution of multiple repetitive cycles G71, G72, G73, the program is continued if the parameter:

=0: by the block following the instruction G71, G72 or G73,

=1: by the block following the block specified on address Q in the instruction G71, G72 or G73.

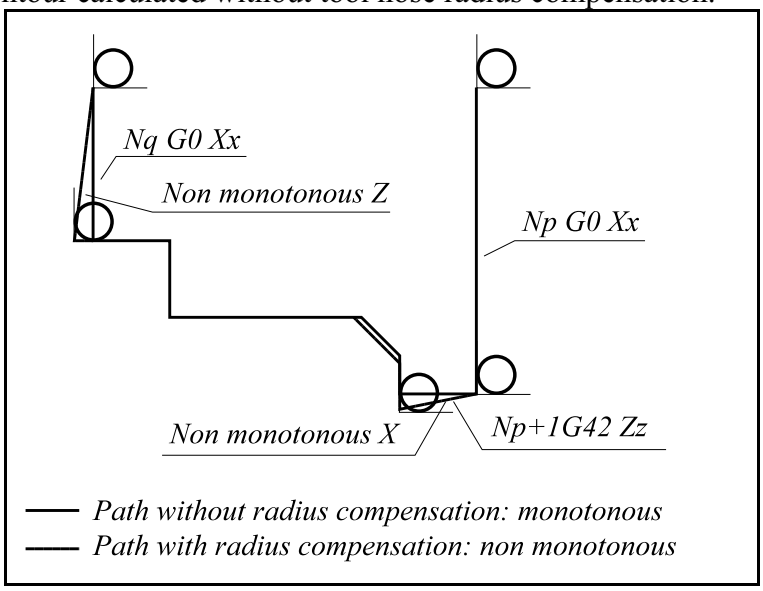
#2 **FPT**: In G71, G72 stock removal cycles, in case there are more pockets on the finishing shape, if the parameter is:

=0: the pockets are cut in the reverse order programmed in the finishing shape,

=1: the pockets are cut in the order programmed in the finishing shape.

#3 **FCK**: In G71, G72 stock removal cycles the contour specified by addresses P and Q is checked for monotony. If the parameter value is
=0: check is done for the contour calculated with tool nose radius compensation,
=1: check is done for the contour calculated without tool nose radius compensation.

On the figure beside can be seen that with parameter setting FCK=0 the control sends alarm messages "The contour not monotonous X, Z", as the compensated path is not monotonous in blocks switching on and off the compensation.
With parameter setting FCK=1 monotony is checked for the uncompensated path so it does not send any error message in blocks switching on and off the compensation.



N1612 Tool Tip Angle (L1...8, floating-point, channel)
Data of angle, unit ° (degrees).
It is the angle of tool tip used in G76 thread cutting cycle.
If in program block
G76 P__(α) Q R
value α is not specified on address P__(α) the parameter value will be applied.

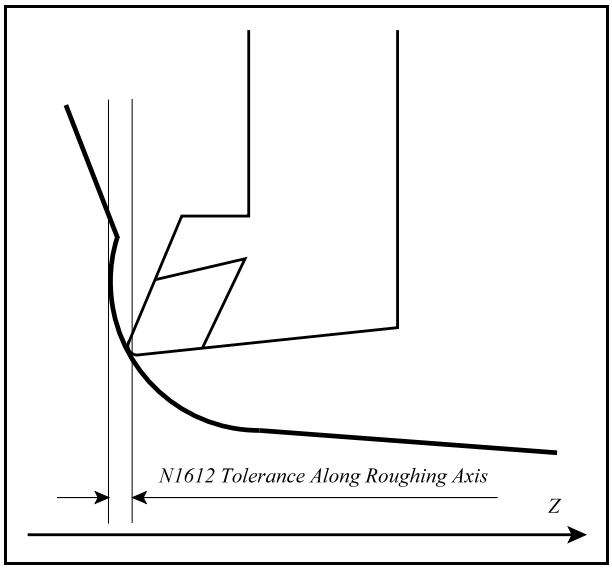
N1613 Tolerance Along Roughing Axis (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on parameter #0 IND shall be interpreted in inch or mm. It is a positive value.

The parameter is considered in stock removal cycles **G71** and **G72** at both type I and type II selections.

In the course of stock removal the control checks the monotony of contour along the roughing axis. If the size of non-monotony is smaller than the value set on this parameter, the control executes the roughing, otherwise it sends alarm messages “The contour not monotonous”.

Let’s suppose the roughing is done along axis Z. The arc programmed between two straight lines has a small vertex in direction Z, e.g., due to some inaccuracy of calculation. If the size of the vertex is smaller than the value set on this parameter, the control doesn’t send error message otherwise it does.



N1614 Tolerance Along Cutting Axis (L1...8, floating-point, channel)

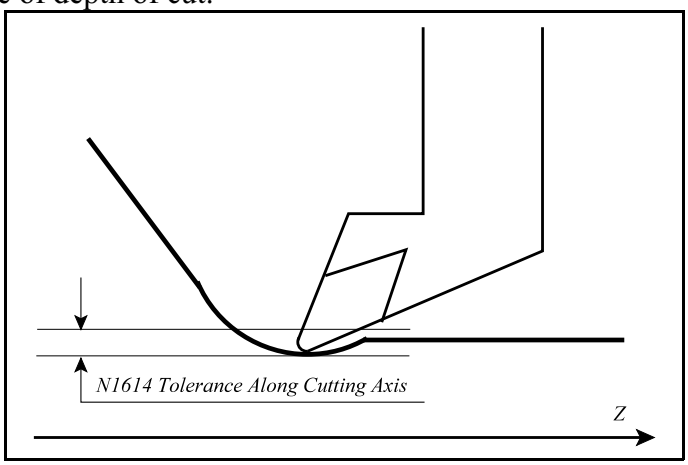
Distance data, the number written on the parameter, based on parameter #0 IND shall be interpreted in inch or mm. It is a positive value.

The parameter is considered in stock removal cycles **G71** and **G72** at type I selection.

In the course of stock removal the control checks the monotony of contour along the cutting axis. If the size of non-monotony is smaller than the value set on this parameter, the control executes the roughing, otherwise it sends alarm messages “The contour not monotonous”.

If the depth of cut (Δd) specified in the cycle is lower than the value of this parameter monotony is checked for the value of depth of cut.

Let’s suppose the roughing of type I is done along axis Z. The arc programmed between two straight lines has a small vertex in direction X, e.g., due to some inaccuracy of calculation. If the size of the vertex is smaller than the value set on this parameter, and the value of the depth of cut the control doesn’t send error message otherwise it does.



18 Macro Parameter Group

N1700 No. of Common #100 (integer)

It is a common parameter valid for all channels.
It determines the number of custom macro variables between #100 and #499, which can be referred to in each channel.
In every channel, the macro variables ranging from 100 to 100 + No. of Common #100 parameter will be common.
This number shall be lower than 400.
If the parameter value is e.g. 40, the macro variables ranging from #100 to #139 will be common for all channels.

N1701 No. Of Common #500 (integer)

It is a common parameter valid for all channels.
It determines the number of custom macro variables between #500 and #999, which can be referred to in each channel.
In every channel, the macro variables with numbers ranging from 500 to 500 + No. of Common #500 parameter will be common.
This number shall be lower than 500.
If the parameter value is e.g. 30, the macro variables ranging from #500 to #529 will be common for all channels.

N1702 Write Prt Low (L1...8, integer, channel)
N1703 Write Prt Hig (L1...8, integer, channel)

Any block of the #500 macro variables can be made read-only by using the above parameters. The starting address of the block is to be written on the Write Prt Low, and the ending address of the block is to be written on the Write Prt Hig parameter. If e.g. Write Prt Low=720, and Write Prt Hig=745, then the macro variables ranging from #720 to #745 will be protected aginst writing.

N1704+n G(901n) n=0...9 (L1...8, integer, channel)

10 parameters can be set with the codes of G functions, which call custom macros with program numbers between O9010, O9011, ..., O9019.
Data values ranging from 1000 to (+)1000 (–1000, –999, –1, 0, 1, 999, 1000) can be written on the parameter.
If a **negative** number is specified on the parameter the macro call will be **modal**.
Whether the macro modal call shall be equivalent to G66, or G66.1 will be decided by parameter **MEQ**.
If **1000, or –1000** is written on the parameter, the appropriate program number will be called by the code **G0**. If 0 is written on the appropriate parameter, that program number will not be called by a G code.

N1714 Start G Macro (L1...8, integer, channel)
N1715 Start Prg No (L1...8, integer, channel)
N1716 No. of G Codes (L1...8, integer, channel)

On the above three parameters a group of G codes without a decimal point may be specified for calling a custom macro.
On the Start G Macro parameter, the number of the beginning G code shall be written. If the beginning code of the group is e.g. G200, the parameter will be filled in the following way: Start G Macro=200.
The program number called by G code specified on parameter Start G Macro is to be specified on Start Prg No parameter. If the G200 code calls the O3400 macro program, then Start Prg No=3400.
Parameter No. of G Codes specifies the number of G codes belonging to the group. If there are 50 codes in the group, then No. of G Codes=50.
After this, the compliance between G codes and program numbers:

1. G code	G200	➜	1. program number	O3400
2. G code	G201	➜	2. program number	O3401
3. G code	G202	➜	3. program number	O3402
.....				
50. G code	G249	➜	50. program number	O3449

If a negative number is specified on the parameter Start G Macro the macro call for the whole group will be **modal**. Whether the macro modal call shall be equivalent to G66, or G66.1 will be decided by parameter **MEQ**.
If No. of G Codes=0 no macro will be called to these G codes

N1717 Start Dec G Macro (L1...8, floating-point, channel)
N1718 Start Prg No. Dec G (L1...8, integer, channel)
N1719 No. of Dec G Codes (L1...8, integer, channel)

On the above three parameters a group of G codes with a decimal point may be specified for calling a custom macro.
On the Start Dec G Macro parameter, the number of the beginning G code shall be written. If the beginning code of the group is e.g. G310.5 the parameter will be filled in the following way: Start Dec G Macro=310.5.
The program number called by G code specified on parameter Start Dec G Macro is to be specified on Start Prg No. Dec parameter. If the G310.5 code calls the O4000 macro program, then Start Prg No. Dec=4000.
After this, the compliance between G codes and program numbers is:

1. G code	G310.5	➜	1. program number	O4000
2. G code	G310.6	➜	2. program number	O4001
3. G code	G310.7	➜	3. program number	O4002
.....				
10. G code	G311.4	➜	10. program number	O4009

If a negative number is specified on the parameter Start Dec G Macro the macro call for the whole group will be **modal**. Whether the macro modal call shall be equivalent to G66, or G66.1 will be decided by parameter **MEQ**.
If No. of Dec G Codes=0 no macro will be called for these G codes.

N1720+n M(900n) n=0...9 (L1...8, integer, channel)

10 parameters can be set with the codes of M functions, which call subprograms with program numbers between O9000, O9001, ..., O9009.
For example; if the M(9000) parameter value is 6, command M06 will call subprogram O9000.

N1730 Start M SubP (L1...8, integer, channel)
N1731 Start Prg No. M SubP (L1...8, integer, channel)
N1732 No. of M Codes (L1...8, integer, channel)

On the above three parameters a group of M codes can be specified for calling **subprograms**.
On the Start M SubP parameter, the number of the beginning M code shall be written. If the beginning code of the group is e.g. M70000, the parameter will be filled in the following way: Start M SubP=700000.
The program number called by M code specified on parameter Start M SubP is to be specified on Start Prg No. M SubP parameter. If the M700000 code calls the O8000 program, then Start Prg No. M SubP=8000.
Parameter No. of M Codes specifies the number of M codes belonging to the group. If there are 30 codes in the group, then No. of M Codes=30.
After this, the compliance between the M codes and program numbers will be as follows:

1. M code	M700000	➜	1. program number	O8000
2. M code	M700001	➜	2. program number	O8001
3. M code	M700002	➜	3. program number	O8002
.....				
30. M code	M700029	➜	30. program number	O8029

If No. of M Codes=0 no subprogram will be called for these M codes.

N1733+n M(902n) n=0...9 (L1...8, integer, channel)

10 parameters can be set with the codes of M functions, which call custom macros with program numbers between O9020, O9021, ..., O9029.
For example; if the M(9020) parameter value is 8, then command M08 will call macro program O9020.

☞ ***Note:*** *The macro call by M code will pass all addresses of the block as arguments to the body of the macro, via the macro variables #1, ... #33. Therefore, if we do not need argument passing, we have to appoint function M for calling subprograms, otherwise an unexpected operation may occur. See also: parameters M(900n).*

N1743 Start M Macro (L1...8, integer, channel)
N1744 Start Prg No. M Macro (L1...8, integer, channel)
N1745 No. of M Macro Codes (L1...8, integer, channel)

On the above three parameters a group of M codes may be specified for calling a custom macro.
On the Start M Macro parameter, the number of the beginning M code shall be written. If the beginning code of the group is e.g. M500, the parameter will be filled in the following way: Start M Macro=500.

The program number called by M code specified on parameter Start M Macro is to be specified on Start Prg No. M Macro parameter. If the M500 code calls the O7000 macro program, then Start Prg No. M Macro=7000.

Parameter No. of M Macro Codes specifies the number of M codes belonging to the group. If there are 12 codes in the group, then No. of M Macro Codes=12.

After this, the compliance between the M codes and program numbers will be as follows:

1. M code M500 ➜ 1. program number O8000
2. M code M501 ➜ 2. program number O8001
3. M code M502 ➜ 3. program number O8002

.....

12. M code M511 ➜ 12. program number O8011

If No. of M Macro Codes=0, no macro will be called to these M codes.

N1746 ABCST (L1...8, bit, channel)

N1746 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
				TM	**SM**	**CM**	**BM**	**AM**

#0 **AM**: Specifies whether to call subprogram with auxiliary function code “A”
=0: no subprogram is called,
=1: subprogram O9030 is called, in which the value of address “A” can be read from the variable #195.

#1 **BM**: Specifies whether to call subprogram with auxiliary function code “B”
=0: no subprogram is called,
=1: subprogram O9031 is called in which the value of address “B” can be read from the variable #196.

#2 **CM**: Specifies whether to call subprogram with auxiliary function code “C”
=0: no subprogram is called,
=1: subprogram O9032 is called in which the value of address “C” can be read from the variable #197.

#3 **SM**: Specifies whether to call subprogram with function code “S”
=0: no subprogram is called,
=1: subprogram O9033 is called in which the value of address “S” can be read from the variable #198.

#4 **MT**: Specifies whether to call subprogram with function code “T”
=0: no subprogram is called,
=1: subprogram O9034 is called in which the value of address “T” can be read from the variable #199.

N1746+n ASCII Code SubPn n=1, 2, 3, 4 (L1...8, integer, channel)
N1750+n Prg No. ASCII Calln n=1, 2, 3, 4 (L1...8, integer, channel)

On the above four parameters 4 different ASCII codes can be specified for calling **sub-programs**. Specifiable values:

D, E, F, H, I, J, K, Q, R, U, V, W, X, Y, Z

The program number called by ASCII code specified on parameter ASCII Code SubPn is to be specified on Prg No. ASCII Calln parameter.

For example: if ASCII Code SubP1=75 and Prg No. ASCII Call1=1500, then address K calls the subprogram nr. O1500.

In case of ASCII Code SubPn=0 no subprogram is called.

The subprograms being called to the addresses specified in the N1747 ASCII Code SubP1, ..., N1750 ASCII Code SubP4 parameters will give the value of their addresses to the subprogram through the #191, ..., #194 variables.

N1755 Macro Contr (L1...8, bit, channel)

N1755 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
		TLC	**PCS**	**BUF**	**PRC**	**SBM**	**ENC**	**MEQ**

#0 **MEQ**: Specifies whether a G code modal call (with a negative sign on any of the G(901n), Start G Macro or Start Dec G Macro parameters) is made
=0: after each movement (equivalent to G66),
=1: for each block (equivalent to G66.1).

#1 **ENC**: When, in the body of the macro called by G code, there is reference to such M, S, T, A, B, C or other ASCII code that is assigned for subprogram call,
or, in the body of the subprogram called by M, S, T, A, B, C or other ASCII code there is reference to such G code that is assigned for macro call,
if the value of the parameter
=0: it will not initiate macro call,
=1: it will initiate macro call.

#2 **SBM**: in single block mode, on macro commands, if the parameter
=0: it will not stop
=1: it will stop and take up a STOP state.

#3 **PRC**: For the values of the #1-#33, #100-#499, #500-#599 macro variables to be displayed, if the variable:
=0: it will display them by the use of 15 decimal digits,
=1: it will round the values after the decimal point on the basis of the N1761 Precision Setting parameter, in which the number of the decimal digits after the decimal point can be specified.

#4 **BUF**: For the values of the #1100...#1131, #1132, #1133...#1135 mac ro variables, if the variable:
=0: it will not buffer the value of the entered variable, but the block interpreter will write the above PLC variables in such a way that it waits for emptying the buffer,
=1: the block interpreter will buffer the value of the variable, it will not wait for emptying the buffer, but the executor will write the above PLC variables.

#5 **PCS**: The parameter controlling the read-out of the instantaneous positions #5041...#5060 (#100101...#100150, #_ABSOT[n]) and the skip positions #5061...#5080 (#100151 ... #100200, #_ABSKP[n]). If any of the 3D rotations (G68.1, G68.2) is being turned on, the instantaneous positions and skip positions at the parameter value
=0: will be read back in the workpiece coordinate system,
=1: will be read back in the programmer's coordinate system, afterreverse rotation. In this case, the length compensation will be taken into account with reverse rotation too if TLC=0.

#5 **TLC**: The parameter controlling the read-out of the instantaneous positions #5041...#5060 (#100101...#100150, #_ABSOT[n]) and the skip positions #5061...#5080 (#100151 ... #100200, #_ABSKP[n]). If the value of the parameter is:
=0: the length compensation will be included in the position information,
=1: the length compensation components will be subtracted from the position information.

N1756 List Contr (L1...8, bit, channel)

N1756 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
							MD9	**MD8**

#0 **MD8**: The subprograms and macros with program numbers ranging from O8000 to O8999, if the parameter is
=0: will not be listed and in single block mode and it will not stop at commands being in the body of the subprogram or macro,
=1: will be listed and in single block mode and it will stop at commands being in the body of the subprogram or macro.

#1 **MD9**: The subprograms and macros with numbers ranging from O9000 to O9999, if the parameter is
=0: will not be listed and in single block mode and it will not stop at commands being in the body of the subprogram or macro,
=1: will be listed and in single block mode and it will stop at commands being in the body of the subprogram or macro.

N1757 Print Contr (L1...8, bit, channel)

N1757 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
								PNT

#0 **PNT**: During the execution of DPRNT macro command (decimal data output), if the parameter is
=0: at the place of the + sign and the leading zeros space codes will be issued,
=1: at the place of the + sign and the leading zeros nothing will not be issued.

Example:

```
DPRNT [ X#130 [53] Y#500 [53] T#10 [2] ]
    #130=35.897421  ——      35.897
    #500=–150.8     ——      –150.8
    #10=214.8       ——      15
```

Data output at **PNT**=0 state:

```
7 6 5 4 3 2 1 0
_____________________
1 1 0 1 1 0 0 0  --- X
1 0 1 0 0 0 0 0  --- space
1 0 1 0 0 0 0 0  --- space
1 0 1 0 0 0 0 0  --- space
1 0 1 0 0 0 0 0  --- space
0 0 1 1 0 0 1 1  --- 3
0 0 1 1 0 1 0 1  --- 5
0 0 1 0 1 1 1 0  --- decimal point (.)
1 0 1 1 1 0 0 0  --- 8
0 0 1 1 1 0 0 1  --- 9
1 0 1 1 0 1 1 1  --- 7
0 1 0 1 1 0 0 1  --- Y
0 0 1 0 1 1 0 1  --- negative sign (–)
1 0 1 0 0 0 0 0  --- space
1 0 1 0 0 0 0 0  --- space
1 0 1 1 0 0 0 1  --- 1
0 0 1 1 0 1 0 1  --- 5
0 0 1 1 0 0 0 0  --- 0
0 0 1 0 1 1 1 0  --- decimal point (.)
1 0 1 1 1 0 0 0  --- 8
0 0 1 1 0 0 0 0  --- 0
0 0 1 1 0 0 0 0  --- 0
1 1 0 1 0 1 0 0  --- T
1 0 1 0 0 0 0 0  --- space
1 0 1 1 0 0 0 1  --- 1
0 0 1 1 0 1 0 1  --- 5
0 0 0 0 1 0 1 0  --- line feed (LF)
```

Data output at **PNT**=1 state:

```
7 6 5 4 3 2 1 0
_____________________
1 1 0 1 1 0 0 0  --- X
0 0 1 1 0 0 1 1  --- 3
0 0 1 1 0 1 0 1  --- 5
0 0 1 0 1 1 1 0  --- decimal point  (.)
1 0 1 1 1 0 0 0  --- 8
0 0 1 1 1 0 0 1  --- 9
1 0 1 1 0 1 1 1  --- 7
0 1 0 1 1 0 0 1  --- Y
0 0 1 0 1 1 0 1  --- negative sign (–)
1 0 1 1 0 0 0 1  --- 1
0 0 1 1 0 1 0 1  --- 5
0 0 1 1 0 0 0 0  --- 0
0 0 1 0 1 1 1 0  --- decimal point  (.)
1 0 1 1 1 0 0 0  --- 8
0 0 1 1 0 0 0 0  --- 0
0 0 1 1 0 0 0 0  --- 0
1 1 0 1 0 1 0 0  --- T
1 0 1 1 0 0 0 1  --- 1
0 0 1 1 0 1 0 1  --- 5
0 0 0 0 1 0 1 0  --- line feed (LF)
```

N1758 Intrrt Contr (L1...8, bit, channel)

N1758 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
			SYM	**MCD**	**ELT**	**TPI**	**STP**	**USD**

#0 **USD**: the interrupt type custom macro
=0: is not used
=1: is used.

#1 **STP**: the interrupt type custom macro, from the aspect of use of local variables (#1, ...#33)
=0: is a macro type, i.e. uses its own local variables (increases the level of local variables),
=1: is a subprogram type, i.e. uses local variables of the calling program.

#2 **TPI**: If there is an interrupt, the macro call
=0: performed during the execution of the block,
=1: performed at the end of the block.

#3 **ELT**: The CP_MINT interruption signal (PLC flag)
=0: is level triggered, i.e. the interruption macro will be repeatedly called, till the signal is on,
=1: is edge triggered, i.e. the interruption macro will be called once for the 0 ➜1 transition of the signal.

#4 **MCD**: The custom macro interrupt enable/disable M codes
=0: are M96/M97,
=1: are the codes specified on M Code MI On, M Code MI Off parameter.

#5 **SYM**: it determines where the interruption macro has to be searched in the storage by the control. If its value:
=0: search will be executed in the folder that contains the main program even if the interruption macro was enabled not in the main program by the use of M96 P,
=1: search will always be executed in that directory of the SystemMacros folder belonging to the appropriate channel.

N1759 M Code MI On (L1...8, integer, channel)
N1760 M Code MI Off (L1...8, integer, channel)

If on the Intrrt Contr parameter #4 MCD=1, the
M code of the number specified on the M Code MI On parameter will enable, the
M code of the number specified on the M Code MI Off parameter will disable
the macro interruption.

N1761 Precision Setting (L1...8, integer, channel)

Value range: 0, ..., 15
If the #3 PRC bit of the N1755 Macro Contr parameter is 1, it will display the values of the #1-#33, #100-#499, #500-#599 macro variables having been rounded to the decimal digits, number of which is specified in the parameter. If, for example, Precision Setting=3, and
#1=12.65762
#2=102.467189,
it will display the values 12.658 and 102.467.

19 Electronic Gear Box Parameter Group

N1800 EGB Contr (L1...8, bit, channel)

N1800 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
						FHC	**FRS**	**RCE**

#0 **RCE**: If the electronic gear box (EGB) function G81.8 is on, upon reset button and CP_EGBRRQ PLC signal
=0: the synchronization will not be switched off,
=1: the synchronization will be switched off after tool retraction.

#1 **FRS**: If the electronic gear box (EGB) function G81.8 is on, in G95 state the feed will be calculated:
=0 based on the revolution of the master (hob),
=1 based on the revolution of the slave (cog), i.e. the pulses coming from the encoder of the master is multiplied with the quotient L/T (where L: number of the hob threads, T: number of cog teeth).

#2 **FHC**: on the axis carrying out the helical compensation in EGB mode the feed-forward=0: is disabled,
=1: is enabled.

N1801 EGB Master (L1...8, integer, channel)
It is the number of the EGB master spindle. Upon G81.8 command the slave axis specified on the EGB Slave parameter will be synchronized to the signals of this spindle.
Value range 1...16 spindle.

N1802 EGB Slave (L1...8, integer, channel)
It is the number of EGB slave axis. Upon G81.8 EGB command this axis will be synchronized to the spindle specified on EGB Master parameter.
Value range: axis 1...32.

N1803 Helical Comp. Axis (L1...8, integer, channel)
It is the number of axis carrying out the helical compensation programmable in G81.8 state.
Value range 1...32 axis.

N1804 Retr. Dist. (A1...32, floating-point, axis)
Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm, if #1 ROT=0.
If #1 ROT=1 the data will be interpreted in degrees.
In EGB mode, it determines the amount of tool retraction (incremental movement) per axis, with a sign. Those axes will participate in the extraction, the position data of which is not 0.

N1805 Retr. Feed (L1...8, floating-point, channel)

Speed data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch/min or mm/min, if #1 ROT=0.
If #1 ROT=1 the data will be interpreted in degree/min.
In G81.8 state it determines the speed of tool retraction.

N1806 Retr. Delay (L1...8, floating-point, channel)

Not used.

20 Chuck/Tailstock Barrier Parameter Group (T only lathe)

N1900 Chuck Config (L1...8, bit, channel)

N1900 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
								TYP

#0 **TYP**: Not used.

N1901 L1 Length of Jaws (L1...8, floating-point, channel)
Not used.

N1902 W1 Width of Jaws (L1...8, floating-point, channel)
Not used.

N1903 L2 Holding Length of Jaws (L1...8, floating-point, channel)
Not used.

N1904 W2 Holding Width of Jaws (L1...8, floating-point, channel)
Not used.

N1905 Chuck Position X (L1...8, floating-point, channel)
Not used.

N1906 Chuck Position Z (L1...8, floating-point, channel)
Not used.

N1907 L1 Length of Tailstock (L1...8, floating-point, channel)
Not used.

N1908 D1 Diameter of Tailstock (L1...8, floating-point, channel)
Not used.

N1909 L2 Length of Tailstock (L1...8, floating-point, channel)
Not used.

N1910 D2 Diameter of Tailstock (L1...8, floating-point, channel)
Not used.

N1911 L3 Length of Tailstock (L1...8, floating-point, channel)
Not used.

N1912 D3 Diameter of Tailstock (L1...8, floating-point, channel)
Not used.

N1913 D4 Diameter of Tailstock (L1...8, floating-point, channel)

Not used.

N1914 Tailstock Position Z (L1...8, floating-point, channel)

Not used.

21 Tool Holder Interference Parameter Group (T only lathe)

N2000 Check Config (L1...8, bit, channel)

N2000	#7	#6	#5	#4	#3	#2	#1	#0
					IFM	**T0**	**ZCE**	**IFE**

#0 **IFE**: Not used.
#1 **ZCE**: Not used.
#2 **T0**: Not used.
#3 **IFM**: Not used.

N2001 Coord. Type (L1...8, integer, channel)
Not used.

N2002 Tool Holder Position X (L1...8, floating-point, channel)
N2003 Tool Holder Position Z (L1...8, floating-point, channel)
Not used.

22 Axis Recomposition Parameter Group

N2100 Recomposition Config (T1...8, bit, machine group)

N2100 A1...32	#7	#6	#5	#4	#3	#2	#1	#0
							RSR	**OTR**

#0 **OTR**: Not used.
#1 **RSR**: Not used.

N2101 Synchronous Master (A1...32, integer, axis)

If the axis is a synchronous slave, the axis number of the master shall be written on this parameter. The master can be in the same channel, however, also in a different one. A master may have several slaves, too. A slave can be the master of another axis, too.
At synchronous control of axes, move command can be programmed or be issued manually (jog, manual handle) on the master axis only. The master, or even the slave axis may park, i.e. do not carry out any movement.
The synchronous control starts by setting the AP_SYNCR PLC flag, if the axis referred to is appointed a synchronous slave. From this on, the interpolator pulses of the master axis will be received also by the slave axis.

N2102 Synchron Config (A1...32, bit, axis)

N2102 A1...32	#7	#6	#5	#4	#3	#2	#1	#0
				PSN	**PKD**	**SYM**	**SER**	**MSY**

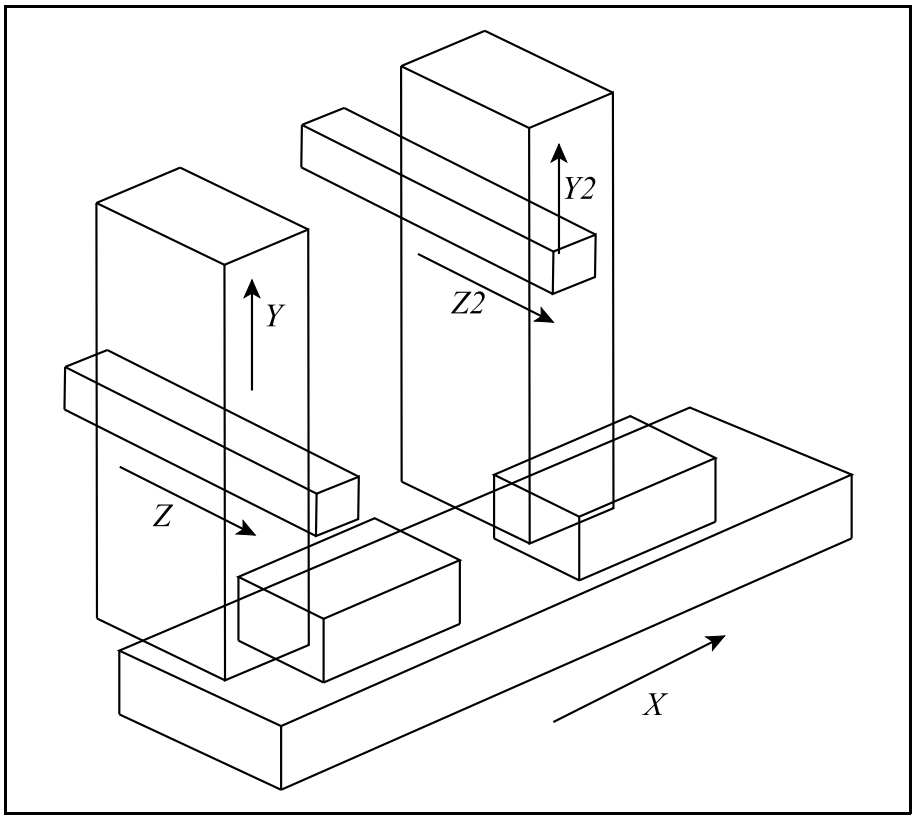
#0 **MSY**: If the axis is a synchronous slave synchronous mirror-image control is
=0: not applied: the master and slave move in the same direction
=1: applied: the master and slave move in the opposite direction.
#1 **SER**: Not used.
#2 **SYM**: If the given axis is a synchronous slave and, at the same time, also a synchronous master, this bit shall be written into 1.
#3 **PKD**: Not used.
#4 **PSN**: If the given axis is specified to synchronous slave on the N2101 Synchronous Master parameter, and the
– PSN parameter value 1:
– the control will always carry out a position synchronisation between the master and slave axes,
– independently of state of the AP_SYNCR PLC flag, immediately after switching on the machine,
– in a switched-on state of the servo, it will draw the position of the slave axis to the position of the master, by the gain determined on N0522 Synchr K parameter,
– it checks the position difference of the master and slave axis,
– in case it exceeds the value determined on the N2103 Synchron Error parameter, a synchron error alarm will be generated.

It serves for the synchronous positioning of gantry machines when on both sides seperate motors are mounted, together with drives and measuring system.

N2103 Synchron Error (A1...32, Floating-point, axis)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm, if #1 ROT=0.
If #1 ROT=1 the data will be interpreted in degrees.
If the given axis is specified to synchronous slave and assigned to position synchron at parameter PSN=1, the parameter specifies the allowable position difference between the master and slave axis.

Parallel machining on a multi-spindle machine:



Let's take a two-spindle milling machine, on which there is a 1-channel milling control. The names of axes are:

X, Y, Z, Y2, Z2

The below M codes are codes suppressing the buffer:

M41: Synchronous control on between Y - Y2 and Z - Z2

M40: Synchronous control off

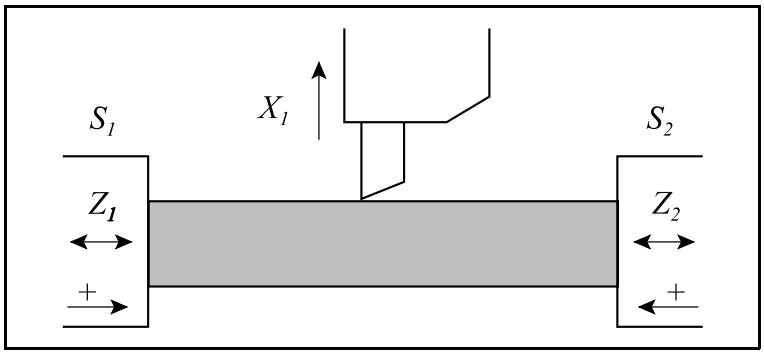
Let's take a situation in which there are two workpieces on the table and we would like to machine both of them simultaneously.

The part program will be the following:

```
....
G54 X0 Y0 Y2=0      (positioning in XY plane)
G43 Z10 H1          (approaching of the first workpiece by the first tool and its
                     compensation)
G43 Z2=10 H51       (approaching of the second workpiece by the second tool and its
                     compensation)
M41                 (synchronous control on)
G1 Z-5              (from this on, we will write the program by using the addresses X,
                     Y, Z until we switch off the synchronization.
G41 G1 X100 D1
G3 X0 Y100 R100
G1 Y0
...
...
M40                 (synchronous control off)
...
```

In the above example, the movement direction of axes Y and Y2, furthermore, Z and Z2 will be the same during their synchronous operation.

Clamping of the workpiece by two spindles



Let's take a two-channel lathe where both spindles move together with axis Z. A long workpiece needs to be machined, clamped on the right-side by spindle 2. The part program is written in channel 1, in a way to ensure that during the machining of the part, axis Z of channel 2 is in synchronous operation with axis Z of channel 1.
The positive directions of the master and the slave are different, therefore the movement direction of the slave is different from the movement direction of the master.
The below M codes are codes suppressing the buffer:

M41: Synchronous control on between Z_2 and Z_1
M40: Synchronous control off
M500...M599: Waiting M codes for the synchronization of channels.

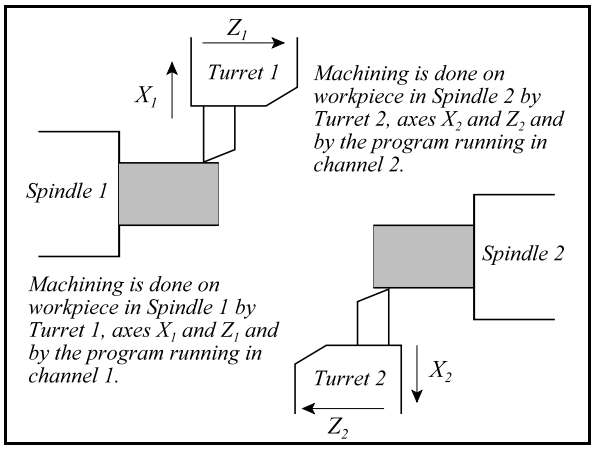
Draft of the machining program:

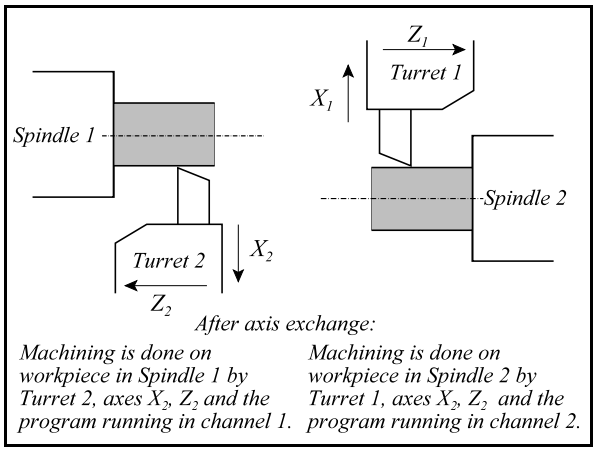
Program running in channel 1		**Program running in channel 2**	
....		....	
(machining in both channels)		(machining in both channels)	
...		...	
M500 P12	(waiting for channel 2)	M500 P12	(waiting for channel 1)
M41	(synchronous control on)	(channel 2 is waiting)	
M501 P12	(waiting for channel 2)	M501 P12	(waiting for channel 1)
...			
(machining in channel 1)		(channel 2 does not do anything)	
...			
M502 P12	(waiting for channel 2)	M502 P12	(waiting for channel 1)
M40	(synchronous control off)	(channel 2 is waiting)	
M503 P12	(waiting for channel 2)	M503 P12	(waiting for channel 1)
....		....	
(machining in both channels)		(machining in both channels)	

N2104 Composit Axis (A1...32, integer, axis)

The number of exchange axis is specified on the parameter of the axis to be exchanged. One of the exchange axis and the axis to be exchanged may be a hypothetic axis, too.

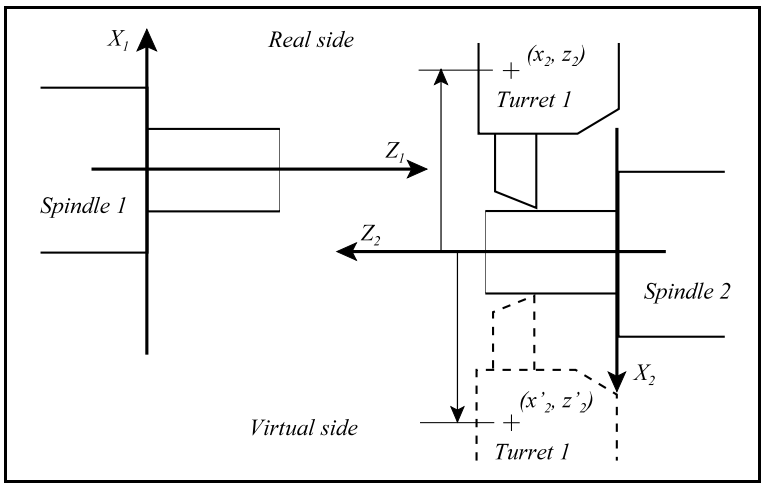
The axis exchange on the n-th axis starts by setting the AP_MIXR PLC flag. The Composit Axis parameter, belonging to the n-th axis will tell, with which axis shall the n-th axis be exchanged. After the control has carried out the necessary operations, it will confirm the exchange by setting the AN_MIXA flag of the n-th axis to be exchanged, to 1. During axis exchange, both axes can work in an other channel, too (not in the one for which it has been appointed by the Axis Assign parameter), it will receive the movement commands from there, and it will receive the work coordinate offsets from the original axis from the other channel. After the exchange, in both channels, it is possible to move the axes, except for cases where one of the axes is a hypothetic one. In case an axis is appointed a hypothetic one in a given channel, after the exchange, in the other channel, or in the same channel, no reference can be made to it.





N2105 Composit Config (A1...32, bit, axis)

N2105 A1...32	#7	#6	#5	#4	#3	#2	#1	#0
						MCO	**MDI**	**MMI**



#0 **MMI**: the parameters of both the exchange axis and the axis to be exchanged, shall be filled in. The direction of axes participating in the exchange, compared to each other, if the parameter is
=0: will be the same,
=1: will be opposite.

#1 **MDI**: Not used.

#2 **MCO**: the parameters of both the exchange axis and the axis to be exchanged, shall be filled in. For the axes participating in the exchange mirror image
=0: is not applied, the program will be written on the real side,
=1: is applied, the program will be written on the virtual side (as if Turret 2 would work).

In the picture we can see the workpiece being in channel 2, which we would like to machine after the axis exchange by the tools on axes X_1 and Z_1.
The real, absolute position of the tool in the coordinate system fixed to the workpiece:

$x_2 = -120$, $z_2 = 80$.

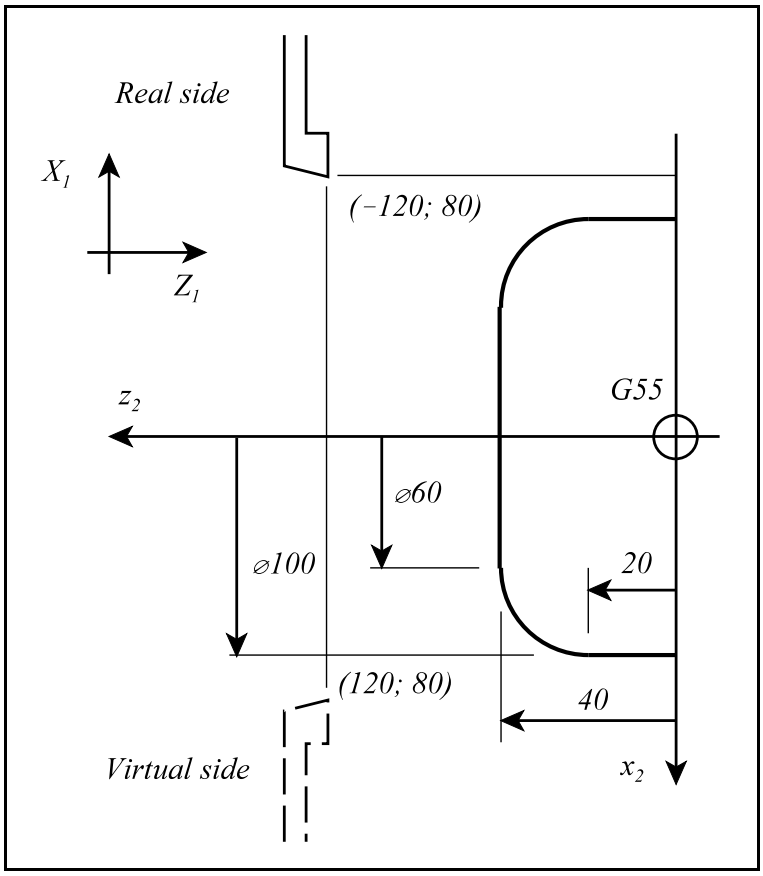
On the below two examples we are going to show how the program shall be written on the - **real side,** at

$MCO_{X2}=0$

parameter state, respectively, how it shall be written on the - **virtual side** at

$MCO_{X2}=1$

parameter state.
To write the program on the virtual side is like not having acknowledged the axis exchange, and if we worked by the original turret 2.



Real side, $MCO_{X2}=0$:
G0 X0 Z40
G1 X-60
G2 X-100 Z20 R20
G1 Z0
G0 X-120
Z80

Virtual side, $MCO_{X2}=1$:
G0 X0 Z40
G1 X60
G3 X100 Z20 R20
G1 Z0
G0 X120
Z80

The tool, of course, will move always on the real side.
In fact, the parameter state $MCO_{X2}=1$ is the mirror image to axis x_2, the center of which is the $x_2=0$ position in the workpiece coordinate system.

N2106 Composit Coordinates (A1...32, floating-point, axis)

Not used.

N2107 Superimposed Master (A1...32, integer, axis)

In case the movement of an axis (master) is to be added to the movement of another axis (slave) the axis number of the master is to be specified on the parameter of the slave. The master can be in the same channel, but also in a different one. A master may have several slaves. The master-slave relationship may create a chain: a slave may be the master of an other axis, then to this slave the pulses of two masters will be added.

A movement command may be issued at the same time both for the master and slave, whether they are in the same or in a different channel.
The superimposed movement starts with setting AP_SPRPNR PLC flag of the slave axis.
To the own movement of the slave axis, the movement of the master axis shall be added.
The movement will be superimposed till the AP_SPRPNR flag is reset.

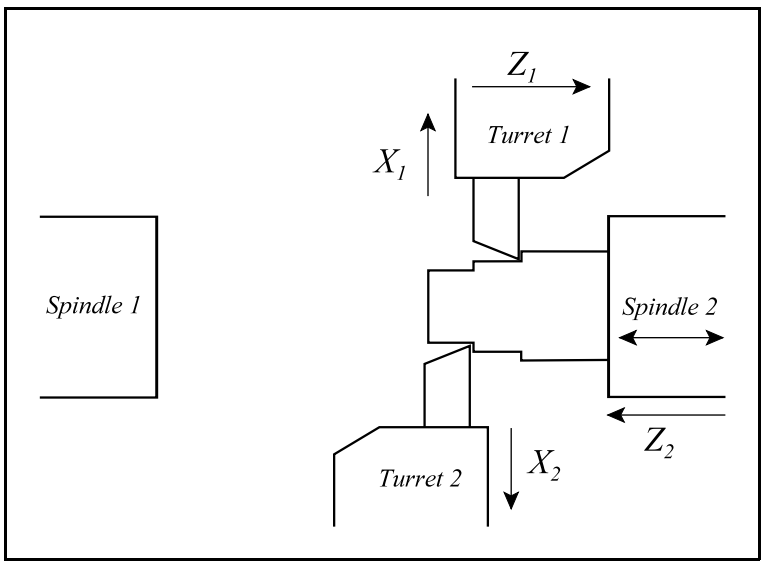
N2108 Superimposed Config (A1...32, bit, axis)

N2108 A1...32	#7	#6	#5	#4	#3	#2	#1	#0
						SUM	**SMR**	**MSU**

#0 **MSU**: If the axis is the slave of a superimposed movement and the MSU parameter is:
=0: the interpolator pulses of the master axis will be added to those of the slave,
=1: the interpolator pulses of the master axis will be subtracted from those of the slave.
#1 **SMR**: Not used.
#2 **SUM**: Not used.

An example of superimposed movement

The roughing of the workpiece is carried out by two tools. Turret 1 moves both in X_1, Z_1 directions.
Turret 2 carries out a movement only in the X_2 direction, and Spindle 2 moves in Z_2 direction. If we would like to work with both turrets, when the workpiece is in Spindle 2, the movement of axis Z_2 of channel 2 shall be superimposed to the movement of axis Z_1 of channel 1. Then axis Z_2 will be the master, and Z_1 the slave.
The positive direction of the master and slave are different, therefore the movement direction of the slave is opposite to that of the master (if the slave is staying, the master is moving).



The below M codes are codes suppressing the buffer:
M42: Superimposing of Z_2 to Z_1 on
M40: Superimposed control off

M500...M510: codes appointed on parameters for the synchronisation of channels.

Draft of the machining program:

Program running in channel 1		**Program running in channel 2**	
....		....	
(machining in both channels)		(machining in both channels)	
...		...	
M500 P12	(waiting for channel 2)	M500 P12	(waiting for channel 1)
M42	(Superimposed control on)	(this channel waits)	
M501 P12	(waiting for channel 2)	M501 P12	(waiting for channel 1)
...			
(machining in channel 1 with axes X, Z)		(machining in channel 2 with axes X, Z)	
...			
M502 P12	(waiting for channel 2)	M502 P12	(waiting for channel 1)
M40	(Superimposed control off)	(this channel waits)	
M503 P12	(waiting for channel 2)	M503 P12	(waiting for channel 1)
....		....	
(machining in both channels)		(machining in both channels)	

23 Multichannel Control Parameter Group

N2200 Machine Group Config (T1...8, bit, machine group)

N2200 T1...8	#7	#6	#5	#4	#3	#2	#1	#0
						SBS	ALS	RST

#0 **RST**: Not used.
#1 **ALS**: Not used.
#2 **SBS**: Not used.

N2201 Waiting M Codes Min (integer)
N2202 Waiting M Codes Max (integer)

Value range: 0, ..., $2^{32}-1$.

These two parameters are common for all channels and machine groups and the minimum and maximum values of M codes are written here necessary for the synchronization of channels.

If both parameter values are 0, no such M codes are used.

These waiting codes are program control M codes suppressing the buffer. They are program control codes because these codes are processed by the NC, such as the M99.

Programming:

These waiting M codes are used together with address P. Address P specifies the numbers of the channels to be synchronized.

For example:

meaning of M503 P124:

In channels 1, 2 and 4, the execution of the program shall be synchronized, at the point on which the relevant M code has been programmed.

In case a channel has run to such a code, the related CN_WTNG PLC input flag is set.

In case a channel has run to such a code and it finds the CP_NOWT PLC output flag set in the related channel, that channel will not wait any more. (If in a channel we do not have program run, by setting this flag, the other channels can continue with the machining.)

Example:
Let's say the minimum value of M codes is 500, the maximum value is 599, and let's have 3 channels:

Program of channel 1	**Program of channel 2**	**Program of channel 3**
... machining	... machining	... machining
N60 M501 P12		
waiting for channel 2	N100 M501 P12	
... machining	...machining	N110 M502 P123
		waiting for channels 1 and 2
	M502 P123	
	waiting for channels 1 and 3	
N130 M502 P123		
... machining	... machining	... machining

Explanation:

Channel 1 will run to code M501 as the first, and will wait till channel 2 runs to it, too. After the synchronization, both channel with continue with their own programs. If in channel 2 there is no program running, the PLC may set CP_NOWT PLC flag and channel 1 will not wait till channel 2 runs to code M501 P12.

Channel 3 will run to code M502 as the first and it will have to wait till channels 1 and 2 run to it, too. In the channels, the machining will continue only after the synchronization, except for cases where in any of the channels the PLC has set CP_NOWT flag.

24 Timers/Counters Parameter Group

N2300 Power on Time (floating-point)

Unit: msec.

This parameter displays the integrated power-on period of the control the **operating time** accumulated from the beginning of its life**,** in msec.

N2301 Operation Time (L1...8, floating-point, channel)

Unit: msec.

This parameter displays the time, accumulated from the beginning of the life of the channel, in **automatic mode**, spent in **START state**, in msec.

N2302 Cutting Time (L1...8, floating-point, channel)

Unit: msec.

This parameter displays the cutting time of the channel, accumulated from the beginning of its life, in **automatic mode**, in **START state** with **feed** (G1, G2, etc), in msec.

N2303 Interval Meter (L1...8, floating-point, channel)

Unit: msec.

Free purpose, time-meter. It displays the accumulated time in msec while PLC flag **CP_TMREN** is set in the channel.

N2304 Part Time (L1...8, floating-point, channel)

Unit: msec.

This parameter displays the **machining time of the workpiece** in the channel, in automatic mode. It displays only time spent in START state! The time-meter starts again when the operator starts the selected program again from the beginning.

N2305 Part Count M (L1...8, floating-point, channel)

It is the number of M code which counts the number of parts machined. If the parameter value is 0, the workpiece counter will be incremented by the codes M02 and M30.

N2306 No. of Total Parts (L1...8, floating-point, channel)

This counter displays the total number of parts machined from the beginning of life. The counter is increased by the code determined on the PART COUNT M parameter.

N2307 No. of Machined Parts (L1...8, floating-point, channel)

It is the number of machined parts. The counter is increased by the code determined on the PART COUNT M parameter. It can be read from the macro variable **#3901**.

N2308 No. of Parts Reqrd (L1...8, floating-point, channel)

It is the number of parts requiered. It can be read from the macro variable **#3902**.

25 Manual Handle Parameter Group

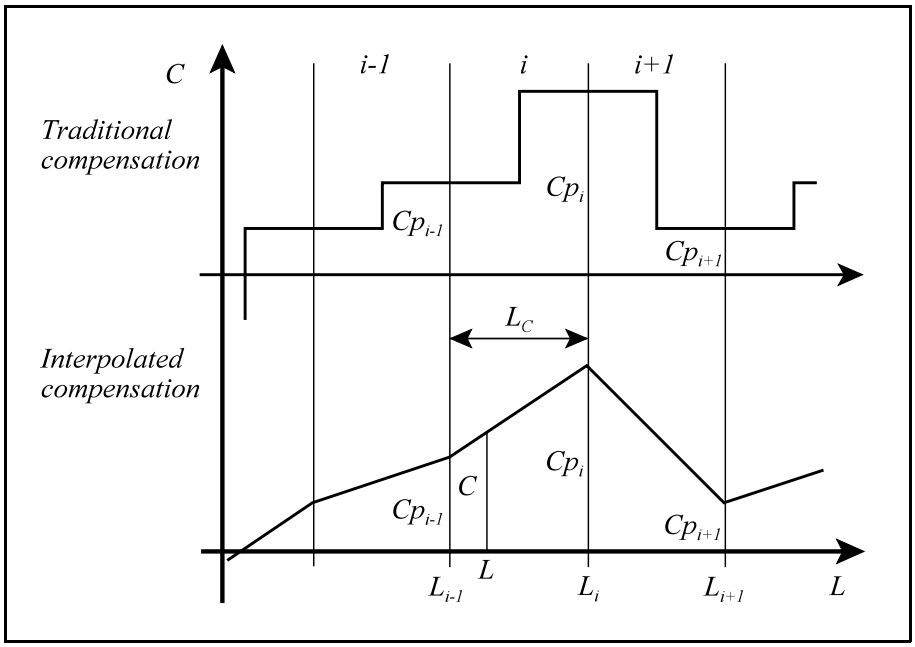
N2400 Handle Direction (A1...32, bit, axis)

N2300 A1...32	#7	#6	#5	#4	#3	#2	#1	#0
	H4M	**H3M**	**H2M**	**H1M**	**H4D**	**H3D**	**H2D**	**H1D**

#0 **H1D**: Axis movement direction for rotation direction of handle 1 on the relevant axis is
=0: the same,
=1: reverse.
#1 **H2D**: Axis movement direction for rotation direction of handle 2 on the relevant axis is
=0: the same,
=1: reverse.
#2 **H3D**: Axis movement direction for rotation direction of handle 3 on the relevant axis is
=0: the same,
=1: reverse.
#3 **H4D**: Axis movement direction for rotation direction of handle 4 on the relevant axis is
=0:the same,
=1: reverse.
#4 **H1M**: The accumulated pulses of handle 1 on the relevant axis:
=0: are moved,
=1: are discarded and axis stops immediately.
#5 **H2M**: The accumulated pulses of handle 2 on the relevant axis:
=0: are moved,
=1: are discarded and axis stops immediately.
#6 **H3M**: The accumulated pulses of handle 3 on the relevant axis:
=0: are moved,
=1: are discarded and axis stops immediately.
#7 **H4M**: The accumulated pulses of handle 4 on the relevant axis:
=0: are moved,
=1: are discarded and axis stops immediately.

26 Pitch Error Compensation Parameter Group

This parameter group serves for the configuration of pitch error compensation. The compensation values (the content of compensation cells) can be found in a separate file.
The **absolute difference from the nominal position** measured on the border of the cell is to be entered at the compensation values.
The compensation values are represented with *float* numbers. Specification of the float number:
4 byte reservation, **7 decimal digit** accuracy, +/-3,4E+/-38 value range.
By storing the compensation values in nanometres, the range of compensation on 7 decimals:
it may range from 0,000001mm to 9,999999mm.
In case the machine error is higher than this in absolute value, a number higher than this may be entered, but the compensation will not have a nanometre-level accuracy.
The control handles for all axes, and for all compensation types altogether 20,000 compensation values.



The compensation is **traditional** if we take the compensation values assigned to cells into consideration at a point within the cell, e.g. at the center of the cell.
The compensation is **interpolated** if the compensation value is not taken into consideration at only one point, but within the cell, the compensation values are evenly distributed depending on the position. The picture above shows the difference between the traditional and interpolated compensation.
NCT2xx controls apply only the interpolated compensation.

The equation of interpolated compensation, if the movement has a positive direction:

$$Cp = Cp_{i-1} + (Cp_i - Cp_{i-1})\frac{L - L_{i-1}}{L_i - L_{i-1}}$$

where: Cp is the positive-direction absolute value of compensation in position L
Cp_{i-1} is the positive-direction compensation value belonging to i-1-th cell
Cp_i is the positive-direction compensation value belonging to i-th cell
L is the current position
L_{i-1} is the position of i-1-th cell

L_i is the position of i-th cell

Let's have:

$$l = \frac{L - L_{i-1}}{L_i - L_{i-1}} = \frac{L - L_{i-1}}{Interval}$$

Equation of compensation:

$$Cp = Cp_{i-1}(1 - l) + Cp_i l$$

If the compensation value one cycle earlier was C_0, the compensated position p_k will be modified in the following way:

$p_k=p_k+C–C_0$

If the movement measured from the encoder changes direction, i.e. the axis move into a negative direction, the equation of compensation in the point with position L will be as follows:

$$Cn = Cn_i + (Cn_{i-1} - Cn_i)\frac{L - L_i}{L_{i-1} - L_i} = Cn_{i-1} + (Cn_i - Cn_{i-1})\frac{L - L_{i-1}}{L_i - L_{i-1}}$$

Introducing designation "l" **the equation of compensation in a negative direction**:

$$Cn = Cn_{i-1}(1 - l) + Cn_i l$$

In case we compensate only one direction, then $Cp_i=Cn_i$.

The way of compensation will be the same as before:

$p_k=p_k+C–C_0$

During direction change, the $C–C_0$ difference will be 0, if the compensation is unidirectional, however, if it is bidirectional, the difference will contain the position-dependent backlash value. The control counts the compensation with a cycle of **10 msec**.

N2500 Pitch Comp. Contr. (A1...32, bit, axis)

N2500 A1...32	#7	#6	#5	#4	#3	#2	#1	#0
					ESC	**CYC**	**BDC**	**EPC**

#0 **EPC**: ***Primary pitch error compensation*** is related to the full stroke of the selected axis.

If the parameter value is:

=0: the primary pitch error compensation is disabled,

=1: the primary pitch error compensation is enabled on the given axis.

#1 **BDC**: In case of a ***bidirectional pitch error compensation***, two different compensation values belong to the same point, depending on the direction in which the axis crosses the given point.

If the parameter value is:

=0: the primary pitch error compensation will be unidirectional,

=1: the primary pitch error compensation will be bidirectional on the given axis.

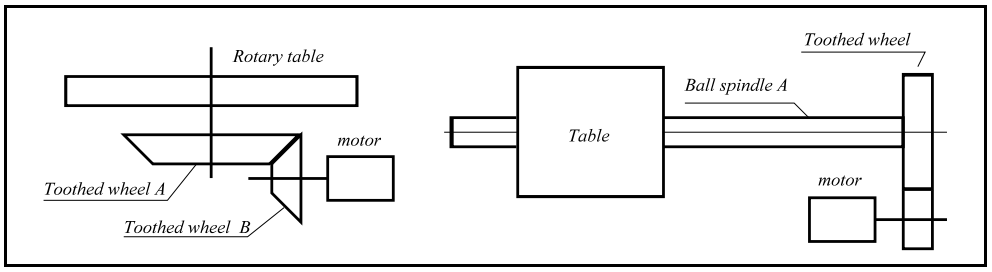
#2 **CYC**: The ***primary cyclic pitch error compensation*** is applied to rotary axes, e.g. a rotary table. The primary cyclic pitch error compensation may be uni- or bidirectional, too, furthermore, it may be applied together with the secondary cyclic pitch error compensation, if the gear of the rotary table has a cyclic error.

If the parameter value is:

=0: the primary pitch error compensation is not cyclic,

=1: the primary pitch error compensation is cyclic on the given axis.

#3 **ESC**: The ***secondary cyclic pitch error compensation*** compensates the pitch errors occurring cyclically on the selected axis e.g. when a gear is between the motor and the axis. In this case #2 PER is to be set in parameter N0106 Axis Properties. The primary, or primary cyclic and secondary cyclic pitch error compensation may be applied together.
If the parameter value is:
=0: the secondary cyclic pitch error compensation is not used,
=1: the secondary cyclic pitch error compensation is used on the given axis.

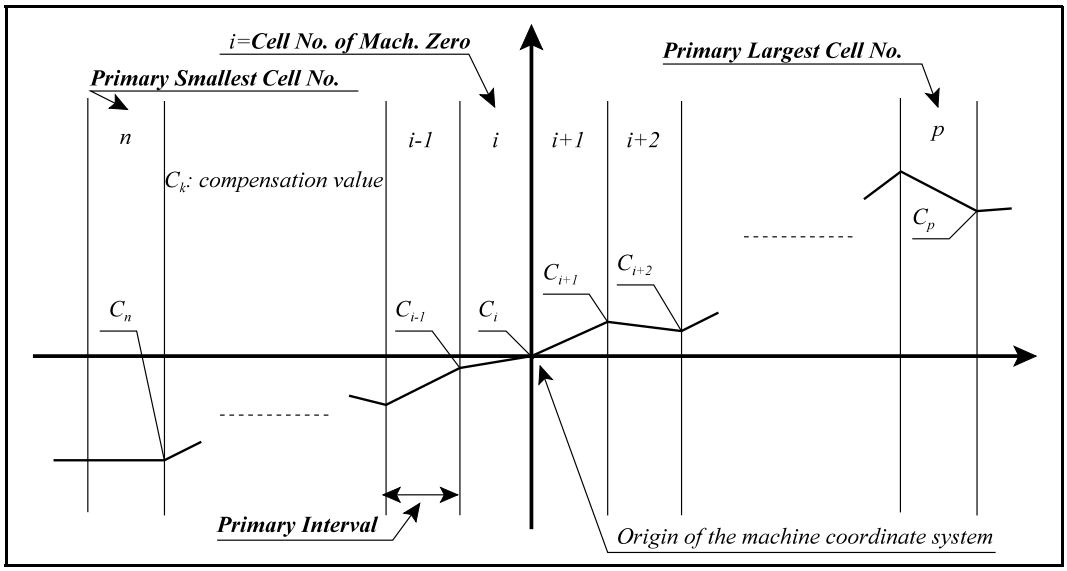


N2501 Primary Interval (A1...32, floating-point, axis)
Distance data, the number written on the parameter, based on #0 IND parameter, shall be interpreted in inch or mm, if #1 ROT=0.
If #1 ROT=1 the data will be interpreted in degrees.
It is the interval of equally spaced compensation cells used during the primary pitch error compensation, on the given axis.

N2502 Cell No. of Mach. Zero (A1...32, integer, axis)
Value range: 10000....29999
It is the number of the compensation cell belonging to the machine zero point, in case of unidirectional primary pitch error compensation.

N2503 Primary Smallest Cell No. (A1...32, integer, axis)
Value range: 10000....29999
It is the smallest number of the compensation cell falling in the furthest negative direction, in case of unidirectional primary pitch error compensation.

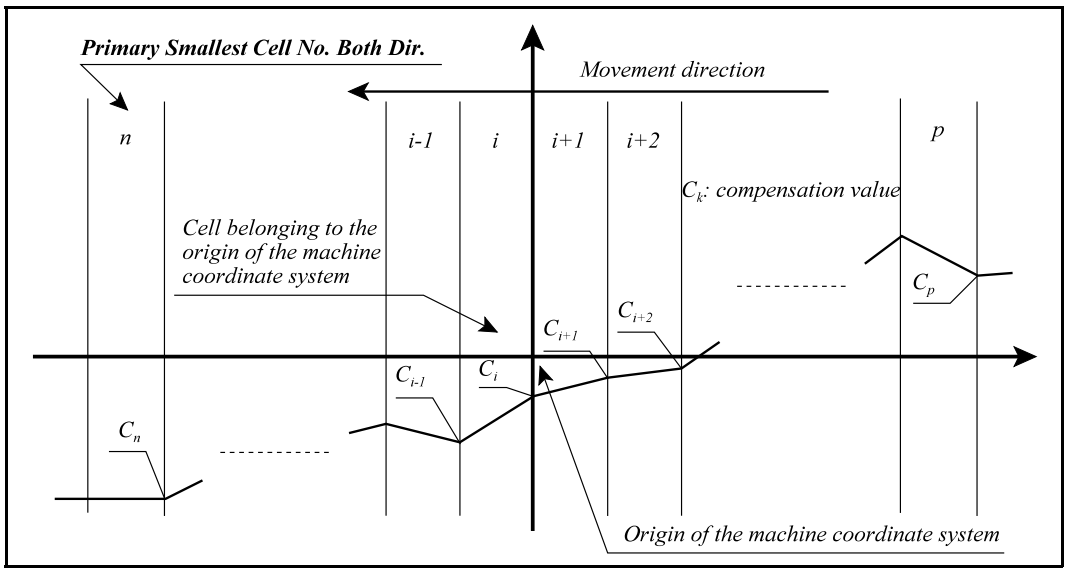
N2504 Primary Largest Cell No. (A1...32, integer, axis)
Value range: 10000....29999
It is the largest number of the compensation cell falling in the furthest positive direction, in case of unidirectional primary pitch error compensation.



N2505 Primary Smallest Cell No. Both Dir. (A1...32, integer, axis)

Value range: 10000....29999

It is the smallest number of the compensation cell falling in the furthest negative direction, in case of bidirectional primary pitch error compensation. (The other compensation cell numbers will be automatically calculated from parameters N2502... N2504.)



N2506 Distance of Cycle (A1...32, floating-point, axis)

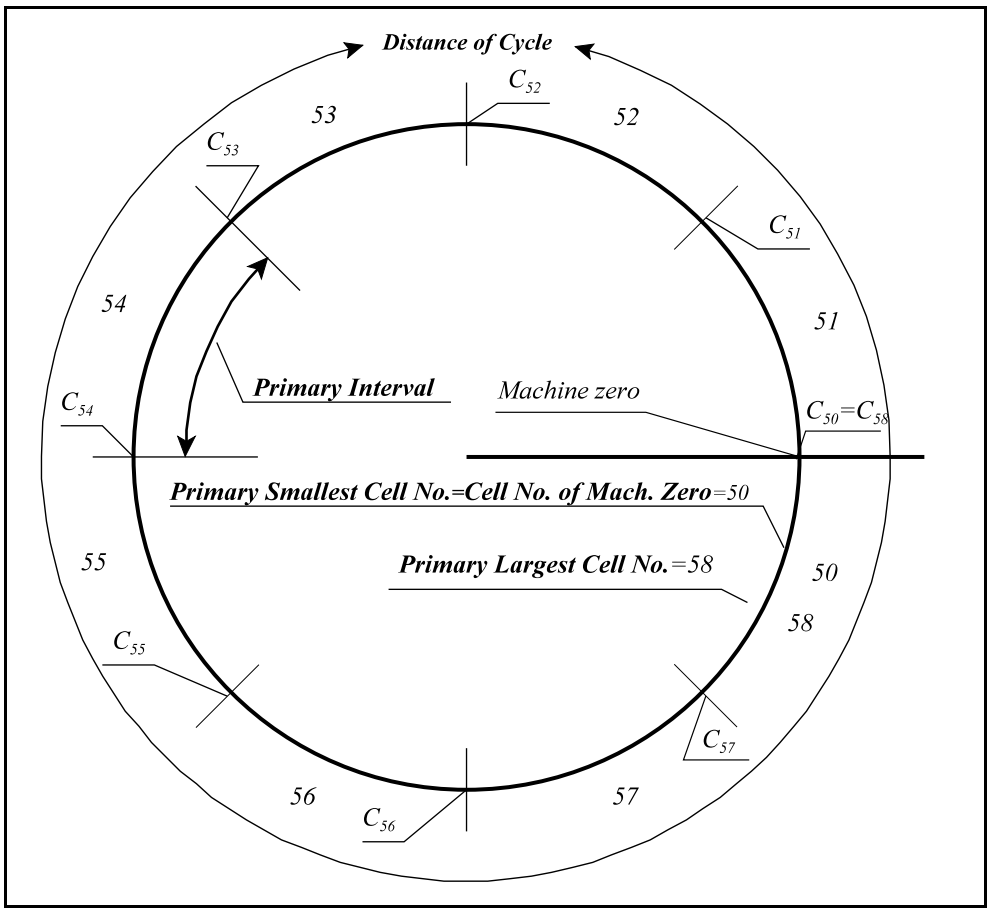
Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch, or mm, if #1 ROT=0.

If #1 ROT=1 the data will be interpreted in degrees.

It determines the distance of the cycle to be compensated, used during the primary cyclic pitch error compensation, on the given axis. Usually it is 360°, however, this is not mandatory. By entering the parameters the below conditions needs to be met:

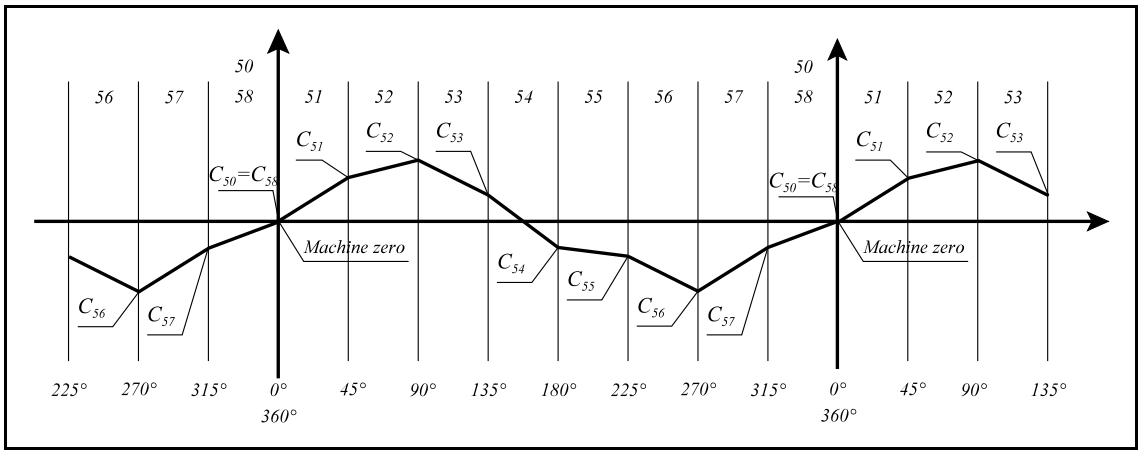
*Distance of Cycle=(Primary Largest Cell No.–Primary Smallest Cell No.)*Primary Interval*

i.e. multiplying the number of used cells with the interval of compensation cells has to result in the parameter.



The same compensation value is to be written into the cell belonging to the machine zero point and into the one with the highest number. Otherwise the compensation value will be accumulated and will result in a mismatched position.

In case of unidirectional compensation, the value entered into the smallest and highest cell is 0, respectively, while in case of bidirectional compensation, e.g. the value of the first and last cell belonging to the positive-direction movement is 0.

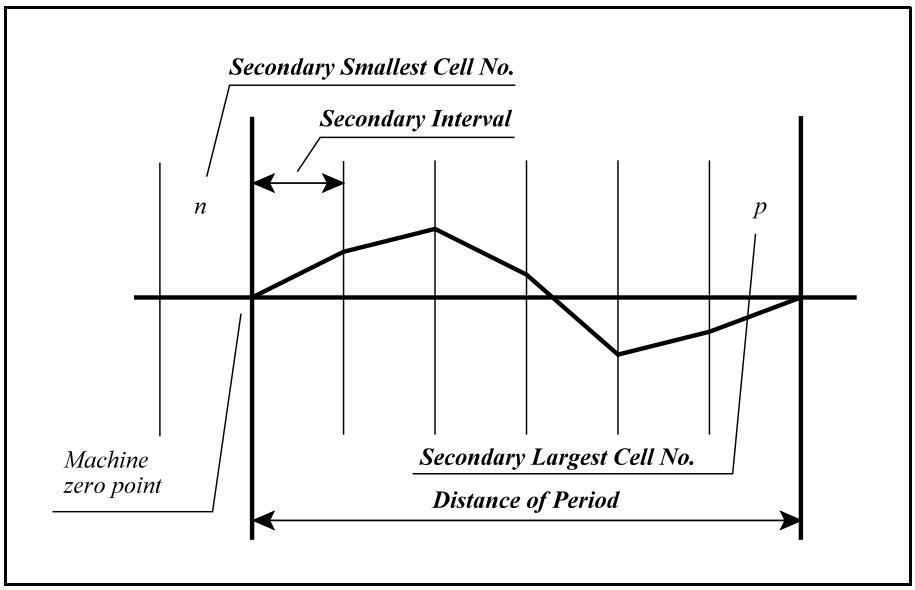


N2507 Secondary Interval (A1...32, floating-point, axis)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch, or mm, if #1 ROT=0.

If #1 ROT=1 the data will be interpreted in degrees.

It is the interval of equally spaced compensation cells used during the secondary cyclic pitch error compensation, on the given axis.



N2508 Secondary Smallest Cell No. (A1...32, integer, axis)

Value range: 10000....29999

It is the smallest number of the compensation cell falling in the furthest negative direction and at the same time the cell belonging to the machine zero point in case of secondary cyclic pitch error compensation.

N2509 Secondary Largest Cell No. (A1...32, integer, axis)

Value range: 10000....29999

It is the largest number of the compensation cell falling in the furthest positive direction, in case of secondary cyclic pitch error compensation.

N2510 Distance of Period (A1...32, floating-point, axis)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch, or mm, if #1 ROT=0.

If #1 ROT=1 the data will be interpreted in degrees.

It determines the distance of period to be compensated during secondary cyclic pitch error compensation on the given axis. By the entering of parameters, the following condition needs to be met:

Distane of Period=(Secondary Largest Cell No.–Secondary Smallest Cell No.)*Secondary Interval

i.e. multiplying the number of used cells with the number of compensation cells has to result in the parameter.

Secondary pitch error compensation is only unidirectional.

27 Straightness Compensation Parameter Group

This parameter group serves for the configuration of compensation of the straightness error. In case of a normal compensation, when compensation is made per cells, the compensation values (the content of compensation cells) can be found in a separate file.
The **absolute difference from the nominal position** measured on the border of the cell is to be entered at the compensation values.
Compensation values are represented by *float* numbers. Specification of the float number representation:

4 byte reservation, **7 decimal digit** accuracy, +/-3,4E+/-38 value range.

By storing the compensation value in nanometers, the range of compensation on 7 decimals:

it may range from 0,000001mm to 9,999999mm.

In case the machine error is higher than this in absolute value, a number higher than this may be entered but the compensation will not have a nanometer-level accuracy.
The control handles for all axes and for all compensation types altogether 20,000 compensation values.

N2600 Straightness Comp. Contr. (L1...8, bit, channel)

N2600 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
								SY

#0 **SY:** If the SY parameter value is:

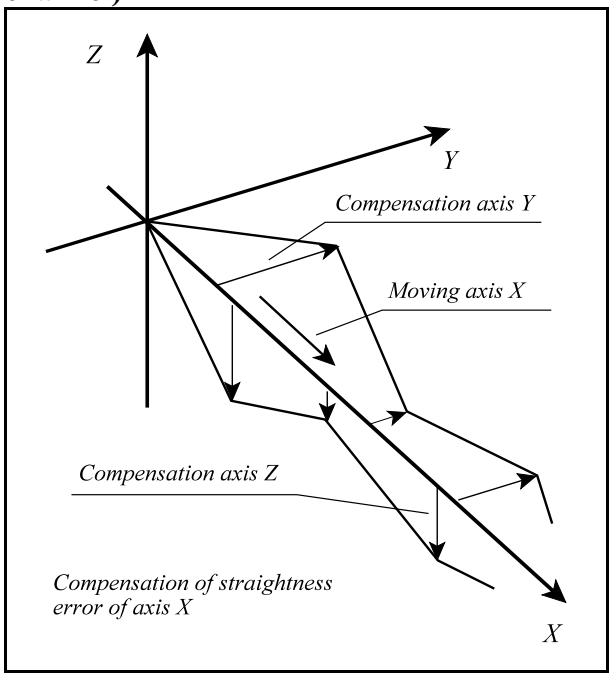
=0: in the given channel, the straightness compensation per cell is effective, (use of Smallest Cell No. of Moving Axis n parameter)

=1: in the given channel, the simplified straightness compensation is used.

N2601 No. of Moving Axis 1 (L1...8, integer, channel)
N2602 No. of Moving Axis 2 (L1...8, integer, channel)
N2603 No. of Moving Axis 3 (L1...8, integer, channel)
N2604 No. of Moving Axis 4 (L1...8, integer, channel)
N2605 No. of Moving Axis 5 (L1...8, integer, channel)
N2606 No. of Moving Axis 6 (L1...8, integer, channel)

Value range: 1...32

It specifies the number of moving axis to be compensated, per channel. If its value is 0, no compensation is made in that group. In case of a simplified compensation, only parameters with indexes 1-3 may be used.



N2607 No. of Compensation Axis 1 (L1...8, integer, channel)
N2608 No. of Compensation Axis 2 (L1...8, integer, channel)
N2609 No. of Compensation Axis 3 (L1...8, integer, channel)
N2610 No. of Compensation Axis 4 (L1...8, integer, channel)
N2611 No. of Compensation Axis 5 (L1...8, integer, channel)
N2612 No. of Compensation Axis 6 (L1...8, integer, channel)

Value range: 1...32

It specifies the axis number of compensation axis belonging to the appropiate moving axis parameter. In case of a simplified compensation, only parameters with indexes 1-3 may be used.

In case of the figure above the straightness of axis X is to be compensated in Y as well as in Z direction. Therefore X axis is specified twice among the parameters of the moving axes:

N2601 No. of Moving Axis 1=1 (if X is axis 1)
N2602 No. of Moving Axis 2=1 (if X is axis 1)

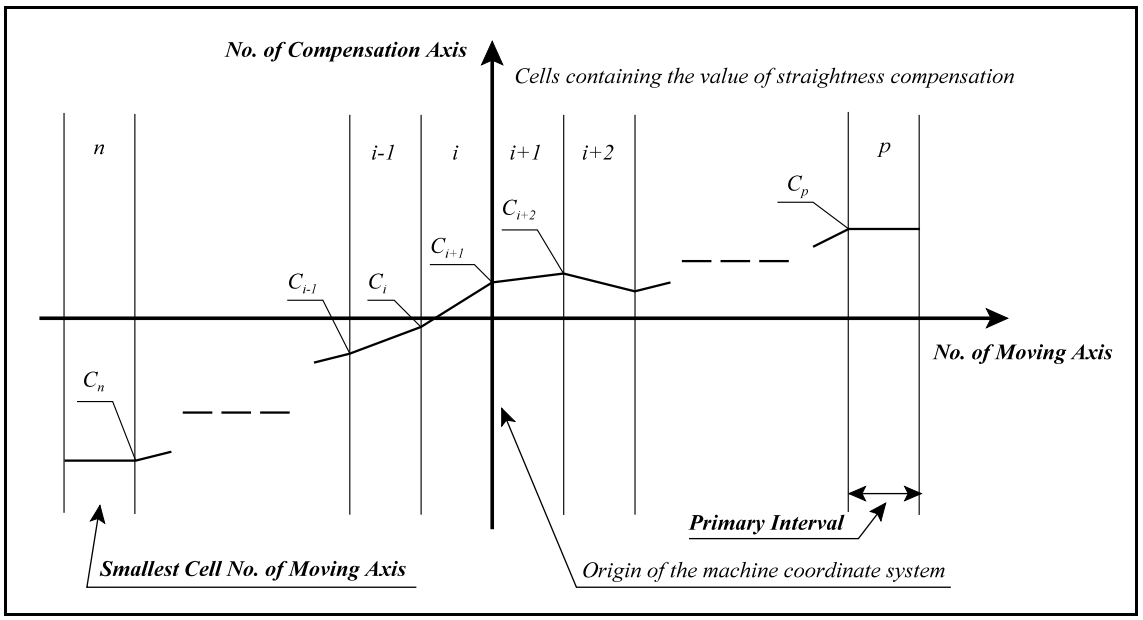
Parameter N2607 No. of Compensation Axis1 belongs to parameter N2601 No. of Moving Axis1 while parameter N2608 No. of Compensation Axis2 belongs to parameter N2602 No. of Moving Axis2:

N2607 No. of Compensation Axis 1=2 (if Y is axis 2)
N2608 No. of Compensation Axis 2=3 (if Z is axis 3)

N2613 Smallest Cell No. of Moving Axis 1 (L1...8, integer, channel)
N2614 Smallest Cell No. of Moving Axis 2 (L1...8, integer, channel)
N2615 Smallest Cell No. of Moving Axis 3 (L1...8, integer, channel)
N2616 Smallest Cell No. of Moving Axis 4 (L1...8, integer, channel)
N2617 Smallest Cell No. of Moving Axis 5 (L1...8, integer, channel)
N2618 Smallest Cell No. of Moving Axis 6 (L1...8, integer, channel)

It is the smallest number of the compensation cell falling in the furthest negative direction on the moving axis to be compensated if the SY parameter value is 0.
The number of other cells carrying out straightness compensation will be calculated from the parameters N2502 Cell No. of Mach. Zero and N2506 Primary Largest Cell No. of Pitch Error Compensation Parameter Group. Therefore these parameters have to be set too if no straightness compensation is applied on the given axis!

☞ **The compensation values** are specified in the compensation cells when SY=0!



The control carries out the straightness compensation in interpolated way, too.
Let's say the position of the moving axis is L. Then:

$$l = \frac{L - L_{i-1}}{L_i - L_{i-1}} = \frac{L - L_{i-1}}{Interval}$$

where L_{i-1}: is the position of the (i-1)-th cell of the moving axis
L_i: is the position of the i-th cell of the moving axis.

Equation of compensation:

$$Cs = Cs_{i-1}(1 - l) + Cs_i l$$

where Cs: is the value of straightness compensation
Cs_{i-1}: is the compensation value entered into the (i–1)-th cell of the compensation axis
Cs_i: is the compensation value entered into the i–th cell of the compensation axis.

Taking into consideration of the compensation:

$p_k = p_k + Cs - Cs_0$

where p_k is the compensated position of the compensation axis,
Cs: is the current value of straightness compensation,

Csi_0: is the previous value of straightness compensation.

N2619 Position A of Moving Axis 1 (L1...8, floating-point, channel)
N2620 Position B of Moving Axis 1 (L1...8, floating-point, channel)
N2621 Position C of Moving Axis 1 (L1...8, floating-point, channel)
N2622 Position D of Moving Axis 1 (L1...8, floating-point, channel)

N2623 Position A of Moving Axis 2 (L1...8, floating-point, channel)
N2624 Position B of Moving Axis 2 (L1...8, floating-point, channel)
N2625 Position C of Moving Axis 2 (L1...8, floating-point, channel)
N2626 Position D of Moving Axis 2 (L1...8, floating-point, channel)

N2627 Position A of Moving Axis 3 (L1...8, floating-point, channel)
N2628 Position B of Moving Axis 3 (L1...8, floating-point, channel)
N2629 Position C of Moving Axis 3 (L1...8, floating-point, channel)
N2630 Position D of Moving Axis 3 (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm.

In case of simplified straightness compensation (SY=1 parameter state) on the moving axis 4 different positions (A, B, C, D) can be specified in the machine coordinate system to which the compensation values are determined by the Value A, B, C, D of Comp. Axis 1, 2, 3, parameters.

The below relationship shall be fulfilled for the parameters:

Position A of Moving Axis i<Position B of Moving Axis i<Position C of Moving Axis i<Position D of Moving Axis i (i=1, 2, 3).

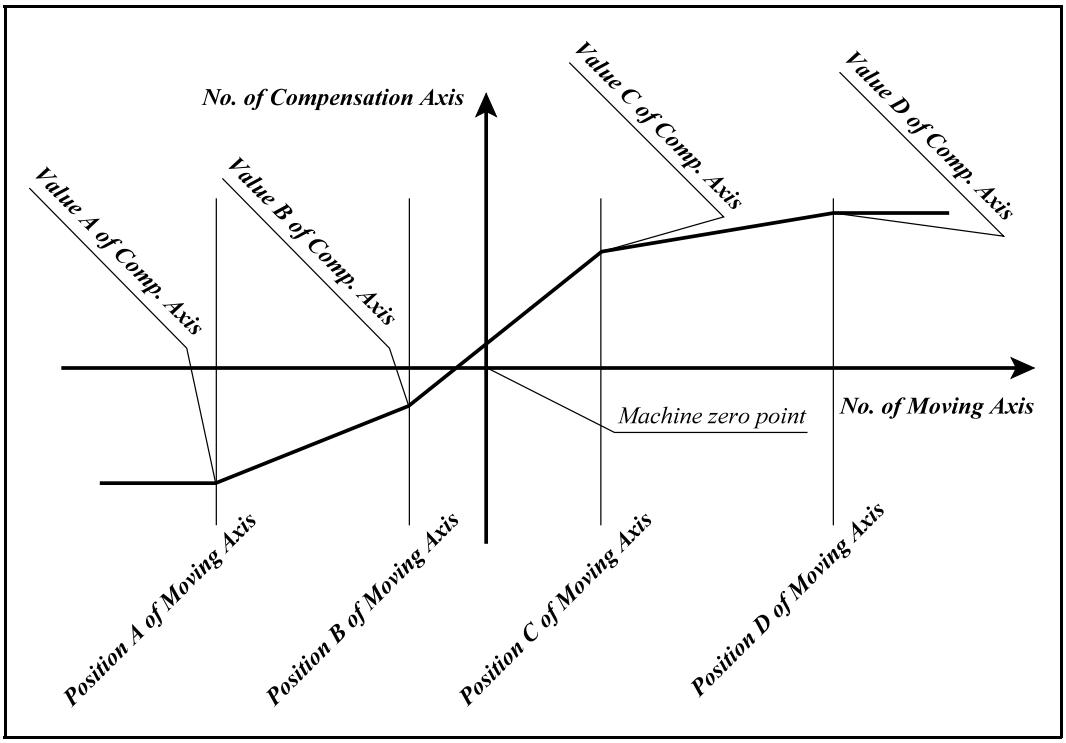
N2631 Value A of Comp. Axis 1 (L1...8, floating-point, channel)
N2632 Value B of Comp. Axis 1 (L1...8, floating-point, channel)
N2633 Value C of Comp. Axis 1 (L1...8, floating-point, channel)
N2634 Value D of Comp. Axis 1 (L1...8, floating-point, channel)

N2635 Value A of Comp. Axis 2 (L1...8, floating-point, channel)
N2636 Value B of Comp. Axis 2 (L1...8, floating-point, channel)
N2637 Value C of Comp. Axis 2 (L1...8, floating-point, channel)
N2638 Value D of Comp. Axis 2 (L1...8, floating-point, channel)

N2639 Value A of Comp. Axis 3 (L1...8, floating-point, channel)
N2640 Value B of Comp. Axis 3 (L1...8, floating-point, channel)
N2641 Value C of Comp. Axis 3 (L1...8, floating-point, channel)
N2642 Value D of Comp. Axis 3 (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm.

In case of simplified straightness compensation (SY=1 parameter state) it is the amount of movement on the compensation axis in the appropriate A, B, C, D position of the moving axis.


No. of Compensation Axis
Value A of Comp. Axis
Value B of Comp. Axis
Value C of Comp. Axis
Value D of Comp. Axis
Machine zero point
No. of Moving Axis
Position A of Moving Axis
Position B of Moving Axis
Position C of Moving Axis
Position D of Moving Axis

28 Three Dimensional Compensation Parameter Group

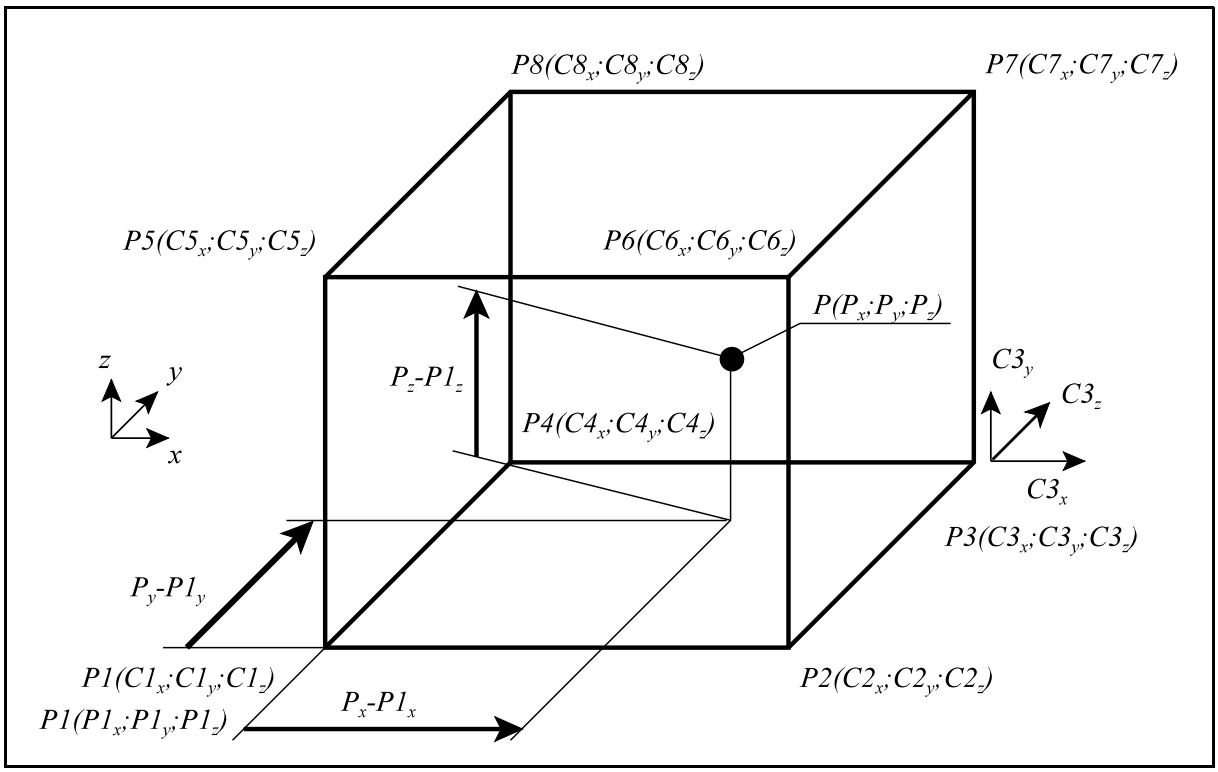
This parameter group serves for the compensation of the three-dimensional error of the machine.
The compensation values (the content of compensation cells) can be found in a separate file.
The **absolute difference from the nominal position** measured on the border of the cell is to be entered at the compensation values.
Compensation values are represented by *float* numbers. Specification of the float number representation:

4 byte reservation, **7 decimal digit** accuracy, +/-3,4E+/-38 value range.

By storing the compensation value in nanometers, the range of compensation on 7 decimals:

it may range from 0,000001mm to 9,999999mm.

In case the machine error is higher than this in absolute value, a number higher than this may be entered, but the compensation will not have a nanometer-level accuracy.
The control handles for all axes, and for all compensation types altogether 20,000 compensation values.
The three-dimensional error compensation is the extension of the simple (one-dimensional) pitch error compensation to the three-dimensional space.
8 compensation points can be assigned to all P(P_x; P_y; P_z)-position the coordinates of which are: Pi(Pi_x; Pi_y; Pi_z), where i=1...8. Position of P1 has special meaning. The 8 points are located on a rectangular parallelepiped.
3 compensation values are specified to all 8 points: Pi(Ci_x, Ci_y, Ci_z), where i=1...8.



Let's introduce the following designation:

$$x = \frac{P_x - P1_x}{P2_x - P1_x} = \frac{P_x - P1_x}{Interval_x}$$

$$y = \frac{P_y - P1_y}{P4_y - P1_y} = \frac{P_y - P1_y}{Interval_y}$$

$$z = \frac{P_z - P1_z}{P5_z - P1_z} = \frac{P_z - P1_z}{Interval_z}$$

The X-direction compensation shall be in Point P:

$$C_x = \left\{\left[C1_x(1-x) + C2_x x\right]\left((1-y) + \left[C3_x x + C4_x(1-x)\right]y\right\}(1-z) +$$
$$+ \left\{\left[C5_x(1-x) + C6_x x\right](1-y) + \left[C7_x x + C8_x(1-x)\right]y\right\}z$$

C_y and C_z compensations valid in Point P are calculated the same way, only the axis indexes of compensation values shall be exchanged: Ci_y, or Ci_z is written instead of Ci_x.
The equation shall be set up as follows:

$$\begin{bmatrix} C_x \\ C_y \\ C_z \end{bmatrix} = \begin{bmatrix} C1_x & C2_x & C3_x & C4_x & C5_x & C6_x & C7_x & C8_x \\ C1_y & C2_y & C3_y & C4_y & C5_y & C6_y & C7_y & C8_y \\ C1_z & C2_z & C3_z & C4_z & C5_z & C6_z & C7_z & C8_z \end{bmatrix} \times \begin{bmatrix} (1-x)(1-y)(1-z) \\ x(1-y)(1-z) \\ xy(1-z) \\ (1-x)y(1-z) \\ (1-x)(1-y)z \\ x(1-y)z \\ xyz \\ (1-x)yz \end{bmatrix}$$

N2700 1st Axis of 3D Compensation (L1...8, integer, channel)
N2701 2nd Axis of 3D Compensation (L1...8, integer, channel)
N2702 3rd Axis of 3D Compensation (L1...8, integer, channel)

Value range: 1...32

3 axes per channel participating in the three-dimensional compensation can be assigned on the parameter.

If one of the parameters has a value of 0, the compensation will be made in plane (two-dimensional) if two of the parameters have a value of 0, the compensation will be one-dimensional. Only linear axes may be appointed.

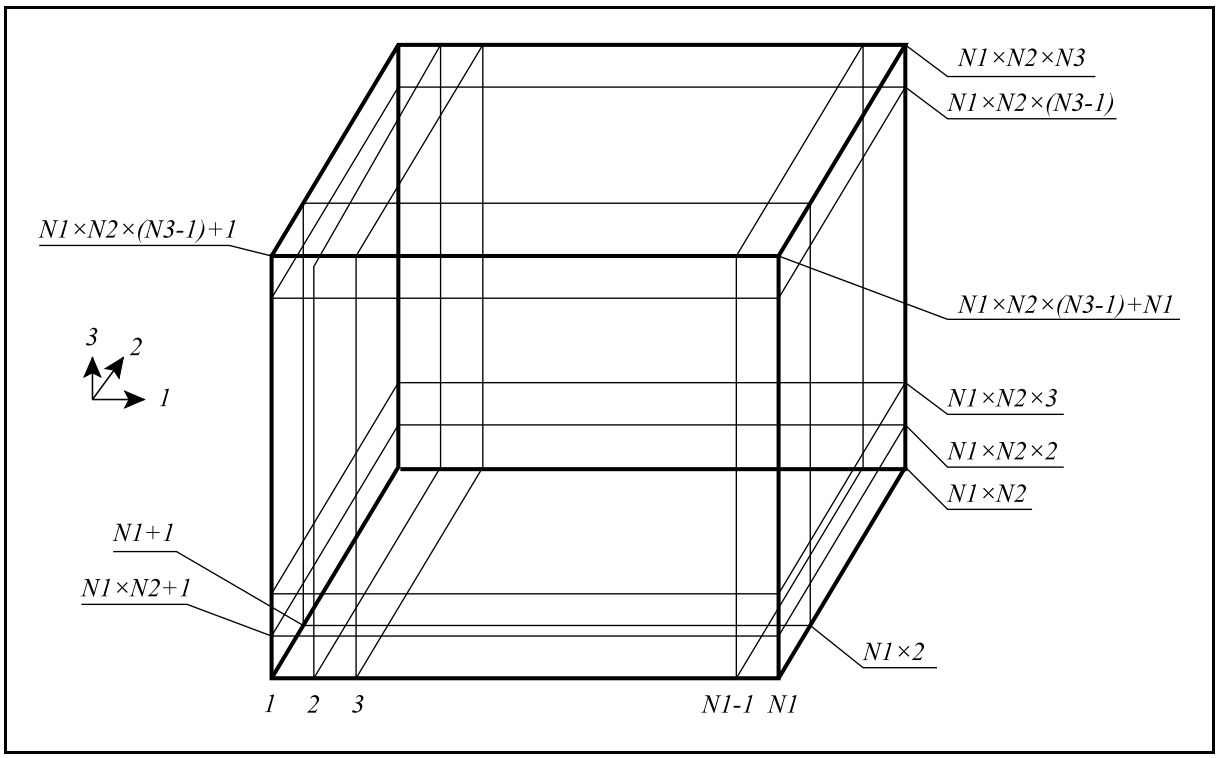
N2703 No. of Points on 1st Axis (L1...8, integer, channel)
N2704 No. of Points on 2nd Axis (L1...8, integer, channel)
N2705 No. of Points on 3rd Axis (L1...8, integer, channel)

For all three axes participating in the three-dimensional compensation, the number of points participating in the compensation may be determined.

For the division of the three-dimensional space let's introduce the following designations:

N1: number of compensation points along axis 1.
N2: number of compensation points along axis 2.
N3: number of compensation points along axis 3.

The assignment of space to compensation point numbers is the following:



3 compensation values belong to each compensation point, in the direction of 3 axes of the space. The compensation values contain the absolute difference from the nominal position.

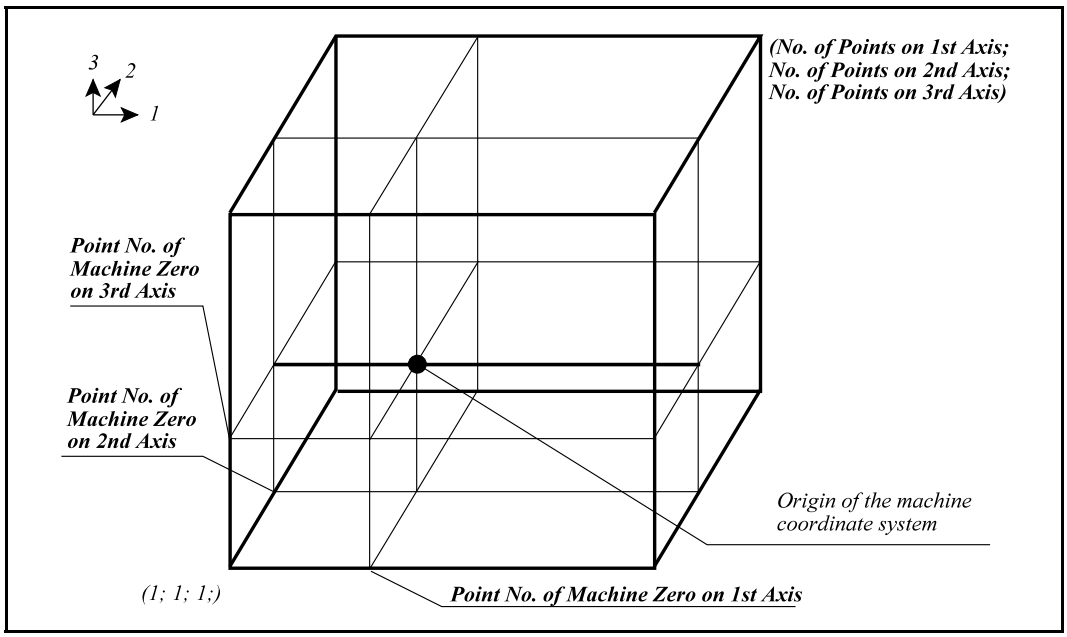
The value of compensation is represented in float. This means that the compensation of a point reserves 3×4=12 bytes.

If the number of compensation points per axis is 25, the maximum compensation point number is: 25^3=15625. The reservation in memory will be:

15625×12=187500 byte

N2706 Point No. of Machine Zero on 1st Axis (L1...8, integer, channel)
N2707 Point No. of Machine Zero on 2nd Axis (L1...8, integer, channel)
N2708 Point No. of Machine Zero on 3rd Axis (L1...8, integer, channel)

The parameters specify the point number belonging to the origin of machine coordinate system for axes participating in the three-dimensional compensation.
The compensation value belonging to this point shall be mandatorily 0, on all axes.



N2709 Comp. Interval on 1st Axis (L1...8, floating-point, channel)
N2710 Comp. Interval on 2nd Axis (L1...8, floating-point, channel)
N2711 Comp. Interval on 3rd Axis (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm.
The parameter specifies the interval of compensation points on the axes participating in three-dimensional compensation.

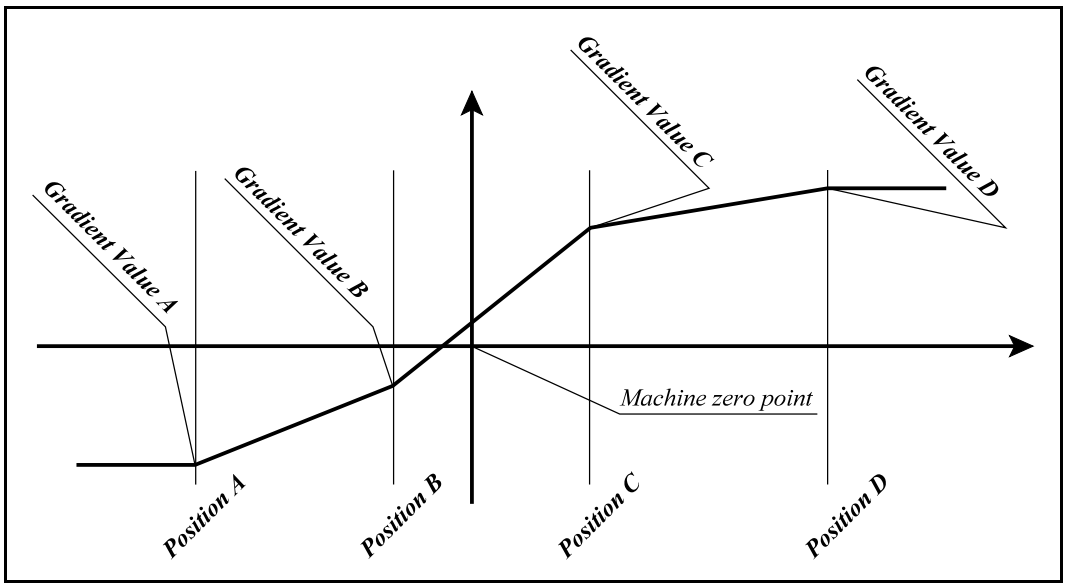
29 Gradient Compensation Parameter Group

The gradient compensation is a simplified pitch error compensation, similarly to the simplified straightness compensation. However, the gradient compensation may operate simultaneously with the normal pitch error compensation. The compensation values of the gradient compensation can be found in the Gradient Compensation Parameter Group.

N2800 Gradient Comp. Contr. (A1...32, bit, axis)

N2800 A1...32	#7	#6	#5	#4	#3	#2	#1	#0
							GCY	**EGC**

#0 **EGC**: If the parameter value
=0: the gradient compensation is not disabled,
=1: the gradient compensation is enabled on the given axis.
#1 **GCY**: If the parameter value is
=0: the gradient compensation is not cyclical,
=1: the gradient compensation cyclical on the given axis.
If the gradient compensation is cyclical, its period is determined by the Pitch Error Compensation Parameter Group's Distance of Cycle parameter.



N2801 Position A (A1...32, floating-point, axis)
N2802 Position B (A1...32, floating-point, axis)
N2803 Position C (A1...32, floating-point, axis)
N2804 Position D (A1...32, floating-point, axis)
Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm, if #1 ROT=0.
If #1 ROT=1 the data will be interpreted in degrees.

In case of a gradient compensation, 4 positions can be determined on all axes in the machine coordinate system, to which the compensation values are determined by the Gradient Value A, B, C, D parameters.
The following relationship shall be fulfilled for the parameters:
Position A<Position B<Position C<Position D
If the gradient compensation is cyclical (GCY=1), the Position A parameter value shall be mandatorily 0, i.e. it is the origin of the machine coordinate system.

N2805 Gradient Value A (A1...32, floating-point, axis)
N2806 Gradient Value B (A1...32, floating-point, axis)
N2807 Gradient Value C (A1...32, floating-point, axis)
N2808 Gradient Value D (A1...32, floating-point, axis)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm, if #1 ROT=0.
If #1 ROT=1 the data will be interpreted in degrees.
In case of a gradient compensation it is the compensation value in the appropriate A, B, C, D position of the axis.
If the gradient compensation is cyclical (GCY=1), the Gradient Value A parameter value shall be mandatorily 0.

30 Tool Management Parameter Group

N2900 Tool M. Config (bit)

N2900	#7	#6	#5	#4	#3	#2	#1	#0
							TLN	**TMU**

#0 **TMU**: If the parameter value

=0: tool management is not performed on the machine, the tool management tables are not used and NC will not display them, and the registers and flags belonging to tool management will not be managed by the NC.

=1: the tool management is used.

#1 **TLN**: it determines the operation of the tool life expiration notice signal (PLC flag SN_TLNL). If the parameter

=0: it calculates for the **tool** in the spindle the difference between the Tool Life and Tool Life Counter and if the difference is lower than the Notice Life value it will set PLC flag SN_TLNL.

=1: it calculates for the **tool group** in the spindle the summarized Tool Life subtracts the summarized Tool Life Counter value from it and if the difference is lower than the Notice Life value, it will set PLC flag SN_TLNL. Then it is advisable to write the same Notice Life value for all tools with the same type number.

N2901 Search Config (L1...8, bit, channel)

N2901 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
	TSP		**NM4**	**NM3**	**NM2**	**NM1**	**TLC**	**TCU**

#0 **TCU**: It determines the selection from among tools with the same type number. If the parameter value is:

=0: the tool which has the lowest life i.e.highest Life Counter value is selected,

=1: the tool which has the lowest value set on the user data of the Tool management table is selected. The fact which user data shall be taken into consideration, depends on N2903 Custom Life Data No. parameter.

#1 **TLC**: If the parameter value

=0: the life counter will start upon M06,

=1: the life counter will start upon T code,

after the PLC has returned the CP_FIN signal.

#2, ..., #5 **NM1, ..., NM4**: in the given channel, if the parameter value

=0: it will not search for a tool in magazines 1., ..., 4.

=1: it will search for a tool in magazines 1., ..., 4.

#7 **TSP**: It controls the displaying of tool numbers in the FST window. If the parameter value

=0: it will display the tool number from the CP_ACTT PLC register bound to the channel,

=1: it will display the tool number from the SP_ACTT PLC register bound to the spindles.

This latter case can be used e.g. on multi-spindle milling machines.

N2902 Tool Management Table Length (integer)

It determines the tool management table length (the number of its rows). As many tools can be registered in the table as the number of rows.
Maximum value: 1,000.

N2903 No. of Custom Columns (integer)

The first 12 columns of the Tool management table from Data Number up to Feed are always available if the tool management is enabled by the N2900 Tool M. Config parameter #0 TMU bit.
The number of user columns can determined on this parameter.
The maximum number of columns can be 20.
In the first column always bit data can be entered.

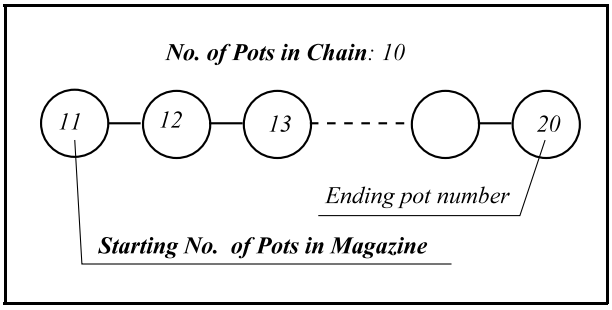
N2904 Custom Life Column No. (integer)

If the N2901 Search Config parameter's #0 TCU bit is set for life management by user data, the parameter defines the user column number by which the tool is selected. From among the tools with the same type number the one with the lowest user data value is selected.

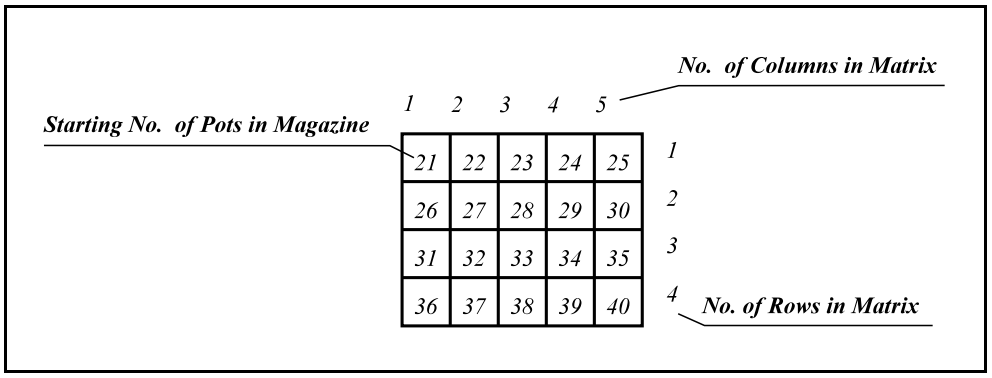
N2905 Magazine Config (bit)

N2905	#7	#6	#5	#4	#3	#2	#1	#0
					MT4	**MT3**	**MT2**	**MT1**

#0, ..., #3 **MT1**, ..., **MT4**: The magazine layout is if the parameter value
=0: chain type,



=1: matrix type.



N2906 Starting No. of Pots in Magazine 1 (integer)

It is the starting pot number of the first magazine in the Tool Pot table. The pot numbers in Tool Pot table are incremented from this value on.

N2907 Starting No. of Pots in Magazine 2 (integer)

It is the starting pot number of the second magazine in the Tool Pot table. The pot numbers in Tool Pot table are incremented from this value on.

N2908 Starting No. of Pots in Magazine 3 (integer)

It is the starting pot number of the third magazine in the Tool Pot table. The pot numbers in Tool Pot table are incremented from this value on.

N2909 Starting No. of Pots in Magazine 4 (integer)

It is the starting pot number of the fouth magazine in the Tool Pot table. The pot numbers in Tool Pot table are incremented from this value on.

N2910 No. of Pots in Chain 1 (integer)

It is the number of tool pots in chain type magazine 1.

N2911 No. of Pots in Chain 2 (integer)

It is the number of tool pots in chain type magazine 2.

N2912 No. of Pots in Chain 3 (integer)

It is the number of tool pots in chain type magazine 3.

N2913 No. of Pots in Chain 4 (integer)

It is the number of tool pots in chain type magazine 4.

N2914 No. of Rows in Matrix 1 (integer)

It is the number of rows in matrix type magazine 1.

N2915 No. of Columns in Matrix 1 (integer)

It is the number of columns in the matrix type magazine 1.

N2916 No. of Rows in Matrix 2 (integer)
It is the number of rows in matrix type magazine 2.

N2917 No. of Columns in Matrix 2 (integer)
It is the number of columns in the matrix type magazine 2.

N2918 No. of Rows in Matrix 3 (integer)
It is the number of rows in matrix type magazine 3.

N2919 No. of Columns in Matrix 3 (integer)
It is the number of columns in the matrix type magazine 3.

N2920 No. of Rows in Matrix 4 (integer)
It is the number of rows in matrix type magazine 4.

N2921 No. of Columns in Matrix 4 (integer)
It is the number of columns in the matrix type magazine 4.

N2922 Spindle Magazines (S1...S16, bit, spindle)

N2922 S1...16	#7	#6	#5	#4	#3	#2	#1	#0
							SBY	**SPM**

#0 **SPM**: A spindle magazine can be assigned to each spindle. The tool, which is machining, is registered always in the spindle magazine of the Tool Pot table. The magazine number of the first spindle is always 10, the magazine number of the second spindle is 20, etc. If e.g. SPM bit is set for spindle 2, a new row will be added to the Tool Pot table.
If the parameter value:
=0: The given spindle will not be registered in the Tool Pot table,
=1: The given spindle will be registered in the Tool Pot table.

#1 **SBY**: A standby magazine can be assigned to each spindle, if e.g. during tool change the tool is placed into an arm and is waiting for to get into the spindle. The standby magazine number belonging to the first spindle is 11, the one belonging to the second one is 21, etc. If e.g. SBY bit is set for spindle 2 to, a new row will be added to Tool Pot table.
If the parameter value
=0: no standby magazine belongs to the given spindle,
=1: a standby magazine will belong to the given spindle.

N2923 No. of Offset Code (L1...8, integer, channel)
The offset number cannot be referred to directly from a part program during tool life management. There are more tools in the Tool Management table with the same T code (type number) therefore different offset numbers are belonging to the same tool type numbers. An offset number can be specified on this parameter, which ensures an indirect reference to the offset number through the Tool Management table. According to the data number of the tool (row number of Tool Management table) being in the spindle, the NC will identify from the Tool management table the offset number belonging to the tool.
In a milling channel, if the parameter value is e.g.

99,

the offset number of the tool in spindle can be referred to by programming **H99** and **D99**.
In a lathe channel, if the parameter value is e.g.

99

the offset number of the tool can be referred to by programming Tnn99.
The tool number shall not be specified for the call of geometry offset number, and the number of digits on this parameter shall be equal to that of specified to call offset number.

N2924 M Code of Particular T (L1...8, integer, channel)

A particular M code can be determined on this parameter which modifies the meaning of address T if it is programmed together with T in a block.
In this case address T doesn't specify a tool type but a magazine and a pocket number, and the NC will pass these data to the appropriate CN_MGZNO and CN_POTNO PLC registers. Then the upper 4 digits of address T specifies the magazine number while the lower 4 digits the pocket number.
In case e.g. the parameter value M Code of Particular T=62, the meaning of the block

M62 T10027

will be the following: get the tool being in magazine No. 1 and in pocket No. 27.

31 Skip Function Parameter Group

N3000 Sensor Input of G31 (L1...8, integer, channel)

Not used.

N3001 G31 Config (L1...8, bit, channel)

N3001 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
								SKF

#0 **SKF**: G31 skip function if the parameter value

=0: uses the modal, or programmed F value,

=1: takes the feed value from parameter N0311 G31 Feed.

N3002 Sensor Input of G36, G37 (L1...8, integer, channel)

A probe input for automatic tool length measurement can be appointed per every channel from 1 to 8,

which will be used by the G37 function in a milling channel, respectively, by the G36 and G37 function in a lathe channel.

If the parameter value is 0, the function is not used.

N3003 G36, G37 Config (L1...8, bit, channel)

N3003 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
						TCA	**TMW**	**TLF**

#0 **TLF**: G36 or G37 functions if the parameter value

=0: uses the modal, or programmed F value,

=1: takes the feed value from parameter N0312 G37 Feed.

#1 **TMW**: G36, or G37 function will modify, if the parameter value is

=0: the geometry value of the selected compensation,

=1: the wear value of the selected compensation,

by the result of the measurement.

#2 **TCA**: G36, or G37 function will if the parameter value

=0: subtract the result of the measurement from the selected offset value,

=1: add the result of the measurement to the selected offset value.

N3004 Rapid Distance (L1...8, floating-point, channel)

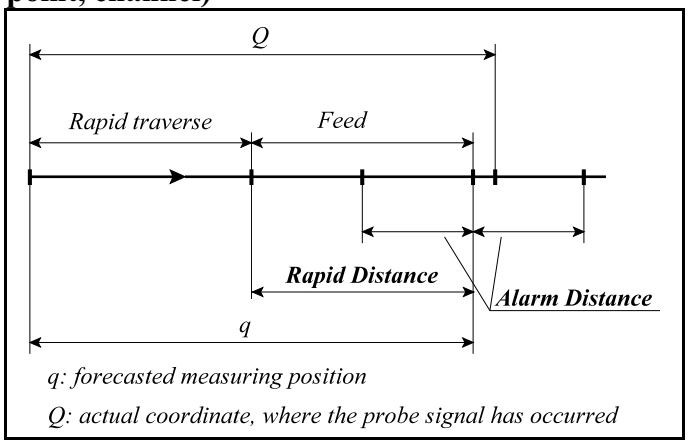
Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm.
In a milling channel, upon

G37 Zq

command, the distance determined on the parameter will be subtracted from the programmed end position coordinate

q - Rapid Distance,

and the axis will go to this position by rapid traverse.



N3005 Alarm Distance (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm.
If the automatic tool length measurement function in milling channel

G37 Zq

does not detect the probe signal inside the Alarm Distance environment of the programmed end position, i.e. if

|Q–q|>Alarm Distance,

it will send an alarm message.

N3006 Rapid Distance X (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm.
In a lathe channel, upon

G36 Xq

command, the distance determined on the parameter will be subtracted from the programmed end position coordinate

q - Rapid Distance,

and the axis will go to this position by rapid traverse.

N3007 Alarm Distance X (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm.
If the automatic tool length measurement function in lathe channel

G36 Xq

does not detect the probe signal inside the Alarm Distance X environment of the programmed end position, i.e. if

|Q–q|>Alarm Distance,

it will send an alarm message.

N3008 Rapid Distance Y (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm.
In a lathe channel, upon

G36 Yq

command, the distance determined on the parameter will be subtracted from the programmed end position coordinate

q - Rapid Distance,

and the axis will go to this position by rapid traverse.

N3009 Alarm Distance Y (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm.
If the automatic tool length measurement function in lathe channel

G36 Yq

does not detect the probe signal inside the Alarm Distance Y environment of the programmed end position, i.e. if

|Q–q|>Alarm Distance,

it will send an alarm message.

N3010 Rapid Distance Z (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm.
In a lathe channel, upon

G36 Zq

command, the distance determined on the parameter will be subtracted from the programmed end position coordinate

q - Rapid Distance,

and the axis will go to this position by rapid traverse.

N3011 Alarm Distance Z (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on #0 IND parameter shall be interpreted in inch or mm.
If the automatic tool length measurement function in lathe channel

G37 Zq

does not detect the probe signal inside the Alarm Distance Z environment of the programmed end position, i.e. if

|Q–q|>Alarm Distance,

it will send an alarm message.

N3012 Sensor Input of Tool Setter S1 (L1...8, integer, channel)

A probe input can be specified per channel
from 1 to 8,
used for the **manual measurement of tool length offsets** when tool offsets belonging to spindle S1 are measured.
If the parameter value is 0, the function is not used.

N3013 Sensor Input of Tool Setter S2 (L1...8, integer, channel)

If a machine is equipped with a sub-spindle and a separate tool sensor belongs to S2 sub-spindle, a probe input per channel can be specified on this parameter

from 1 to 8,

used for the **manual measurement of tool length offsets** when tool offsets belonging to spindle S2 are measured.

If the parameter value is 0, the function is not used.

N3014 Sensor Input of Workpiece Setter S1 (L1...8, integer, channel)

A probe input can be specified per channel

from 1 to 8,

used for the **manual measurement of work zero offsets** when work offsets belonging to spindle S1 are measured.

If the parameter value is 0, the function is not used.

N3015 Sensor Input of Workpiece Setter S2 (L1...8, integer, channel)

If a machine is equipped with a sub-spindle and a separate workpiece sensor belongs to S2 sub-spindle, a probe input per channel can be specified on this parameter

from 1 to 8,

used for the **manual measurement of work zero offsets** when work offsets belonging to spindle S2 are measured.

If the parameter value is 0, the function is not used.

N3016 Tool/WP Setter Config (L1...8, bit, channel)

N3014 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
							WPS	**ONS**

#0 **ONS**: During manual measurement of tool length/work zero offset by a setter, the offset number, which will be compensated/which will be used for the calculation of the work offset, if the parameter value is

=0: is selected manually on the control panel,

=1: is selected by the PLC through CP_OFFSNO register.

#1 **WPS**: During manual measurement of work zero offset by a setter, the workpiece coordinate system to be measured, if the parameter value is

=0: is selected manually by the operator on the control panel,

=1: is selected by the control, based on the appropriate parameters Coordinate System No. of S1, Coordinate System No. of S2.

N3017 Coordinate System No. of S1 (L1...8, integer, channel)

During manual measurement of work zero offset by a setter and N3016 Tool/WP Setter Config parameter #1 WPS=1 is, this parameter will specify the number of workpiece coordinate system (G54, ..., G59) belonging to Spindle S1 which is compensated by the result of measurement.

Possible values: 54, 55, 56, 57, 58, 59.

N3018 Coordinate System No. of S2 (L1...8, integer, channel)

During manual measurement of work zero offset by a setter and N3016 Tool/WP Setter Config parameter #1 WPS=1 is, this parameter will specify the number of workpiece coordinate system (G54, ..., G59) belonging to Spindle S2 which is compensated by the result of measurement.
Possible values: 54, 55, 56, 57, 58, 59.

N3019 Servo Limit during Torque Limit Skip (A1...32, floating-point, axis)

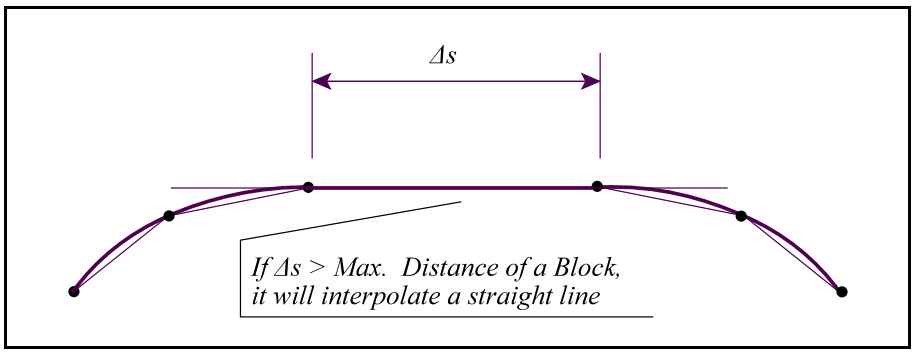
During measurement of the torque of an axis motor (execution of the G31 Q98 function), the control suspends monitoring the servo error in accordance with the parameter N0520 Serrl2 on the axis participating in the measurement, and takes the limit given here into account.

32 HSHP Control Parameter Group

N3100 Max. Distance of a Block (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on #0 IND parameter, shall be interpreted in inch or mm.

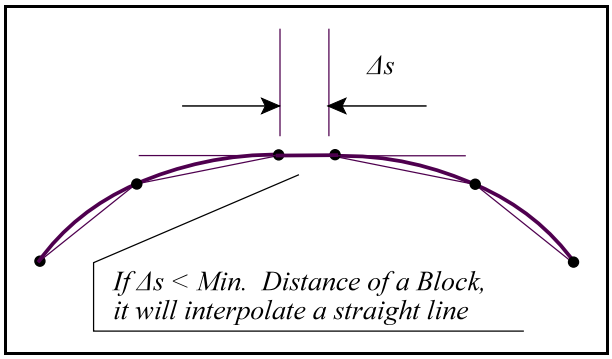
If the length of a straight line block (G1) is longer than the distance specified on this parameter, smooth interpolation (G5.1 Q2) is cancelled for that block. Blocks longer than the parameter value are executed with normal line interpolation.



N3101 Min. Distance of a Block (L1...8, floating-point, channel)

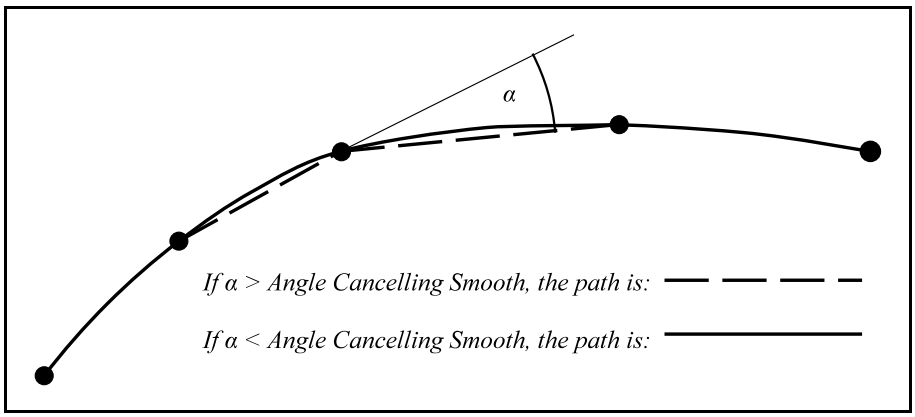
Distance data, the number written on the parameter, based on #0 IND parameter, shall be interpreted in inch or mm.

If the length of a straight line block (G1) is shorter than the distance specified on this parameter, smooth interpolation (G5.1 Q2) is cancelled for that block. Blocks shorter than the parameter value are executed with normal line interpolation.



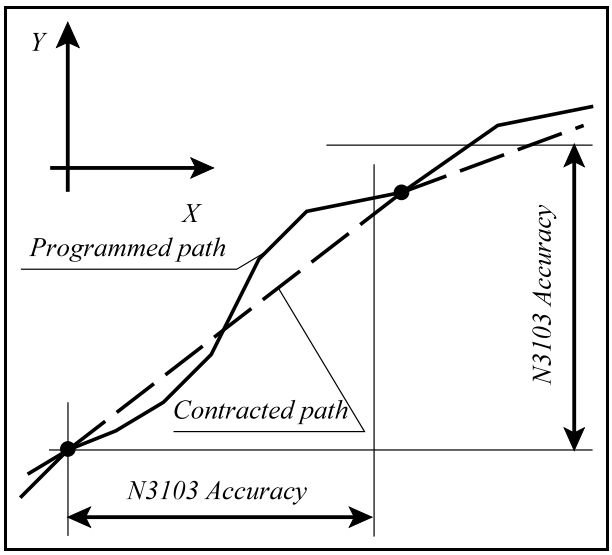
N3102 Angle Cancelling Smooth (L1...8, floating-point, channel)

Data of angle, the number written on the parameter shall be interpreted in degrees.
If the angle included by two straight line blocks (G1) is higher than the angle specified on this parameter, smooth interpolation (G5.1 Q2) is cancelled for the two blocks.



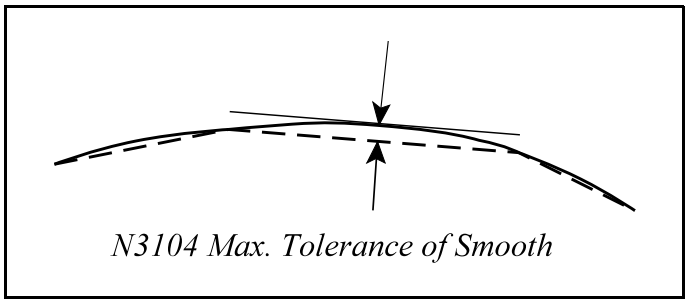
N3103 Accuracy (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on #0 IND parameter, shall be interpreted in inch or mm. It is a positive value.
The parameter serves to specify the accuracy level in High Speed High Precision machining mode (HSHP) (G5.1 Q>0).
G01 interpolation blocks specifying short straight lines are contracted so far as the movement on an axis becomes higher than the value set on this parameter. Then the contracted block will be executed.



N3104 Max. Tolerance of Smooth (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on #0 IND parameter, shall be interpreted in inch or mm.
When smooth interpolation is on (G5.1 Q2), the number written in the parameter informs about the maximum permissible tolerance of the finished path from the straight line in the course of finishing the programmed straight line segment. If the tolerance is greater than the specified value, it will not finish along the given segment, but it will interpolate a straight line.



N3104 Max. Tolerance of Smooth

N3105 Min. Tolerance of Smooth (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on #0 IND parameter, shall be interpreted in inch or mm.

When smooth interpolation is on (G5.1 Q2), the number written in the parameter informs about the minimum permissible tolerance of the finished path from the straight line in the course of finishing the programmed straight line segment. If the tolerance is smaller than the specified value, it will not finish along the given segment, but it will interpolate a straight line.

☞ ***Warning!***

*The parameters **N3100-N3103** are effective both for the smooth interpolation (**G5.1 Q2**) and the smooth interpolation of type 2 (**G5.1 Q3**).*

*The following parameters **N3106-N3108** are effective **only** in the case of the smooth interpolation of type 2 (**G5.1 Q3**).*

*In the case of the smooth interpolation of type 2 (**G5.1 Q3**), not only does the control connect the programmed points with a Bezier curve, but it moves the programmed points so that the curve will be as smooth as possible. Furthermore, in the case of 5D movements, when the tool center point is guided, it smooths movements of the rotary axes too.*

N3106 Tolerance of Smooth2 (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on #0 IND parameter, shall be interpreted in inch or mm.

In the case of the smooth interpolation of type 2 (G5.1 Q3), it remains within this tolerance even after moving the programmed points of the blocks.

N3107 Dist. Cancelling of Angular Calc. in Smooth2 (L1...8, floating-point, channel)

Distance data, the number written on the parameter, based on #0 IND parameter, shall be interpreted in inch or mm.

During the smooth interpolation of type 2 (G5.1 Q3), in any case, it smooths a segment shorter than this, independently of the parameter N3102 Angle Cancelling Smooth. The specified value should not be greater than the double value of the parameter N3106 Tolerance of Smooth2!

N3108 Tolerance of Rot. Ax. in Smooth2 (A1...16, floating-point, axis)

Unit: degree.

During the smooth interpolation of type 2 (G5.1 Q3), the smooth of the rotary axes remains within this tolerance for each rotary axis.

N3110 Min. Dist. on Rot. Ax. in Smooth2 (L1...8, floating-point, channel)

Not used.

N3111 Max. Ang. Diff. on Rot. Ax. in Smooth2 (L1...8, floating-point, channel)

Not used.

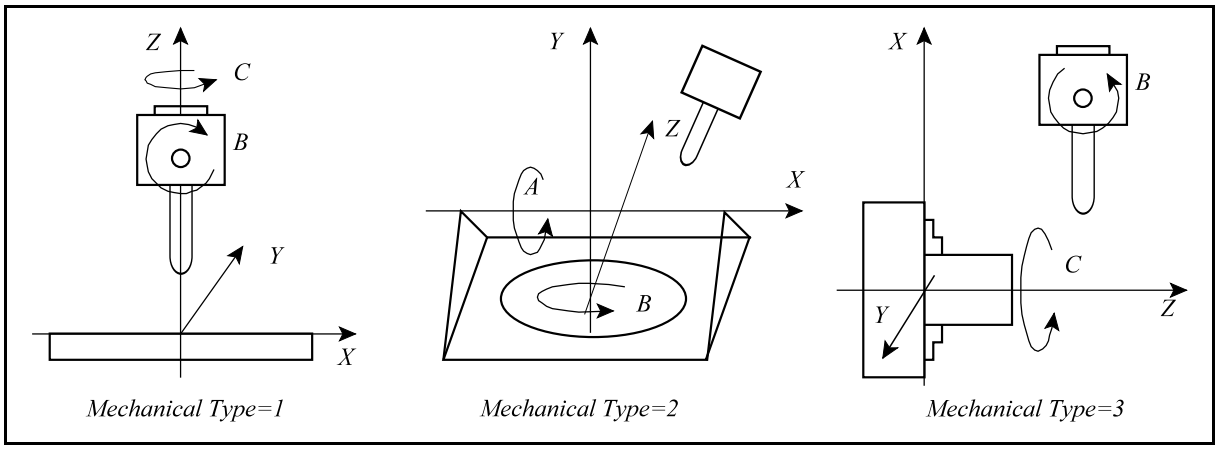
N3112 Dist. on Rot. Ax. Cancelling of Angular Calc. in Smooth2 (L1...8, floating-point, channel)

Not used.

33 5 Axis Machining Parameter Group

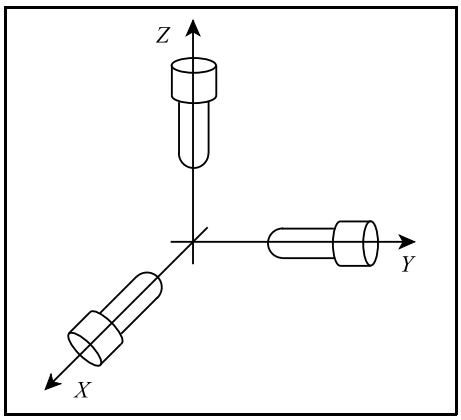
N3200 Mechanical Type (L1...8, integer, channel)

Its possible values: 0, 1, 2, 3.
If the parameter:
=0: 5D machining on the machine tool is not possible.
If it is possible, three types of machine can generally be distinguished for 5D machining. The type of the machine is defined by the number written in the parameter:
=1: Machine tool of head-head configuration: both rotary axes tilt the tool,
=2: Machine tool of table-table configuration: both rotary axes tilt the table,
=3: Machine tool of head-table configuration: one of the rotary axis rotates the tool, the other one rotates the table.



N3201 Tool Axis Direction (L1...8, integer, channel)

Its possible values: 1, 2, 3.
That axis has to be specified in this parameter, in the direction of which the tool points when the rotary axes involved in tool length compensation are in the 0 position. If the value of the parameter:
=1: direction is the axis X (the first main axis),
=2: direction is the axis Y (the second main axis),
=3: direction is the axis Z (the third main axis).
If the tool is tilted relative to the main direction defined in the parameter, the inclination angle of the tool can be specified in the Reference Angle 1 és a Reference Angle 2 parameters.



N3202 Reference Angle 1 (L1...8, floating-point, channel)
N3203 Reference Angle 2 (L1...8, floating-point, channel)

They are not used.

N3204 No. of the First Rot. Ax. (L1...8, integer, channel)

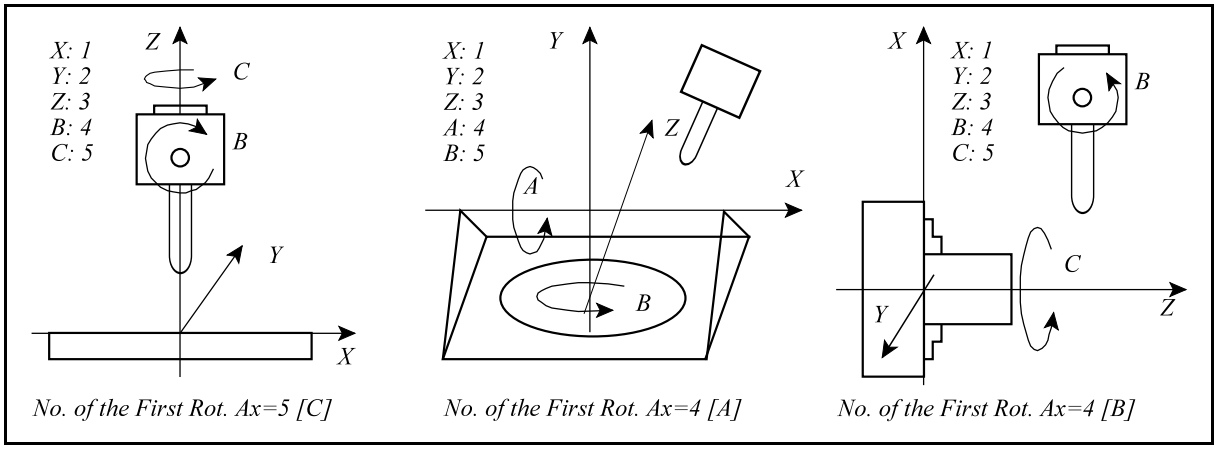
Its possible values: 0, 1, 2, ..., 32.

In the case of machine tools of head-head or table-table configuration, when 5D machining is executed, that rotary axis will be called first axis, which bears mechanically the second rotary axis. The axis number of the bearing axis has to be written in this parameter.

In the case of machine tools of head-table configuration, always the number of the rotary axis tilting the tool (the head) has to be written in this parameter.

If the first rotary axis does not exist and it is not involved in the 5D transformations, 0 will have to be written in this parameter and the IA1 bit of the 5D Control parameter.

If the first rotary axis cannot be rotated (it is an accessory adjusted to a fix angle position, e.g. a tilt head), 0 will have to be written in this parameter and 1 will have to be written in the IA1 bit of the 5D Control parameter. In this case, the angle position of the axis has to be taken from the Angle of the First Rot. Ax. Parameter.



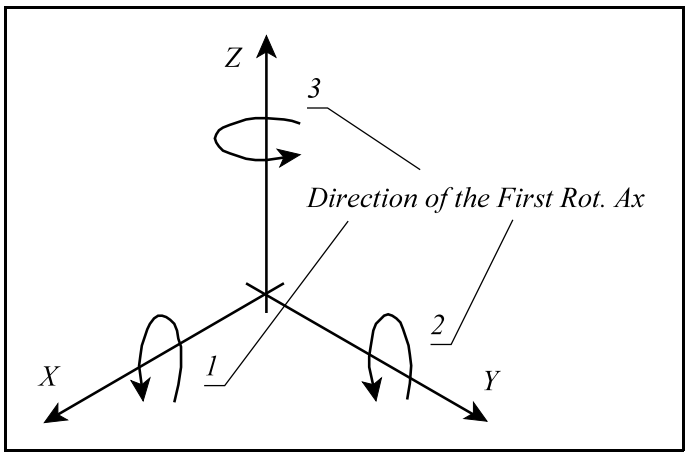
No. of the First Rot. Ax=5 [C] *No. of the First Rot. Ax=4 [A]* *No. of the First Rot. Ax=4 [B]*

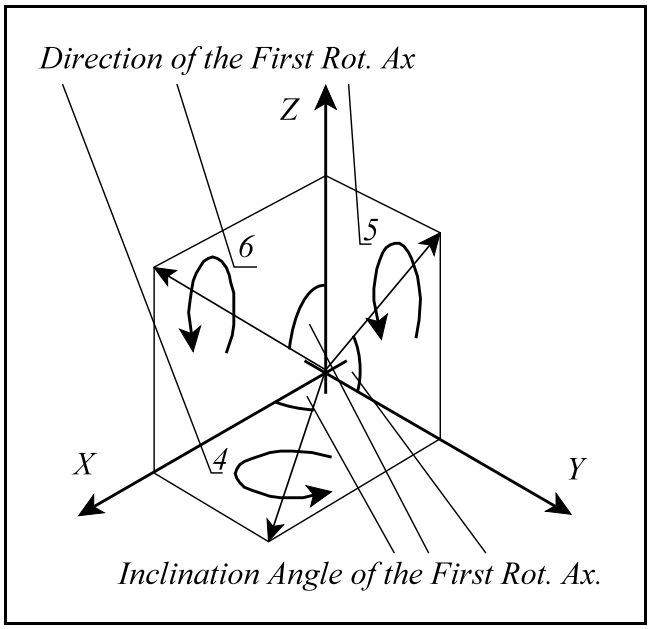
N3205 Direction of the First Rot. Ax. (L1...8, integer, channel)

Its possible values: 1, 2, ..., 6.

This parameter defines the direction of the first rotary axis, i.e. the linear main axis, around which it rotates:

=1: it rotates around the axis X (the first main axis),
=2: it rotates around the axis Y (the second main axis),
=3: it rotates around the axis Z (the third main axis),
=4: it is tilted relative to the axis X in the plane XY,
=5: it is tilted relative to the axis Yin the plane YZ,
=6: it is tilted relative to the axis Z in the plane ZX.



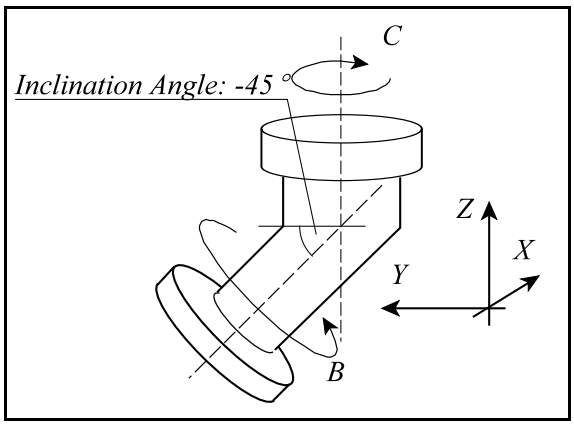


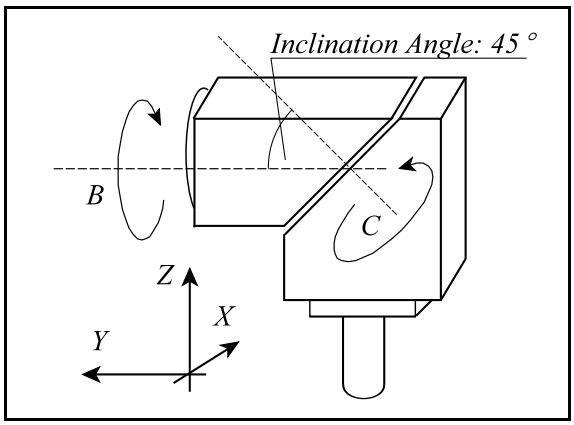
N3206 Inclination Angle of the First Rot. Ax. (L1...8, floating-point, channel)

Its unit of measurement: degree.

The value of this parameter will be 0 if the value of the N3202 Direction of the First Rot. Ax. parameter is 1, 2 or 3.

The inclination angle of the rotary axis can be specified in this parameter if the value of the N3202 Direction of the First Rot. Ax. parameter is 4, 5 or 6.





N3207 Angle of the First Rot. Ax. (L1...8, floating-point, channel)

Its unit of measurement: degree.

The angle position of the axis can be specified in this parameter if the value of the N3202 No. of the First Rot. Ax. parameter is 0 and the value of the IA1 of the 5D Control parameter 1, i.e. the first rotary axis cannot be moved (it is an accessory adjusted to a fix angle position, e.g. a tilt head).

N3208 No. of the Second Rot. Ax. (L1...8, integer, channel)

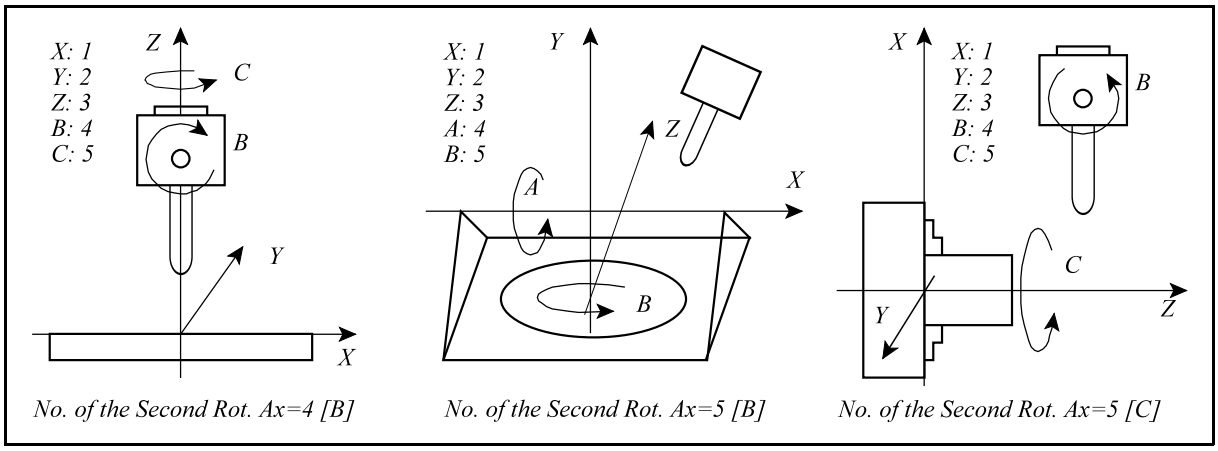
Its possible values: 0, 1, 2, ..., 32.

In the case of machine tools of head-head or table-table configuration, when 5D machining is executed, that rotary axis will be called second axis, which is borne mechanically by the first rotary axis. The axis number of the borne axis has to be written in this parameter.

In the case of machine tools of head-table configuration, always the number of the rotary axis tilting the workpiece (the table) has to be written in this parameter.

If the second rotary axis does not exist and it is not involved in the 5D transformations, 0 will have to be written in this parameter and the IA2 bit of the 5D Control parameter.

If the second rotary axis cannot be rotated (it is an accessory adjusted to a fix angle position, e.g. a tilt head), 0 will have to be written in this parameter and 1 will have to be written in the IA2 bit of the 5D Control parameter. In this case, the angle position of the axis has to be taken from the Angle of the Second Rot. Ax. Parameter.



N3209 Direction of the Second Rot. Ax. (L1...8, integer, channel)

Its possible values: 1, 2, ..., 6.

This parameter defines the direction of the second rotary axis, i.e. the linear main axis, around which it rotates:

=1: it rotates around the axis X (the first main axis),

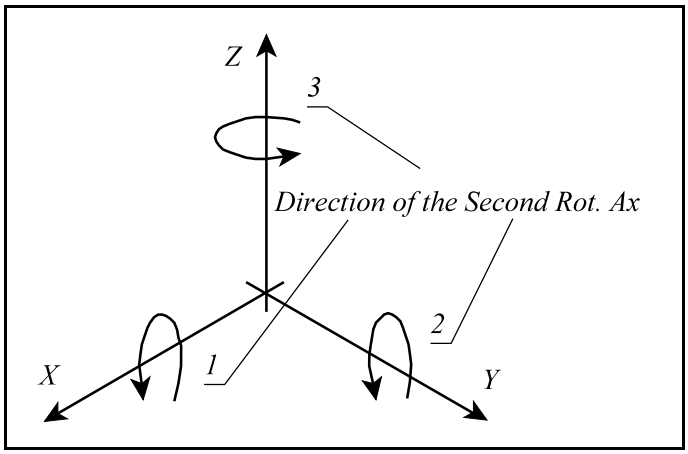
=2: it rotates around the axis Y (the second main axis),

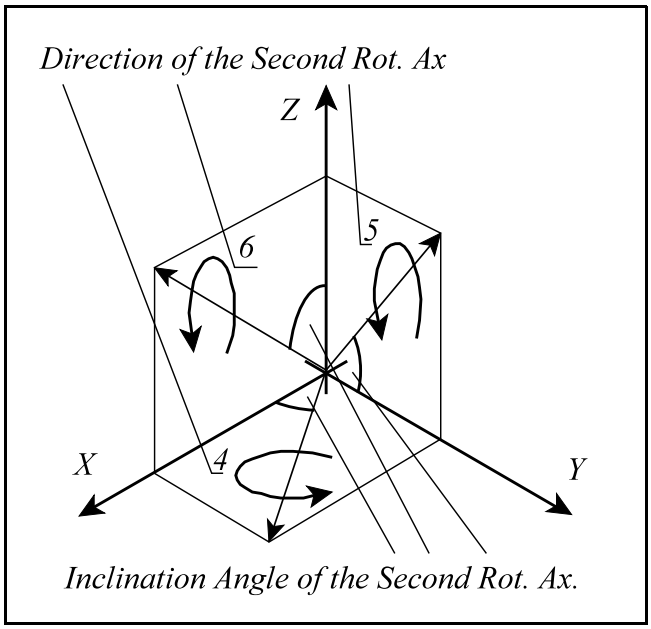
=3: it rotates around the axis Z (the third main axis),

=4: it is tilted relative to the axis X in the plane XY,

=5: it is tilted relative to the axis Yin the plane YZ,

=6: it is tilted relative to the axis Z in the plane ZX.



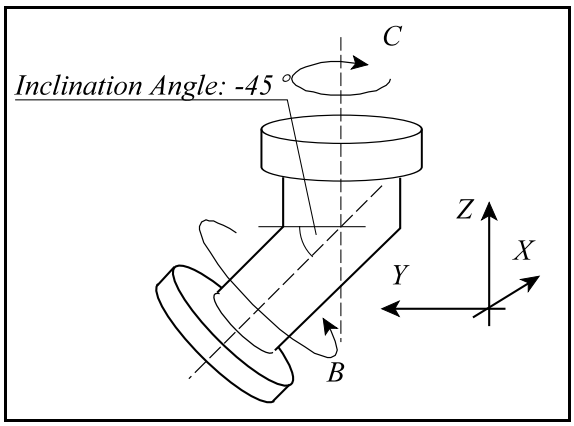


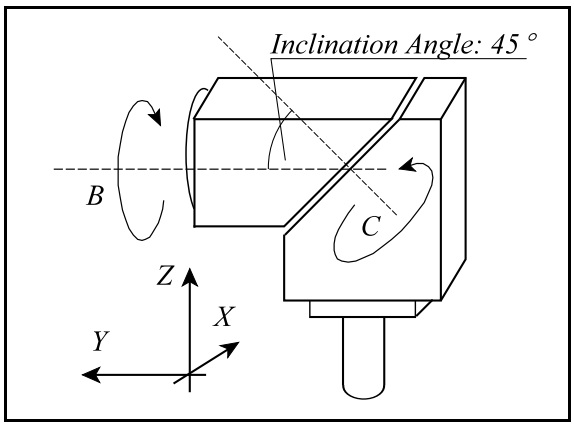
N3210 Inclination Angle of the Second Rot. Ax. (L1...8, floating-point, channel)

Its unit of measurement: degree.

The value of this parameter will be 0 if the value of the N3206 Direction of the Second Rot. Ax. parameter is 1, 2 or 3.

The inclination angle of the rotary axis can be specified in this parameter if the value of the N3206 Direction of the Second Rot. Ax. parameter is 4, 5 or 6.





N3211 Angle of the Second Rot. Ax. (L1...8, floating-point, channel)

Its unit of measurement: degree.

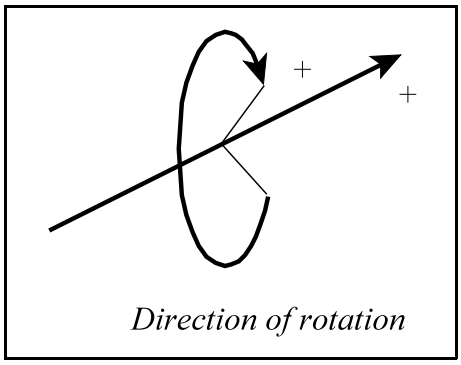
The angle position of the axis can be specified in this parameter if the value of the N3206 No. of the Second Rot. Ax. parameter is 0 and the value of the IA2 of the 5D Control parameter 1, i.e. the second rotary axis cannot be moved (it is an accessory adjusted to a fix angle position, e.g. a tilt head).

N3212 5D Control (L1...8, bit, channel)

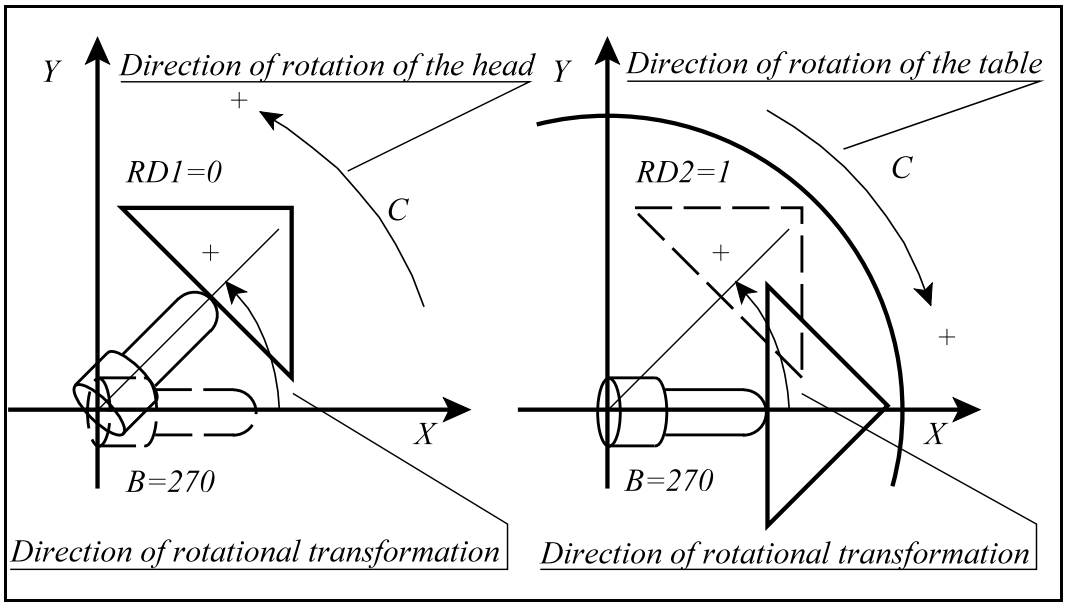
N3212 L1...8	#7	#6	#5	#4	#3	#2	#1	#0
	FOS	**WCS**	**DPC**	**TCF**	**RD2**	**RD1**	**IA2**	**IA1**

#0 **IA1**, #1 **IA2**: If the value of the No. of the First Rot. Ax. or No. of the Second Rot. Ax. parameter is 0, and the value of the IA1/IA2 parameter:
=0: the first/second rotary axis does not exist, it is not involved in the transformations,
=1: the first/second rotary axis is an axis with fixed position, and its angle position has to be taken from the Angle of the First Rot. Ax./Angle of the Second Rot. Ax. parameter.

#2 **RD1**, #3 **RD2**:

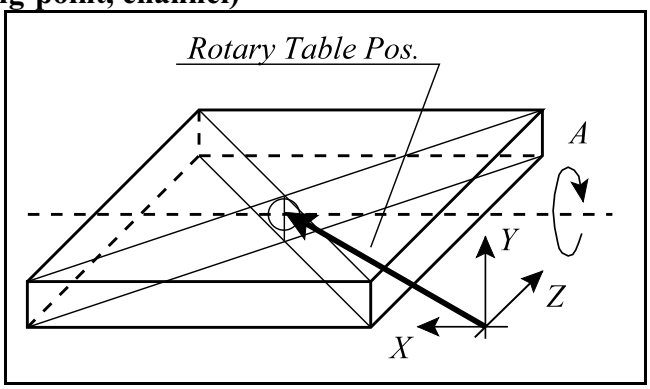


If one looks from the negative direction towards the positive direction along the linear axes defined in the Direction of the First Rot. Ax. and Direction of the Second Rot. Ax. parameters, the direction of rotational transformation around it will be positive, according to the right-hand rule. The value of the RD1, RD2 parameters will be:
=0: if the direction of rotation of the physical axis corresponds with the direction of the rotational transformation,
=1: if the direction of rotation of the physical axis is opposite to the direction of the rotational transformation.
The RD1 bit is related to the first rotary axis, and the RD2 bit is related to the second one. In general, the value of the parameter is 0 when the tool is rotated, and it is 1 when the table is rotated.



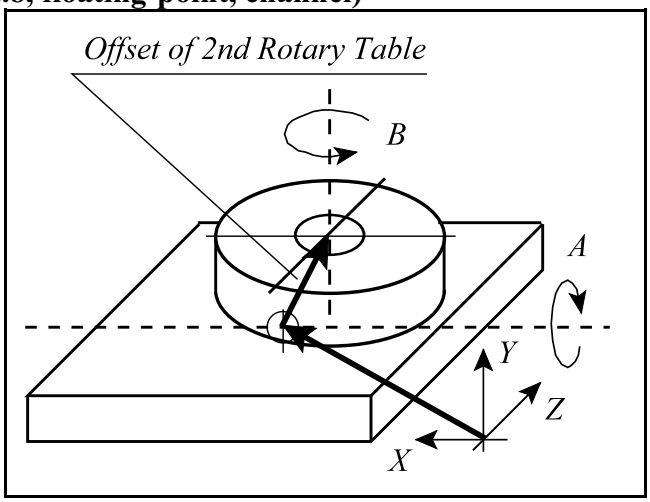
#4 **TCF**: It is not used.
#5 **DPC**: It is not used.
#6 **WCS**: It is not used.
#7 **FOS**: In the case of length compensation in tool axis direction (G43.1), dynamic zero point offset (G54.2) and execution of the G53.1 when the rotary axes participating in the function are moving, if the value of the parameter is:
=0: the linear axes will be also moving,
=1: the linear axes will not be moving, only the rotary axes will.

N3213 Rotary Table Pos. X (L1...8, floating-point, channel)
N3214 Rotary Table Pos. Y (L1...8, floating-point, channel)
N3215 Rotary Table Pos. Z (L1...8, floating-point, channel)



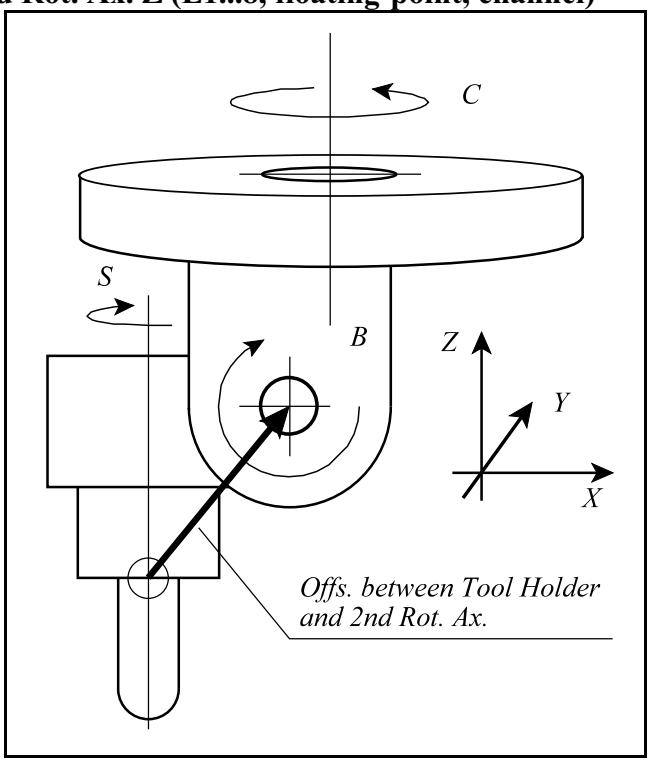
Its unit of measurement: mm or inch.
This parameter will have to be set if the value of the N3200 Mechanical Type parameter is 2 or 3, i.e. when the machine tool configuration is table-table or head-table.
In the case of table-table configuration, it has to be filled when the first rotary axis is measured (the axis number is given in the No. of the First Rot. Ax. parameter).
In the case of head-table configuration, it has to be filled when the second rotary axis is measured (the axis number is given in the No. of the Second Rot. Ax. parameter).
The position of the rotary table has to be given in these parameters, along the main axes designated in the channel, in the machine coordinate system.

N3216 Offset of 2nd Rotary Table X (L1...8, floating-point, channel)
N3217 Offset of 2nd Rotary Table Y (L1...8, floating-point, channel)
N3218 Offset of 2nd Rotary Table Z (L1...8, floating-point, channel)



Its unit of measurement: mm or inch.
This parameter will have to be set if the value of the N3200 Mechanical Type parameter is 2, i.e. when the machine tool configuration is table-table.
This parameter defines the position of the second rotary axis (the axis number is given in the No. of the Second Rot. Ax. parameter) relative to the measured position of the first rotary axis (Rotary Table Pos parameters).
The position of the second rotary table relative to the first one has to be specified along the main axes.

N3219 Offs. between Tool Holder and 2nd Rot. Ax. X (L1...8, floating-point, channel)
N3220 Offs. between Tool Holder and 2nd Rot. Ax. Y (L1...8, floating-point, channel)
N3221 Offs. between Tool Holder and 2nd Rot. Ax. Z (L1...8, floating-point, channel)



Its unit of measurement: mm or inch.
This parameter will have to be set if the value of the N3200 Mechanical Type parameter is 1, i.e. the configuration of the machine tool is head-head; or 3, i.e. the configuration of the machine tool is head-table..
When the configuration is head-head, the parameter defines the position of the second rotary axis (the axis number is given in the No. Of the Second Rot. Ax. parameter) from the tool holding point.
When the configuration is head-table, the parameter defines the position of the first rotary axis (the axis number is given in the No. Of the First Rot. Ax. parameter) from the tool holding point.
The distances have to be given along the main axes designated in the channel.

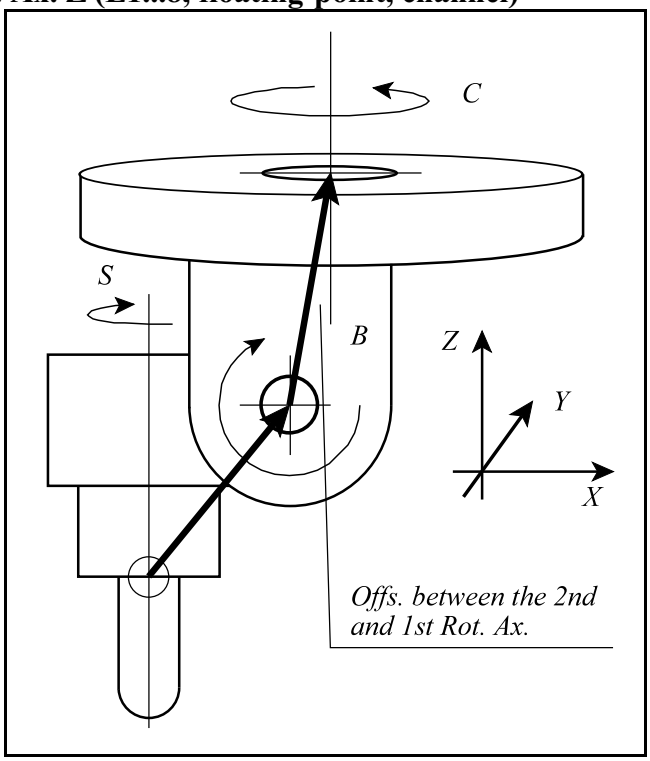
N3222 Offs. between the 2nd and 1st Rot. Ax. X (L1...8, floating-point, channel)
N3223 Offs. between the 2nd and 1st Rot. Ax. Y (L1...8, floating-point, channel)
N3224 Offs. between the 2nd and 1st Rot. Ax. Z (L1...8, floating-point, channel)

Its unit of measurement: mm or inch.

This parameter will have to be set if the value of the N3200 Mechanical Type parameter is 1, i.e. the configuration of the machine tool is head-head.

The parameter defines the position of the first rotary axis (the axis number is given in the No. Of the First Rot. Ax. parameter) from the second rotary axis.

The distances have to be given along the main axes designated in the channel.



N3225 Deviation Limit for Rapid (L1...8, floating-point, channel)
N3226 Deviation Limit for Feedrate (L1...8, floating-point, channel)

They are not used.

34 Pallet Management parameter group

N3300 Pallet Contr. (bit)

N3300	#7	#6	#5	#4	#3	#2	#1	#0
					PRI	**IDT**	**MNT**	**PAL**

#0 **PAL**: Enabling the use of the pallet managenet table:
=0: the control does not use the pallet management table;
=1: the control uses the pallet management table.

#1 **MNT**: Its value depends on the mechanical construction of the machine. There can also be a loading-unloading point beside the pallet magazine. In this case, there will be a distinct row for the loading-unloading point in the pallet management table. If the value of the parameter is:
=0: there is no loading-unloading point;
=1: there is loading-unloading point.

#2 **IDT**: It is not used

#3 **PRI**: In the pallet management table it can be specified which part program has to be used for machining the workpieces located on the pallets.
If, in the pallet management table, there is
=0: the programs are identified by 4-digit or 8-digit program numbers. The programs are stored in the root of the Programs directory under the file name Ooooo.nct or Ooooooooo.nct. They can also be written/read by PLC command.
=1: the programs are identified by their paths and file name. These data cannot be written/read by PLC command.

N3301 Pallet Pool Length (DWORD)

The value of the parameter depends on the mechanical construction of the machine. In the pallet management table, there is always a row that shows the data of the pallet being in the working place. Optionally, an loading-unloading point can also be designated (MNT=1) which shows in the table the data of the pallet being in the loading-unloading point. In addition, it can be determined here how many seats in the pallet magazine are available for the pallets, therefore, how many rows have to be added to the pallet management table for registration.

In the case of the two-pallet system where both pallets can be outside, the Pallet Pool Length=2. For example, the pallet is moved to the change position 1 using X-movement, and then it will be pushed out to the pallet magazine position 1. In this case, the working space is empty, and there are two pallets in the magazine. Then, the pallet 2 will be carried in from the change position 2 using X-movement. In this case, the magazine position 2 in the pallet magazine is empty, and there is pallet in the working space.

In the case of the system with changing arm where a changing arm carries in the pallet having the workpiece to be machined from the pallet magazine, and carries out the pallet having the workpiece machined, the Pallet Pool Length=1. In this case, the pallet having

the workpiece machined comes into pallet magazine and the pallet having the workpiece to be machined comes into the working space.

N3302 No. of Custom Columns (DWORD)

The possible values of the parameter are: 0, ..., 4

Max. 4 custom columns can be added to the pallet magazine which can also be edited manually or can be written or read using PLC command.

In the first custom column, it is possible to set eight bit data. It will always appear if the value of the parameter is 1~4.

In the custom columns 2 to 4, it is possible to set floating-point data.

35 Emulator Parameter Group

N5000 NC Mode (integer)

A parameter which cannot be set by the user. It is mandatory to keep it in a

Real Run

state.

In every other cases, NCT Ltd. cannot guarantee the safety of the machine and the operator.

36 PC Settings Parameter Group

N5100 Date_Time (Month/Day/Year Hour:Minute:Second AM or PM)

It carries out the setting of the calendar and clock built into the control. The interpretation of numbers is carried out in the above indicated order.
Clock can be set by
– a 12-hour setting, in which case AM or PM is to be typed in,
– a 24-hour setting, in which case AM and PM is not to be typed in.

N5101 IP Address (nnn.nnn.nnn.nnn)

The Internet Protocol (IP) address received from the system administrator or internet service provider is specified here. The IP-address is a 32-bit number, the designation of which is four numbers between 0 and 255 separated by points.

N5102 IP Mask (nnn.nnn.nnn.nnn)

The number of the sub-network mask received from the system administrator or internet service provider is specified here. This number, together with the IP-address identifies the network segment the computer belongs to.
The sub-network mask is a 32-bit number, the designation of which is four numbers between 0 and 255 separated by points. The default values of the sub-network mask are usually numbers 0 and 255 (for example: 255.255.255.0), but there may occur other number values, as well.

N5103 IP GateWay (nnn.nnn.nnn.nnn)

The IP-address of the default gateway to be added can be specified here. This is the address of the local IP-router being on the same network as the computer, which forwards the traffic to addresses beyond the local network. The values in the individual fields shall be between 0 and 255.
The gateway is a router which connects separate IP-network segments. For example, via a network segment gateway it may be connected to another network segment, a wide-spread network or the Internet.

N5104 IP Primary DNS (nnn.nnn.nnn.nnn)

The IP-address of the primary DNS-server of the computer can be specified here. At first, it is the server by which the assignment of those DNS names, queried by the computer, to IP-addresses, is carried out, for the resolution of which there are no local name resolution information (DNS-names or names in the Hosts file) available.

N5105 IP Secondary DNS (nnn.nnn.nnn.nnn)

The IP-address of the secondary DNS-server of the computer can be specified here. The system will use this server if the DNS-server indicated in the primary DNS-server field is not available or if it is not able to decode the DNS-names queried by the computer to IP-addresses.

N5106 IP Primary WINS (nnn.nnn.nnn.nnn)

It is the IP address of the primary WINS (Windows Internet Name Service) server.

N5107 IP Secondary WINS (nnn.nnn.nnn.nnn)

It is the IP address of the secondary WINS (Windows Internet Name Service) server.

N5108 IP DHCP (integer)

If the parameter Value is

=0: the IP addresses of the control shall be distributed manually,

=1: the IP address will be received by the control from a DHCP (Dynamic Host Configuration Protocol) server, dynamically.

N5109 Language (integer)

The necessary language shall be selected from a drop-down menu. A restart will be needed.

N5110 Keyb Init Delay (integer)

It is effective only if a keyboard is attached to the control. It is not relevant to the touchscreen.

After pushing a keyboard button, it sets the waiting time, after which it starts repeating. If button is kept pushed, it will wait for a short time at small values, and for a longer time at higher values.

Value limit: 250...1000

The parameter will be effective only after power off the control.

N5111 Keyb RepeatRate (integer)

It is effective only if a keyboard is attached to the control. It is not relevant to the touchscreen.

After pushing a keyboard button, it sets the repetition rate. If button is kept pushed, the repetition will be slow at smaller values and fast at higher values.

Value limit: 2...30

The parameter will be effective only after power off the control.

Notes

Alphabetical Index