HELLER

Programming Instructions

MC 2000 - MC 14000, HMI Automotive, SINUMERIK 840D sl

PA.000066-EN-00

Contract data	
Designation	Machining centre
Machine type	MC 2000 - MC 14000, HMI Automotive,
Control	SINUMERIK 840D sl

Programming Instructions

These programming instructions describe additional functionalities that have been developed by HELLER beyond the standard scope of the Sinumerik 840D. The basic principles of programming are described in the Siemens documents "Programming Manual - Fundamentals" and "Programming Instructions - Production Engineering". These documents are frequently referred to in these Instructions.

Intended readership

The programming instructions are intended for both the user as owner, programmer and operator of the machine and for all those who are concerned with the programming of the machine. It must be made available to this group of persons.

Preconditions

The operators of this machine should be trained or otherwise qualified to operate machine tools.

Safety

Throughout the document there are instructions regarding work safety, which are important for avoiding harm to health and loss of life.

Anyone required to work with this machine must have read and understood the section entitled SAFETY in this manual.

Gebr. HELLER Maschinenfabrik GmbH

Postfach 14 28
D-72604 Nürtingen

Gebrüder-Heller-Straße 15
D-72622 Nürtingen

Telefon: +49-(0)70 22-77-0
Telefax: +49-(0)70 22-77-5610
E-Mail: sales@hnt.heller-machines.com
Internet: http://www.heller-machinetools.com

Contents

CHAPTER 1

NC machining program structure

1 NC machining program structure

1.1 Progress marker

1.1.1 Intended application

The control uses the progress marker for:

- Reselecting the last machining step.
- Handling of header specifications (e.g. runout, transport cycle, machining cycle, ...)

1.2 Syntax and format of the progress marker and entry label

Cycles syntax

Principal syntax distinction in accordance with the program level.

cs_step	cs_set_step	Main program level
cs_step1	cs_set_step1	1. Sub-program level
cs_step2	cs_set_step2	2. Sub-program level

1.2.1 Progress marker in the main program level

Obligatory entry label

The following entry labels must be part of each machining program:

STEP998	Entry if workpiece is set to NOK by the operator.
STEP999	Entry if workpiece is finished.

Setting the progress marker

The progress marker is stored as a real parameter.
Examples of progress markers:
- `cs_set_step(10)`
- `cs_set_step(33.1)`

An entry label can contain letters, numbers or underscore. The entry label begins with STEP, to which the real value of the progress marker is appended. Care must be taken to ensure that any decimal places contained in the progress marker are underscored. The label is followed by a colon, example Entry label:
- `STEP10:`
- `STEP33_1:`

Reading the progress marker

Example: `R0=sr_step`

1.2.2 First sub-program level progress marker

Setting the progress marker

The progress marker is stored as a real parameter.
Examples of progress markers:
- `cs_set_step1(11)`
- `cs_set_step1(11.1)`

An entry label can contain letters, numbers or underscore. The entry label begins with STEP, to which the real value of the progress marker is appended. Care must be taken to ensure that any decimal places contained in the progress marker are underscored. The label is followed by a colon, example Entry label:
- `STEP11:`
- `STEP11_1:`

Reading the progress marker

Example: `R0=sr_step1`

End of sub-program and progress marker

The sub-program has to end with `cs_set_step1(0)`.

For example see:
"First sub-program level program structure" page 27

1.2.3 Second sub-program level progress marker

Setting the progress marker

The progress marker is stored as a real parameter.
Examples of progress markers:
- `cs_set_step2(13)`
- `cs_set_step2(13.1)`

An entry label can contain letters, numbers or underscore. The entry label begins with STEP, to which the real value of the progress marker is appended. Care must be taken to ensure that any decimal places contained in the progress marker are underscored. The label is followed by a colon, example Entry label:

- `STEP13:`
- `STEP13_1:`

Reading the progress marker

Example: `R0=sr_step2`

End of sub-program and progress marker

The sub-program has to end with `cs_set_step2(0)`.

The principles of the procedure are described in chapter: "First sub-program level program structure" page 27

1.3 Integrating the branch distributor into the machining program

cs_step

Main program level

The "cs_step" cycle must be programmed before the first machining block and before the first entry label. This cycle branches to the corresponding entry label based on the current progress marker. Local variables, for example, can be set before "cs_step".

cs_step1

1. Sub-program level

The "cs_step1" cycle must be programmed before the first machining block and before the first entry label. This cycle branches to the corresponding entry label based on the current progress marker. Local variables, for example, can be set before "cs_step1".

cs_step2

2. Sub-program level

The "cs_step2" cycle must be programmed before the first machining block and before the first entry label. This cycle branches to the corresponding entry label based on the current progress marker. Local variables, for example, can be set before "cs_step2".

1.4 Machining program structure

1.4.1 Principal program structure

```
         cs_step
STEP1:
         ;Machining step 1
         ...
         cs_set_step(2) ;next step
STEP2:
         ;Machining step 2
         ...
         cs_set_step(3) ;next step
STEPn:
         ;Machining step n
         ...
STEP998:
         M770 ; machining complete
         cs_set_step(999); end of machining
STEP999:
         M60 ; workpiece change
         M30 ; end of program
         ...
HOME:
         ;Home position movement
         ...
         cs_home
         M30
         ...
Traverse path:
         ...
         M30
```

1.4.2 Optimised program structure

```
        ;Restart machining
        cs_step
STEP1:
        ;Preparation for re-entering machining step 1
        ...
STEP_1: ;Machining step 1
        ...
        cs_set_step(2)
        gotof STEP_2
STEP2:  ;Preparation for re-entering machining step 2
        ...
STEP_2: ;Machining step 2
        ...
        cs_set_step(n)
        gotof STEP_n
STEPn:  ;Preparation for re-entering machining step n
        ...
STEP_n: ;Machining step n
        ...
STEP998:
        M770 ; machining complete
        cs_set_step(999)
STEP999:
        ;Preparation for workpiece/tool change
        M60 ; workpiece change
        M30 ; end of program
        ...
HOME: home position movement:
        ...
        cs_home
        M30
Traverse path:
        ...
        M30
```

1.4.3 First sub-program level program structure

```
         Sub-program machining
         cs_step1
STEP1:   ;Machining step 1
         ...
         cs_set_step1(2) ;next step
STEP2:   ;Machining step 2
         ...
         cs_set_step1(3) ;next step
         ...
         ...
STEPn:
         ;Machining step n
         ...
         cs_set_step1(0)
         M17 ; end of program
```

1.5 Variables for operating modes

Main process variables

The main process variable can be checked for its current value at any time. However, this causes a main process/advance process synchronisation and hence time losses.

Advance process variables

The advance variables reflect only the value of the operating modes and their auxiliary functions at the time of the NC program start.

Operating mode	Main process variables	Advance process variables
VK	i_vk	gd_vk
EB	i_eb	gd_eb
ES	i_es	gd_es
ER	i_er	gd_er
AUTO	i_auto	gd_auto
MDA	i_mda	gd_mda
JOG	i_jog	gd_jog
REF	i_ref	
Runout	i_leerf	
Unl cycles	i_leert	gd_leert
Move to home position	i_gstf	gd_gstf
Stop at end of cycle	i_hte	
Traverse path	i_bahnf	gd_bahnf
Measuring equipment capability	i_meas	gd_meas_device
Cont op	i_cont_run	gd_cont_run

HELLER

1.6 Variables for workpiece data in the work area

Main process variables

The main process variable can be checked for its current value at any time. However, this causes a main process/advance process synchronisation and hence time losses.

Advance process variables

The advance process variables reflect the value of the workpiece and/or workpiece type at the time of the NC program start.

Information	Information flow	Advance process variables	Main process variables	Type
Workpiece ID		gd_wp_id1	id_wp_id1	DINT
Workpiece type	PLC->NC	gd_wp_typ1	ib_wp_typ1	BYTE
Operation index	PLC->NC	gd_op_index1	iw_op_index1	INT
Workpiece present	PLC->NC Bit 0 = 1, wp 1 present Bit 1 = 1, wp 2 present Bit 2 = 1, wp 3 present Bit 3 = 1, wp 4 present	gd_wp_present1	ib_wp_present1	BYTE
Workpiece OK *1	PLC->NC Bit 0 = 1, wp 1 OK Bit 1 = 1, wp 2 OK Bit 2 = 1, wp 3 OK Bit 3 = 1, wp 4 OK	gd_wp_io1	ib_wp_io1	BYTE
Workpiece is to be measured	PLC->NC Bit 0 = 1, measure wp 1 Bit 1 = 1, measure wp 2 Bit 2 = 1, measure wp 3 Bit 3 = 1, measure wp 4	gd_wp_meas1	ib_wp_meas1	BYTE
Machine NOK workpiece	PLC->NC Bit 0 = 1, wp 1 NOK, machine Bit 1 = 1, wp 2 NOK, machine Bit 2 = 1, wp 3 NOK, machine Bit 3 = 1, wp 4 NOK, machine	gd_wp_nio_processed1	ib_wp_nio_processed1	BYTE
Special info for workpiece	PLC->NC Bit 6: OP index completely machined. Bit 7: Oversize machining	 gd_op_index_finished1 gd_wp_oversize_cut1	 i_op_index_finished1 i_wp_oversize_cut1	BOOL

*1 If there are workpieces available, but none of them is OK, no machining operation is performed. The program is then accessed in STEP998. As a result, machining is set to "finished" and a workpiece change is performed.

*1 If the workpieces are already finish machined, no machining operation is performed. The program is then accessed in STEP999 and a workpiece change is performed immediately.

1.7 Variables for workpiece data on the loading station

Main process variables

The main process variable can be checked for its current value at any time. However, this causes a main process/advance process synchronisation and hence time losses.

Information	Information flow	Main process variables	Type
Workpiece ID		id_rp_wp_id1	DINT
Workpiece type	PLC->NC	ib_rp_wp_typ1	BYTE
Operation index	PLC->NC	iw_rp_op_index1	INT
Workpiece present	PLC->NC Bit 0 = 1, wp 1 present Bit 1 = 1, wp 2 present Bit 2 = 1, wp 3 present Bit 3 = 1, wp 4 present	ib_rp_wp_present1	BYTE
Workpiece OK *1	PLC->NC Bit 0 = 1, wp 1 OK Bit 1 = 1, wp 2 OK Bit 2 = 1, wp 3 OK Bit 3 = 1, wp 4 OK	ib_rp_wp_io1	BYTE
Workpiece is to be measured	PLC->NC Bit 0 = 1, measure wp 1 Bit 1 = 1, measure wp 2 Bit 2 = 1, measure wp 3 Bit 3 = 1, measure wp 4	ib_rp_wp_meas1	BYTE
Machine NOK workpiece	PLC->NC Bit 0 = 1, wp 1 NOK, machine Bit 1 = 1, wp 2 NOK, machine Bit 2 = 1, wp 3 NOK, machine Bit 3 = 1, wp 4 NOK, machine	ib_rp_wp_nio_processed1	BYTE
Special info for workpiece	PLC->NC Bit 6: OP index completely machined. Bit 7: Oversize machining	 i_rp_op_index_finished1 i_rp_wp_oversize_cut1	BOOL

*1 If there are workpieces available, but none of them is OK, no machining operation is performed. The program is then accessed in STEP998. As a result, machining is set to "finished" and a workpiece change is performed.

*1 If the workpieces are already finish machined, no machining operation is performed. The program is then accessed in STEP999 and a workpiece change is performed immediately.

HELLER

1.8 Variables for work piece information

NC data can be read and written by means of the system variables.

Current machining of pallet in work area

The variable describes here refer to the current machining and the pallet that is located in the work area.

Advance process variables

Advance process variables are always executed as advanced processes and reflect the value that the work piece or the work piece type has when the NC program starts.

Main process variables

Main process variables always acquire their value based on the current state of the main process.
If main process variables occur in a parts program the flow is stopped until the preceding block is processed and the values of the main process variables are available.

PAV = Pallet management

Information	Information flow	Flow (V) Main process (H)		Type
NC INFO Zero point offset pallets	PAV -> PLC -> NC	gv_palnv	V	STRING[78]
NC INFO Zero point offset workpiece	PAV -> PLC -> NC	gv_beanv	V	STRING[78]
Pallet number	PAV -> PLC -> NC	gm_palnr	H	INT
Workpiece number and machining index (No. of the current machining)	PLC -> NC	gm_beaind	H	INT
Pallet assignment (machining with machining status "unmachined" or "partly machined" and not "disabled" in bit string form)	PAV -> PLC -> NC	gm_beastat	H	INT
NC- Parameters pallet based	PAV <-> PLC <-> NC Is read before the pallet is exchanged.	gm_palncpara	H	INT
NC parameter machining-specific	PAV <-> PLC <-> NC Are read at the end of each machining.	gm_beancpara	H	INT
Pallet status (bit coded)	PAV -> PLC -> NC Bit 0 = 1, unmachined Bit 1 = 1, partly machined Bit 2 = 1, finished Bit 3 = 1, reject Bit 7 = 1, empty	gm_palzu	H	INT

Information	Information flow	Flow (V) Main process (H)		Type
Additional pallet info (bit coded)	PAV -> PLC -> NC Bit 0 = 1, present Bit 1 = 1, disabled Bit 2 = 1, to measure	gm_palzi	H	INT
Machining status (bit coded) (current machining)	PAV -> PLC -> NC Bit 0 = 1, unmachined Bit 1 = 1, partly machined Bit 2 = 1, finished Bit 3 = 1, reject	gm_beazu	H	INT
Additional machining info (bit coded)	PAV -> PLC -> NC Bit 0 = 1, data record created Bit 1 = 1, disabled Bit 2 = 1, to measure Bit 7 = 1, machining assigned	gm_beazi	H	INT
Loading type	PLC -> NC	gm_beltyp	H	INT
Machining bit string for influencing status	PLC <- NC	gm_beabl	H	INT

1.9 Variables for NC programming

The following variables are available for NC programming:

$MN_USER_DATA_FLOAT[0]	Machine number	read only	Type real
GM_PALPOS	Pallet change position	read only	Type int
i_m1_select	M01 selected	read only	Type Bool

1.10 Variable programming for users

1.10.1 Synchronous actions and real time variables

Synchronous Actions

Synchronous actions with ID addresses 1-15 are available to the user.

Real time variables

The following real time variables are available to the user:

$AC_MARKER	The user has $AC_MARKER[0-3] available.
$AC_PARAM	The user has $AC_PARAM[0-3] available.
$AC_TIMER	The user has $AC_TIMER[1-2] available.

1.10.2 R Parameter

The following R-parameters are available to the user:

R90...R99	Freely available for the user.
R100...R499	Freely available for the user.
R500...R599	Reserved for HELLER technology applications.
R600...R999	Freely available for the user.

CHAPTER 2

M functions, G functions

2 M functions, G functions

2.1 M functions

You may only use the M-functions specified in the list. However, it is possible that undocumented M-functions exist provoking unwanted reactions when in use.

Home position after NC reset

M Code	Version: Start of block	Version: Movement	Version: Block End	Group	See also	Explanation
M0			X	1		Programmed stop
M1			X	1		Optional stop
M2			X	1		End of program without return
M17			X	1		End of sub-program
M30			X	1		End of program with return
M3	X			2		Spindle in clockwise direction
M4	X			2		Spindle in counter-clockwise direction
M5	X			2		Spindle stop
M19			X	-		Spindle stop oriented to WZW-position
M70						Spindle in axis mode
M6			X	-		Tool change
M46			X	-		Slow tool change
M7		X		5		Work area shower On
M8		X		6		External coolant On
M9		X		5 - 9, 21	*1	Coolant Off
M20		X		7		IKM air
M21		X		7		IKM water (pressure stage M121 - M127)
M22						Reserved
M23		X		7		Exhaust tool
M28	X			21		Disable IKM flow monitor query
M34	X			12	*2	Normal clamping pressure
M35	X			12	*2	Clamping pressure reduced
M36	X			12	*2	No clamping pressure
M40	X			4	*3	Automatic gear shifting

M Code	Version			Group	See also	Explanation
	Start of block	Movement	Block End			
M41	X			4	*3	Gear step 1
M42	X			4	*3	Gear step 2
M43	X			4	*3	Gear step 3
M44	X			4	*3	Gear step 4
M50		X		8		MSK air
M51		X		-		MSK air pulse
M52		X		8		MSK water
M60			X	-		Pallet change
M61			X	-		Pallet change position 1
M62			X	-		Pallet change position 2
M63			X	-		Reserved
M64			X	-		Identify workpiece as reject
M72		X		11		Tool change door open
M73		X		11		Tool change door closed
M74	X			13		Release tool in the spindle for pickup
M75	X			13		Clamp tool in the spindle for pickup
M76		X		20	*7	Loading during machining Priority to provisioning (only for rack-type magazine)
M77		X		20	*8	Loading during machining Priority to loading (only for rack-type magazine)
M108		X		14		Coolant pump ON
M109	X					X/Y-flushing off
M121		X		9	*4	Coolant high pressure, stage 1
M122		X		9	*4	Coolant high pressure, stage 2
M123		X		9	*4	Coolant high pressure, stage 3
M124		X		9	*4	Coolant high pressure, stage 4
M125		X		9	*4	Coolant high pressure, stage 5
M126		X		9	*4	Coolant high pressure, stage 6
M127		X		9	*4	Coolant high pressure, stage 7
M134	X			15		Hydraulic workpiece clamping in work area Function additional line depressurised
M135	X			15		Hydraulic workpiece clamping in work area Additional line function connected including spindle disable

M Code	Version			Group	See also	Explanation
	Start of block	Movement	Block End			
M136	X			15		Hydraulic workpiece clamping in work area Additional line function connected Control line depressurized including spindle disable
M137	X			15		Hydraulic workpiece clamping in work area Additional line function connected
M138	X			15		Hydraulic workpiece clamping in work area Additional line function connected Control line depressurized
M142	X			16	*5	Hydraulic workpiece clamping in work area Clamping check off
M143	X			16	*5	Hydraulic workpiece clamping in work area Clamping check on
M200 + x (x=1, 31) automatic clamping axis x On, clamp axis x (*6)						
M201			X	31		Automatic clamping axis X on Clamp axis 1/X
M202			X	32		Automatic clamping axis Y on Clamp axis 2/Y
M203			X	33		Automatic clamping axis Z on Clamp axis 3/Z
M204			X	34		Automatic clamping axis A on Clamp axis 4/A
M205			X	35		Automatic clamping axis B on Clamp axis 5/B
M206			X	36		Automatic clamping axis C on Clamp axis 6/C
M207			X	37		Automatic clamping axis 7 on Clamp axis 7
M208			X	38		Automatic clamping axis 8 on Clamp axis 8
M209			X	39		Automatic clamping axis 9/CM on Clamp axis 9/CM
M210			X	40		Automatic clamping axis 10 on Clamp axis 10
M211			X	41		Automatic clamping axis 11/XM on Clamp axis 11/XM
M212			X	42		Automatic clamping axis 12/YM on Clamp axis 12/YM
M213			X	43		Automatic clamping axis 13/ZM on Clamp axis 13/ZM
M214			X	44		Automatic clamping axis 14/AM on Clamp axis 14/AM

M Code	Version			Group	See also	Explanation
	Start of block	Movement	Block End			
M215			X	45		Automatic clamping axis **15/CT on** Clamp axis 15/CT
M216			X	46		Automatic clamping axis **16/CT on** Clamp axis 16/CT
M217			X	47		Automatic clamping axis **17/RB on** Clamp axis 17/RB
M218			X	48		Automatic clamping axis **18 on** Clamp axis 18
M219			X	49		Automatic clamping axis **19/Z2 on** Clamp axis 19/Z2
M220			X	50		Automatic clamping axis **20 on** Clamp axis 20
M221			X	51		Automatic clamping axis **21 on** Clamp axis 21
M222			X	52		Automatic clamping axis **22 on** Clamp axis 22
M223			X	53		Automatic clamping axis **23 on** Clamp axis 23
M224			X	54		Automatic clamping axis **24 on** Clamp axis 24
M225			X	55		Automatic clamping axis **25 on** Clamp axis 25
M226			X	56		Automatic clamping axis **26 on** Clamp axis 26
M227			X	57		Automatic clamping axis **27 on** Clamp axis 27
M228			X	58		Automatic clamping axis **28 on** Clamp axis 28
M229			X	59		Automatic clamping axis **29 on** Clamp axis 29
M230			X	60		Automatic clamping axis **30 on** Clamp axis 30
M231			X	61		Automatic clamping axis **31 on** Clamp axis 31
M240 + x (x=1, 31) **automatic clamping off axis x**, release axis x						
M241			X	31		Automatic clamping axis **X off** Release axis 1/X
M242			X	32		Automatic clamping axis **Y off** Release axis 2/Y
M243			X	33		Automatic clamping axis **Z off** Release axis 3/Z
M244			X	34		Automatic clamping axis **A off** Release axis 4/A

M Code	Version			Group	See also	Explanation
	Start of block	Movement	Block End			
M245			X	35		Automatic clamping axis **B off** Release axis 5/B
M246			X	36		Automatic clamping axis **C off** Release axis 6/C
M247			X	37		Automatic clamping axis **7 off** Release axis 7
M248			X	38		Automatic clamping axis **8 off** Release axis 8
M249			X	39		Automatic clamping axis **9/CM off** Release axis 9/CM
M250			X	40		Automatic clamping axis **10 off** Release axis 10
M251			X	41		Automatic clamping axis **11/XM off** Release axis 11/XM
M252			X	42		Automatic clamping axis **12/YM off** Release axis 12/YM
M253			X	43		Automatic clamping axis **13/ZM off** Release axis 13/ZM
M254			X	44		Automatic clamping axis **14/AM off** Release axis 14/AM
M255			X	45		Automatic clamping **15/CT off** Release axis 15/CT
M256			X	46		Automatic clamping **16/CT off** Release axis 16/CT
M257			X	47		Automatic clamping axis **17/RB off** Release axis 17/RB
M258			X	48		Automatic clamping axis **18 off** Release axis 18
M259			X	49		Automatic clamping axis **19/Z2 off** Release axis 19/Z2
M260			X	50		Automatic clamping axis **20 off** Release axis 20
M261			X	51		Automatic clamping axis **21 off** Release axis 21
M262			X	52		Automatic clamping axis **22 off** Release axis 22
M263			X	53		Automatic clamping axis **23 off** Release axis 23
M264			X	54		Automatic clamping axis **24 off** Release axis 24
M265			X	55		Automatic clamping axis **25 off** Release axis 25
M266			X	56		Automatic clamping axis **26 off** Release axis 26

M Code	Version: Start of block	Version: Movement	Version: Block End	Group	See also	Explanation
M267			X	57		Automatic clamping axis 27 off Release axis 27
M268			X	58		Automatic clamping axis 28 off Release axis 28
M269			X	59		Automatic clamping axis 29 off Release axis 29
M270			X	60		Automatic clamping axis 30 off Release axis 30
M271			X	61		Automatic clamping axis 31 off Release axis 31
M318	X			0		NC/PLC synchronisation
M500 - M549						Available for customer-specific functions
M770		X		-		Info to PAV: Machining complete
M771		X		-		Info to PAV: Request loading system
M772		X		-		Info to PAV: Work piece must be measured

Home position after NC reset (shaded)

*1 **M9 all coolant functions off**

M9 turns off the coolant on all groups.

All coolant functions are turned off for tool changing. This operation can be controlled with the GV variable `GV_MEDIA`.

*2 **M34 - M36 variable clamping pressure setting**

The clamping pressure setting is made via a proportional valve on the machine.

M34 sets the clamping pressure recorded in the PAV pallet data base.
M34 becomes automatically effective at the beginning of the program.

A reduced clamping pressure (in bar) can be preselected using the variable $A_DBW[34]. The preselected clamping pressure is activated by M35.

M36 is used to release the clamping pressure. There is no changeover from clamped to released, the pressure is simply reduced to 0 bar.

*3 **M40, M41, M42 (M43, M44) gear step selection**

For machines with shift gearbox, the gear step can be selected with M41 to M42 or M41 to M44 (depending on the number of gear steps available.

The speed ranges and spindle torque moments assigned each gear step are indicated in the gear diagrams.

M40 automatic gear step selection
M40 automatically selects the gear step with the most suitable torque range based on the programmed speed. If the gear unit is already in this step, no gear change takes place.
M40 is the home position.
M40 becomes active at the start of the block.
In the block with M40, however, it must be expected that the spindle will be stopped to change the gear step.

M41 gear step 1
M41 changes to gear step 1 with the highest torque range. If the programmed speed is greater than the limiting speed of this gear step, an error message is given. If the gear unit is already in this step, no gear change takes place.
In the block with M41, however, it must be expected that the spindle will be stopped to change the gear step.

M42 gear step 2
M42 changes to gear step 2 with the corresponding torque range. If the programmed speed is greater than the limiting speed of this gear step, an error message is given. If the programmed speed is less than the limiting speed for step 1, gear step 2 will still be selected. The programmed speed is available with reduced torque.
In the block with M42, it must be expected that the spindle will be stopped to change the gear step.
If gear shiftings are to be totally prevented during machining, M42 can be programmed globally at the start of the program.

M43, M44 additional gear steps
If there are more than 2 changeable gear steps, these can be changed in a similar manner to M41, M42.

*4 **Internal coolant supply - pressure stages**
M121 ... M127 can be used to select different pressure stages on single-spindle machines and internal coolant feed IKM (M21) with optional high-pressure pump 7.
The pressure is set as shown in the following table (applies to 50 bar / 70 bar systems):

	Pressure in bar	Pressure in PSI
M121	5 / 10	72 / 143
M122	10 / 20	143 / 286
M123	15 / 30	215 / 430
M124	20 / 40	286 / 570
M125	30 / 50	430 / 720
M126	40 / 60	570 / 863
M127	50 / 70	720 / 1006

The pressure stages are distributed similarly for systems that have a higher maximum pressure

*5 **M142 pressure switch control line (4th line) off**
M142 suppresses pressure monitoring in the control line during a switchover using M135...M138.
The control line can be used to check the end positions of hydraulic movements. Pressure falls during the switchover process and only climbs again once the switchover process has been correctly completed.
For this purpose, the function M142 can be carried out before the switchover process. This prevents an error message during the switchover process when the pressure falls in the control line.

M143 pressure switch control line (4th line) on
M143 checks whether the pressure in the control line has climbed back to the pressure monitoring level after a temporary fall (during M142).
In addition to the convenient hydraulic function, this makes it possible to "safely" monitor device functions.

Programming example with monitoring:
Switchover of a swivelling device:
M134 ; not swivelled under normal conditions
M142 ; monitoring control line off
M135 ; switchover pulse pressure on 4th line, swivelling
G4 F2; Dwell time, swivelling time
M143 ; monitoring control line on
M134 ; normal status, no pressure on 4th line

*6 **M200 Read-in disable during clamping sequence active**
In case M200 is programmed in the same block as B-axis positioning, the following block is not read-in before the B-axis is fully clamped.
Application:

If strong tangential forces affect the B-axis during acceleration of the Z-axis, it may be necessary to wait until the B-axis clamping sequence is completed.

*7 **M76 loading during machining, priority to provisioning (only for rack-type magazine)**
Loading only possible when machining is in process in the 1st channel. Cycle time remains unchanged.

*8 **M77 loading during machining, priority to loading (only for rack-type magazine)**
Loading also possible when machining is in process in the 1st channel. Cycle time may be extended.

HELLER

2.2 G-functions - default setting after reset

The HELLER standard and Siemens standard vary in a few points.

Group	Description	Standard setting
1	Modal effective movement commands	G0 HELLER standard
2	Block-by-block effective movements, dwell time	-
3	Programmable frame, working area limitation and polar coordinate programming	-
4	FIFO	STARTFIFO
6	Level selection	-
7	Tool radius offset	G40
8	Adjustable zero point offset	G500
9	Frame suppression	-
10	Exact positioning - path control mode	G60
11	Exact positioning block-by-block	-
12	Block changing criteria during exact positioning (G60/G09)	G601
13	Workpiece dimensioning, imperial / metric	G71
14	Workpiece dimensioning absolute/incremental	G90
15	Feed type	G94
16	Feed correction and interior and exterior curvature	CFTCP HELLER standard
17	Approach behaviour, retract behaviour tool offset	NORM
18	Corner behaviour tool compensation	G450
19	Curve transition at spline start	BNAT
20	Curve transition at spline end	ENAT
21	Acceleration pattern	SOFT HELLER standard
22	Tool compensation types	CUT2D
23	Collision monitoring on inner contours	CDOF
24	Pilot control	FFWOF
25	Reference tool orientation	ORIWKS #
26	Re-approach point for REPOS	RMI
27	Tool compensation at change of orientation on the exterior corners	ORIC #
28	Working area limitation on/off	WALIMON
29	Radius - diameter	DIAMOF
30	Compressor on/off	COMPOF #
31	User – G group	G810 #
32	User – G group	G820 #

Group	Description	Standard setting
33	Adjustable fine tool compensation	FTOCOF #
34	Subsequent grinding tool orientation	OSOF #
35	Punching and nibbling	SPOF #
36	Punching with delay	PDELAYON #
37	Feed pattern	FNORM #
38	Assignment rapid inputs/outputs for punching/nibbling	SPIF1 #
39	Programmable contour accuracy	CPRECOF
40	Tool radius offset constant	CUTCONOF
41	Thread cutting interruption	LFOF
42	tool carrier	TCOABS
43	Approach direction WAB	G140
44	Path division WAB	G340
45	Path reference of the FGROUP axes:	SPATH
46	Level definition for fast retraction:	LFTXT
47	Mode switchover for external NC code	G290
48	Approach/retract behaviour WRK	G460
49	Point-to-point movement	CP
50	Orientation programming	ORIEULER
51	Orientation interpolation	ORIVECT
52	Workpiece based WKS	PAROTOF
53	Frame rotation on tool direction	TOROTOF
54	Rotation of the rotation axis	ORIROTA
55	Rapid traverse with/without linear interpolation	RTLION
56	Inclusion of the tool wear	TOWSTD
57	Automatic corner override	FENDNORM
58	Reserved for clearing out of the software end position	RELIEVEOF
59	Technology – G group	DYNNORM
60	Working area limitation	WALCS0

CHAPTER 3

Tool change

3 Tool change

3.1 T Tool preparation

T command

The T command is used to prepare an applicable tool from the specific tool group. Tool management decides which tool from the tool group is taken.

Examples:

```
T="55"
T="BOHRER"
```

For simplified programming, the tool identifier can be specified directly in case:
- the tool identifier is numerical
- the tool identifier is within the limits for integer variables 0 to 2147483647
Example: T55

T0

T0 empties the provisioning place if the spindle is empty.
T0 brings an empty cartridge to the provisioning place if there is a tool in the spindle

If further commands are programmed in an NC block after the T command, the T command is transmitted first and then the rest of the NC block.

Tools with fast tool break monitoring (SBBK)

Programming of the SBBK code

Before provisioning a tool with the desired SBBK, the GV-Variable `GV_HPBTD=true` must be set.

Inside the control, the SBBK code is then set in the tool data. The SBBK code is not written on the code chip when unloading.

Examples

Example of program start:

```
GV_HPBTD=true
T="55" M6
```

Machining example:

```
; ...
T="44" M6
GV_HPBTD=true
T="55"
; ...
; Machining with T="44"
; ...
T="55" M6
T="66"
; ...
; Machining with T="55"
; ...
T="66" M6 ;now T="55" is checked
```

3.2 M6/M46 - tool change

Effectiveness	If further commands are programmed in an NC block in addition to the change command, all other commands are executed first and the change command M6/M46 is executed last.
Synchronous Actions	Prior to carrying out the tool change all synchronization must be deleted (cancel). Tool change may not be selected via block-by-block operating synchronization.
GV variable	GV variables must be programmed prior to the tool change. Execution takes place in the active WKS directly after the tool change. GV variables only operate during the next tool change and are also cleared by the NC reset key.
T command	T command must be pre-programmed first (in the same NC block at the latest). Failure to do so will result in a tool change on the spindle tool, i.e. no tool change is executed.
Change speed	The tool change is carried out slowly if you program `M46` instead of `M6`. The tool change is also carried out slowly if the tool user date $TC_TPC1 Bit 0 = 1 is set for one of the two participating tools.
Change regarding the spindle tool	If the programmed tool is already located in the spindle, i.e. no tool change is required, the usual traversing movements are performed as follows: - Positioning of the Z-axis before tool change in the MKS with D0 at position GV_WZW_ZPOS is eliminated. - The target positions in the WKS after the tool change, GV_X to GV_B, are approached in one movement.
Limit speed	The "tool specific limit speed" takes effect after the tool change is carried out. The "tool specific limit speed" can be obtained from the manufacturer. You must enter it in the tool user date `$TC_TPC2`. If this tool user date is not defined then the machine limit speed takes effect.
CYCLE800	Following the tool change, the swivelling cycle CYCLE800 is deactivated.
Geometric switchover	After the tool change, the geometric axis switchovers are cancelled, setting the machine's basic configuration.

Residual current protection circuit

After tool change, the spindle is in position control if no rotation direction information has been programmed using GV_M[x] with x = 3, 4 or 5.

Tool break

The replaced tool is inspected for a tool break if the special code `$TC_TPC3` bit 3 is set in the tool user data.

Example M6

`G0 Z100 T15 D2 M6`

Provisioning on tool with identifier "15" is executed in parallel to the traverse movement in Z, the tool length of cutting edge 2 of the spindle tool being calculated in Z. Tool "15" is then loaded into the spindle to replace the current spindle tool.

Example:

```
T15
...
G54
GV_X=0  GV_Y=200  GV_Z=150  GV_S=1000  GV_M[0]=3
GV_D=2  GV_M[1]=121  GV_M[2]=21
M6
```

See section below for an explanation on GV variables.

3.2.1 GV-variables for M6/M46

GV_X	Target position X in WKS following tool change.
GV_Y	Target position Y in WKS following tool change.
GV_Z	Target position Z in WKS following tool change.
GV_A	Target position A in WKS following tool change.
GV_B	Target position B in WKS following tool change.
GV_S	Spindle speed following tool change.
GV_M[0-4]	Number of M functions following tool change.
GV_D	Tip selection for new tool.
GV_MEDIA	Coolant during WZW. Bit 0 = 1, M8 during WZW. Bit 1 = 1, M7 during WZW.
GV_WZW_ZPOS	Positioning Z-axis before WZW in the MKS with D0.
GV_SBBK_TOL	Tolerance for tool breakage monitoring (SBBK) Value range 0.1 to 10 degrees

Zero point

The active zero point offset is taken into consideration.

No rotation, reflection, and no scaling may be selected in the active zero point offset.

Waiting for fast tool break monitoring

Fast tool break monitoring causes the program to wait for measurement result after tool change. Only then the next NC block is executed. The waiting time could be used for programming of GV variables. By the use of GV_ variables, the next machining position can already be approached when fast tool break monitoring is running.

3.2.2 Tool change by means of the CS_TOOL cycle

CS_TOOL

This cycle might be used but only in exceptional cases.

CS_TOOL("name", edge, slow, bkz)

Input parameters:

name	Tool identifier
edge	Cutting edge Selection of the active cutting edge
slow	Change speed 1 = slowly, as M46
bkz	Additional drill head clamping 1 = with additional clamping for drill head

Example: `CS_TOOL("123",2,1,1)`
Tool 123 is slowly loaded, additional clamping for drill head is performed, and cutting edge 2 is selected.

Example: `CS_TOOL("123",,,1)`
Tool 123 is loaded as usual and addidional clamping for drill head is clamped.

If a tool with clamped additional clamping head clamping is located in the spindle, no considerations need be made for the replacement.

3.2.3 Customer defined adjustments of the tool change process

Two user defined cycles are provided for customizing adjustments in the tool change process.
- US_WZW_START.SPF
- US_WZW_END.SPF

The cycles are executed immediately **before** and **after** the process if they are included in the directory "user cycles".

3.3 Oversize tool

If an oversize tool is part of the tool change then an approach and departure strategy will be required that differs from the standard one. An oversize tool with an unaligned spindle must not be allowed to enter the collision area.

Replacing an oversize tool ...

The following must occur in the tool change positions X and Y prior to approach if an oversize tool is located in the spindle.

- The spindle must be oriented.
- The Z-axis must be in the tool change position or pulled back to `GV_WZW_ZPOS`.
- The traversing range restriction must be cancelled for oversize tools.

Exchanging an oversize tool ...

The tool speed cannot become operational until the spindle with the oversize tool is out of the collision area. Therefore a travel movement in X and Y is necessary after the tool is changed.

If you have programmed GV_X and GV_Y then this position will be approached. A position outside of the collision area will approached and an error message displayed if this position is located inside the collision area.

If you have not programmed a GV_X and GV_Y then this traversing movement will be generated.

i

The traversing range restriction is active after the tool change.

3.4 Tool compensation check

3.4.1 Tool compensation check on spindle tool

CS_TCHCK

CS_TCHCK(REAL LD_TL, REAL LD_TR, REAL LD_TOL, INT LD_D)

Input parameters:

LD_TL	Tool reference length
LD_TR	Tool reference radius
LD_TOL	Tool correction tolerance
LD_D	Tip to be checked

This cycle can only be used for milling cutters and drills, i.e. tool type 1 to 299 as well as turning tools with tool type 500 to 599. If a different tool type is used, the error message: 66075 "Tool type cannot be checked" is given.
With other tool types, the assignment of cutting data to tool length and tool radius is not clearly defined.

For details about the error message, see:
"66075 - Tool type cannot be checked, tool: %4" page 286

3.4.2 Tool compensation check for indicated tool

CS_TNOCHCK

CS_TNOCHCK(STRING[35] LD_TNAME, INT LD_DUPLO_NO, REAL LD_TL, REAL LD_TR, REAL LD_TOL, INT LD_D)

Input parameters:

LD_TNAME	Tool name
LD_DUPLO_NO	Duplo number of tool
LD_TL	Tool reference length
LD_TR	Tool reference radius
LD_TOL	Tool correction tolerance
LD_D	Tip to be checked

This cycle can only be used for milling cutters and drills, i.e. tool type 1 to 299 as well as turning tools with tool type 500 to 599. If a different tool type is used, the error message: 66075 "Tool type cannot be checked" is given.
With other tool types, the assignment of cutting data to tool length and tool radius is not clearly defined.

For details about the error message, see:
"66075 - Tool type cannot be checked, tool: %4" page 286

CHAPTER 4

Tool data and cutting data

4 Tool data and cutting data

4.1 Tool and cutting edge data

Address	Attributes	
Tool user data		
$TC_TPC1	Tool change speed 0: fast 1: slow	
$TC_TPC2	Max. speed	
$TC_TPC3	Special code: *1	Bit 3 = 1, decimal value 8 - SBBK Bit 5 = 1, decimal value 32 - facing slide attachment
$TC_TPC4	Envelope datum: Diameter HD	
$TC_TPC5	Envelope datum: Length HL (based on the spindle reference point)	
$TC_TPC6	Envelope datum: Height HH	
$TC_TPC7	Reference value SBBK	

Cutting edges user data	
$TC_DPC1	Tip-specific facing slide type

Monitoring user data	
$TC_MOPC1	IPM method tool break
$TC_MOPC2	IPM method overload

Magazine place user data	
$TC_MPPC1	Tool to magazine place
$TC_MPPC2	Cartridge to magazine place

*1 Add the decimal values for combinations of special codes.

4.2 Import and export of tool and cutting edge data

Application

The import and export of tool and cutting edge data allows the transfer of a tool and/or its data between 2 machines, even of the different model series H and/or MCi/MCH.
The tool identifier must be given.
The control type of the machine to which the tool is to be transferred can be stated (3rd call up parameter). If no specification is made, it is assumed that the tool is being transferred to an H/MC-machine
The specification of the duplo number is optional. If no specification is made, the tool is exported with the smallest duplo number.
During import, the tool is created with the old identifier and the old duplo number, as it was exported.
If there is already a tool with the same identifier and same duplo number on the target machine that is not loaded, these data are overwritten.

Procedure

H/MC machine is target machine

The cycle CS_TDTRANS writes a parts program with the name "Tool identifier_duplo number" into the NC_Data/parts_programs directory. This parts program is saved as a parts program on the target machine and executed in MDA. After execution of the parts program, the tool and all its data have been created.

Procedure

MCi/MCH machine is target machine

The cycle CS_TDTRANS writes a parts program with the name "Tool identifier_duplo number" into the NC_Data/parts_programs directory. The file ending ".mpf" must be changed to ".tld". In the target machine's tool catalogue, this file can then be imported directly from the data carrier (USB stick) via "import".

Data transfer

The tool and cutting edge data are all transferred with the exception of the following status bits of the tool date $TC_TP8, these are reset to 0:

Bit	Designation in the HMI screen	New value 0 = not set
0	Active tool (A)	0
5	Tool being changed (W)	0
7	Tool was in use (E)	0

8	Tool in the buffer storage	0
10	For unloading	0
11	For loading	0

CS_TDTRANS

CS_TDTRANS(LD_WZBEZ, LD_DPLNO, LD_STNG)

Input parameters:

LD_WZBEZ	Tool identifier (name)
LD_DPLNO	Duplo number
LD_STNG	Control type target machine 0 = H (SolutionLine) 1 = MCi/MCH (PowerLine)

Example

```
CS_TDTRANS('T05', 2, 1)
```

File created: T05_2_tld.mpf

Before import to MCi/MCH change to: T05_2_tld.tld

Chapter 5

Workpiece change, pallet change

5 Workpiece change, pallet change

5.1 Workpiece change, pallet change

Pallet management and pallet change

A pallet change may not be programmed in the NC program when working with pallet management.
The pallet management carries out the pallet change automatically.

Customer-specific adjustments

Two user defined cycles are provided for customizing adjustments in the approach to the pallet change position.

- US_PAW_START.SPF
- US_PAW_END.SPF

The cycles are executed **instead** of the standard approach if they are included in the directory "user cycles".

To program a tool change in the customised modifications, only CS_TOOL can be used.

CHAPTER 6

Cycles

6 Cycles

6.1 Traversing range restrictions

Application

Setting user specified traversing range restrictions in the machining axes X1, Y1, Z1, A, and B.
All input parameters are optional. The software limit switch goes into effect if no limits are defined.
The traversing range restriction always affects the machine coordinate system MKS even during active transformation.

CS_POSLIM

```
CS_POSLIM(REAL LD_MIN_X, REAL LD_MAX_X, REAL LD_MIN_Y, REAL
LD_MAX_Y, REAL LD_MIN_Z, REAL LD_MAX_Z, REAL LD_MIN_A, REAL
LD_MAX_A, REAL LD_MIN_B, REAL LD_MAX_B, )
```

Input parameters:

LD_MIN_X	Traversing range restriction X minus
LD_MAX_X	Traversing range restriction X plus
LD_MIN_Y	Traversing range restriction Y minus
LD_MAX_Y	Traversing range restriction Y plus
LD_MIN_Z	Traversing range restriction Z minus
LD_MAX_Z	Traversing range restriction Z plus
LD_MIN_A	Traversing range restriction A minus
LD_MAX_A	Traversing range restriction A plus
LD_MIN_B	Traversing range restriction B minus
LD_MAX_B	Traversing range restriction B plus

i

If additional limits have been defined by the HELLER system cycles then the strongest restriction goes into effect.
The user defined traversing range restrictions are no longer effective after NCK reset.
Software limit switch 2 goes into effect for programmed values located outside of the software limit switch 2.
Traversing range restrictions cannot be programmed for modulo axes.

HELLER

6.2 Precision Package (spindle growth compensation)

Application

Spindle growth compensation becomes effective when the motors are switched on and feed starts. It does not usually need any programming/operation. If spindle growth compensation has to be frozen or switched off for special machining purposes, the cycle CS_OTC can be used for this.

CS_OTC

CS_OTC(BOOL LD_MODE_ON_OFF, INT LD_MODE, BOOL LD_MODE_OFF)

Input parameters:

LD_MODE_ON_OFF	false (default) switch off true switch off
LD_MODE	0 (default) all compensation 1 only position-dependent (spindle growth) 2 only position-dependent
LD_MODE_OFF	false (default) only freeze compensation values reduce true compensation values

6.3 MDE/BDE data for an external MDE/BDE system

6.3.1 CS_BDEID – numerical value transfer (ID) to the DB13 of the PLC

Explanation

The cycle transfers a numerical value to the PLC via Dual-Port-RAM. In turn, the PLC provides the numerical value in the DB13 address DB13.DBD28 for external access.
The numerical transfer is secured by a semaphor in order to avoid accidental overwriting of values that have not yet been read.

CS_BDEID

CS_BDEID(ID, LOOP)

Input parameters:

ID	Value to be transferred Value range: -2147483648 ... 2130706432 (double word) If this parameter is missing, the DP-RAM is standardised, i.e. the semaphors are reset and the numerical value initialised with 0.
LOOP	Number of waiting loops with semaphor set, i.e. when the PLC has not read the value yet. 1 loop corresponds to around 1 second of waiting time. If this parameter is missing, there is no wait when a semaphor is set. If the semaphor is not cancelled within the necessary PLC loops, the following message is given: "Ident ID cannot be transferred - last value not read yet, ID=xxx".

Example

```
%_N_BEARBEITUNG_MPF
;$PATH=/_N_MPF_DIR
       CS_BDEID()          ;standardise DP-RAM, reset semaphor
       ...
       CS_BDEID(4711, 3)   ;transfer value
       ...
       M30
```

6.3.2 CS_BDECNT – set OK/NOK counter in the DB13 of the PLC

Explanation

The cycle sets or changes the OK and NOK counters included in the DB13 of the PLC.

OK counter:	DB13.DBD0
NOK counter:	DB13.DBD4

The cycle is usually used for standardising the OK and/or NOK counter in the DB13. This allows workpiece type-specific OK/NOK counters be used for an external BDE system, for example.
The numerical transfer is secured by a semaphor in order to avoid accidental overwriting of values that have not yet been read.

CS_BDECNT

CS_BDECNT(CNT_IO, CNT_NIO, LOOP)

Input parameters:

CNT_IO	Value to be transferred for the OK counter. Value range: -2147483648 ... 2130706432 (double word) If this parameter is missing, no value is transferred.
CNT_NIO	Value to be transferred for the NOK counter. Value range: -2147483648 ... 2130706432 (double word) If this parameter is missing, no value is transferred.
LOOP	Number of waiting loops with semaphor set, i.e. when the PLC has not read the value yet. 1 loop corresponds to around 1 second of waiting time. If this parameter is missing, there is no wait when a semaphor is set. If the semaphor is not cancelled within the necessary PLC loops, the following message is given: "(N)OK counter cannot be transferred - last value has not been read yet, (N)OK counter=xxx".

Example

```
%_N_BEARBEITUNG_MPF
;$PATH=/_N_MPF_DIR
       ...
       CS_BDECNT(0, 0, 3)  ;Zero the OK and NOK counters
       ...
       M30
```

CHAPTER 7

Tool monitoring WZU

7 Tool monitoring WZU

7.1 General, overview

Monitoring and responses

Various types of monitoring of the machining process can be carried out by options in the control system.

Types of monitoring:

SZU	SZU tool life or number of duty cycles monitoring
SBBK	Fast tool break monitoring SBBK
IPM, LUW	Spindle power IPM
IPM, VUW	Feed force IPM
AC	- Adaptive Control AC - Measuring probe

Reactions:

Stop EWS AWS

If one of these monitoring operations is activated, various reactions are possible. From a simple stop of the machining process or the use of DUPLOtools (spare tools, sister tools) right through to intelligent alternative strategies (AWS).

Combination of monitoring

Various monitoring processes can be combined. If required, measuring processes (measuring probes) can also be included to achieve optimum monitoring of the whole machining process.

Unmanned operation

For unmanned operation, the machining process must be equipped with suitable monitoring processes to protect tools and machine.

7.1.1 Monitoring processes, overview

Tool life/number of items monitoring SZU

Monitoring of **tool life** is effected by summating the usage time of a cutting edge during feed blocks (G1, G2 .....) and subtracting the result from the available tool life of the cutting edge at the end the program. No action is necessary in the NC program for this.

Monitoring of **quantity** is achieved by subtracting the number of workpieces produced from the number of workpieces available for the cutting edge at the end of the program. The function for subtracting the tool life must be activated in the NC program at the end of the program via the SETPIECE(n) command. (n) specifies the number of workpieces produced.

If the tool life or number of parts is less than zero, the tool is "disabled", i.e. it can no longer be used at the next tool change.

This type of monitoring is available according to the Siemens standard. The selection of the above monitoring takes place in the cutting edge data for the tool affected.

Fast tool break monitoring SBBK

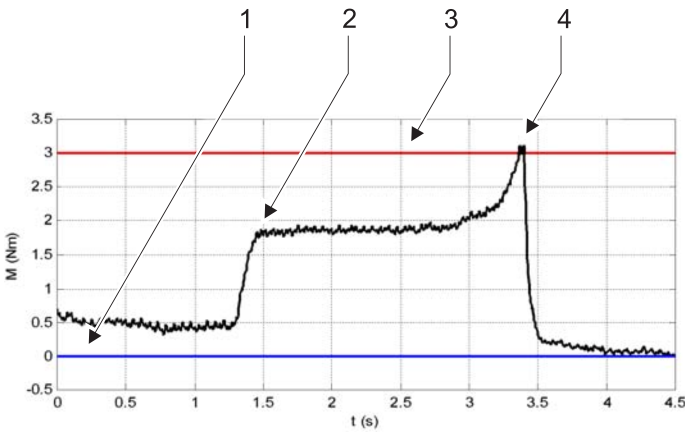
Tools are checked for breaks by a mechanical sensor element at the magazine provisioning place immediately after a tool change outside the machine's work area.
Selection takes place in the tool data.

IPM Integrated Process Monitoring

Tools are monitored for wear (overload) or breakage by the evaluation of the digital drive data of spindles and/or feed axes.

Monitoring by IPM is divided into the areas of LUW spindle drive power monitoring and VUW feed force monitoring on feed drives.

Display of power curve as diagram



7.1.1 - 1 Tool monitoring with alarm response

1 Lower limit (blue)
2 Signal (black)
3 Upper limit: (red)
4 Alarm response

Spindle power monitoring LUW with IPM

Cyclic scanning of the power data of the spindle and comparison with setpoint values allows tools to be monitored for wear and breakage.

Feed force monitoring VUW with IPM

Cyclic scanning of the power data of the feed axes and comparison with setpoint values allows tools to be monitored for wear and breakage.

Replacement tool strategy EWS

Several tools of the same type are used, so-called replacement tools (or DUPLOtools or sister tools) with the same tool identifier. Differentiation in the magazine is by the so-called DUPLOnumber.

If a tool attains the "disabled" status (e.g. due to expiry of the tool life), the next replacement tool with the next DUPLOnumber is loaded during the next tool change.

EWS automatically becomes active if an available replacement tool is found in the magazine on tool change instead of the disabled tool. If no replacement tool is found, the machine stops on tool change with a error message.

Alternative strategy AWS

If one of the above monitoring processes is triggered, a suitable response must be provided in the NC program.

An appropriate response must be conceived by the user for the specific application case.

The response to such an event can be in the following manner:

- M0 machine stop, message to operator, manual operator intervention, program abort
- Set pallet to reject, automatic pallet change, simple response in unmanned operation
- Omit subsequent operation, machine remainder
- Repeat interrupted machining
- Alternative machining, intelligent program sequence.

Dynamic AWS

The alternative strategy becomes effective immediately after the occurrence of a monitoring event at any point in the program (e.g. tool breakage with IPM).

Adaptive Control AC

Adaptive control makes it possible to control process data dynamically during the process.

The most common application of AC is for adaptive feed control, based on the spindle power values or feed force values.

In principle, AC takes place by way of 840Dsynchronous actions, which run cyclically in the background in an interpolation cycle (e.g. 12 ms) alongside the actual program.

Details see Programmer Manual SIEMENS Production Engineering chapter 10 "Synchronous movement actions"

7.2 Activating the monitoring process

Detailed application description

Chapter 7.2 describes only how the process is activated.

A detailed description of the application is provided from chapter 7.3.

7.2.1 Activating SZU, tool life/ number of duty cycles monitoring

SZU

SZU can be activated for each cutting edge, by marking the field "Tool life" and/or "Number of items" in the tool data. It is a precondition that a realistic tool life or number of parts is entered in the tool data.

In addition, the SETPIECE[n]command must be programmed at the end of the program for part quantity monitoring.

[n] specifies the number of workpieces produced. At the end of each program, this quantity is subtracted from the number of parts of the tool in use and being monitored.

7.2.2 Activate SBBK, tool break monitoring

SBBK is activated for each tool in the user-specific tool datum $TC_TPC3 by setting bit 3 (decimal value = 8). As a result, the tool is checked for breakage after each tool change.

Explanation see chapter:
"Tool and cutting edge data" page 64

`GV_SBBK_TOL` indicates the test tolerance in ° of the motor for SBBK.
The system provides 0.5°.

Explanation see chapter:
"GV-variables for M6/M46" page 55

Example

`T1 M6`	Load tool 1

`G1 X100 F1000 T2`	Machine with tool 1
`T2 M6`	Load tool 2 Replace tool 1 If $TC_TPC3 bit 3 of T1 is set, then T1 is checked for tool breakage.

7.2.3 Activating IPM, spindle power, feed force

Spindle power IPM (LUW)

Activation of IPM.

`N100 CS_IPMON`	All methods on
`N100 CS_IPMON("BREAK")`	Only break method on
`N100 CS_IPMON("OVERLOAD")`	Only overload on
`N999 CS_IPMOF`	All methods off
`N999 CS_IPMOF("BREAK")`	
`N999 CS_IPMOF("OVERLOAD")`	

An appropriate monitoring method must be assigned to each tool before activation.

The spindle power as monitoring task is assigned as the sensor. The sensor location is automatically the tool spindle (e.g. MA_C1).

On multi-spindle machines, all spindles in use (spindle 1 and 2) are each assigned to a monitoring method.

Each monitoring method can be defined with the "breakage" or "overload" event.

Details on settings of monitoring methods see Operator Manual.

Feed force IPM (VUW)

Activation is as for LUW.

The feed force and the planned sensor location (e.g. machine axis MA_X) must be assigned in the monitoring task as the sensor.

Position-dependent activation and deactivation of monitoring

In special cases, the monitoring activated using CS_IPMON can be temporarily deactivated and reactivated depending on the position during a synchronisation action with the variable $AC_MARKER[8]. With CS_IPMON, $AC_MARKER[8] is automatically set to "On" = 0.

$AC_MARKER[8]

Values of $AC_MARKER[8]

`Bit 0=1`	Deactivation of all monitoring events

`Bit 1=1`	Deactivation of break monitoring
`Bit 2=1`	Deactivation of overload monitoring (wear)
`bits all=0`	Activation of monitoring turned on using CS_IPMON

Example

`$AC_MARKER[8]=1`
Deactivation of IPM.

Monitoring only active between positions X100 and X200 .

```
WHEN $AA_IM[MA_X]>=100 DO $AC_MARKER[8]=0
```

```
WHEN $AA_IM[MA_X]>=200 DO $AC_MARKER[8]=1
```

Method selection through cutting edge change CS_D(1...9)

If another cutting edge D1...D9 (D12) with cutting-edge specific monitoring settings is to be selected during the machining process, this must be done using cycle CS_D . A correct cutting-edge related change of method is only possible using CS_D .

Example

`N100 CS_D(1)`	Number of cutting edges D1 with specific monitoring method
`N110`	Monitored machining, cutting edge D1 method
`N300 CS_D(2)`	Selection of cutting edge D2with change of method
`N310`	Monitored machining, cutting edge D2 method

EWS

The EWS automatically becomes active if several replacement tools are available in the magazine and the active spare tool is given the status "disabled".

7.2.4 Activating AWS, alternative strategy

Dynamic AWS CS_ESCON

Activation is done by CS_ESCON, specifying the selected monitoring method.

`N100 CS_ESCON(1)`	Activation after EWS
`N100 CS_ESCON(2)`	Activation after IPM_Break
`N100 CS_ESCON(4)`	Activation after SBBK
`N100 CS_ESCON(7)`	Activation after EWS=IPM=BBK

(ESCON value is bit-coded):

`bit 0`	EWS
`bit 1`	IPM
`bit 2`	SBBK

AWS jump destination alternative label CS_ESCLAB

The jump destination is specified after activation. This takes the form of a label (marker).

Example:

```
N100 CS_ESCLAB("LABEL_AWS1")
```

`CS_ESCON(7)`	Switch on monitoring
`CS_ESCLAB("LABEL_AWS1")`	Define alternative label
`T123 M6`	Insert tool into spindle
`N110 G1 X100`	with IPM monitoring
`N120 G1 Y300`	
`N999`	End of monitored machining
`N1000`	Sequential machining
`N8999 M30`	End of normal program sequence
`N9000`	AWS program section
`N9010 LABEL_AWS1:`	Continue with AWS here
`N9020 G0`	AWS actions
`N9025 ...`	AWS actions
`N9030 M30`	End of program AWS actions

7.2.5 Activating AC, Adaptive Control

Adaptive Control AC

To activate, either a local NC block or a modal synchro-action is programmed before the NC block(s) to be controlled adaptively.

For details. please refer to Siemens Production Engineering" PA, Chapter 10.

AC local NC block

Turn on ADAPTIVE CONTROL.

Feed control with override setting.

Example

```
WHENEVER $AA_LOAD[MA_C1] > 80 DO $AC_OVR = 80

WHENEVER $AA_LOAD[MA_C1] > 100 DO $AC_OVR = 0

WHENEVER $AA_LOAD[MA_C1] <= 80 DO $AC_OVR =100
```

`N1040 G1 X400`	AC only in N1040
`N1050 G1 Y500`	Without AC

Modal AC

Switch on ADAPTIVE CONTROL.

Feed control with override setting.

Spindle is axis MA_C1.

$AA_LOAD is drive utilisation in %.

Example

```
ID=1 WHENEVER $AA_LOAD[MA_C1] > 80 DO $AC_OVR = 80

ID=2 WHENEVER $AA_LOAD[MA_C1] > 100 DO $AC_OVR = 0

ID=3 WHENEVER $AA_LOAD[MA_C1] <= 80 DO $AC_OVR =100
```

`N1040 G1 X400`	
`N1050 G1 Y500`	
`N1060 G1 X-400`	All blocks with AC
`CANCEL (1,2,3)`	Delete modal synchronous actions 1,2,3

7.3 Tool life/no. of items monitoring SZU

End of tool life, end of number of items

If the tool life or number of items expires, the tool is marked as "disabled" (with "G"). The tool can finish the remainder of the machining, but cannot be fitted in the spindle again at a tool change.

If the remainder of the machining operation is not to be carried out at the end of tool life, the tool status $TC_TP8 must be evaluated in the program to interrupt the machining in advance.

Pre-warning tool life number of parts

To inform the operator in good time about the end of the tool life, a pre-warning limit can be specified. When the pre-warning limit is reached, a message is displayed on the screen that warns the operator of the approaching end of the tool life.

The tool is then marked with "V", pre-warning limit reached.

The pre-warning limit is entered in the tool data menu in the same way as the tool life and production quantity.

7.4 Tool break monitoring SBBK

7.4.1 Fast tool break monitoring SBBK

SBBK function outside work area

A check is made as part of the M6 / M46 tool change with a mechanical scanning element at the transfer location to the tool magazine in part in parallel with the approach movement to the next machining operation. If SBBK detects a tool breakage, the machine stops before the next feed block. Simultaneously AWS is started, if programmed.

7.5 IPM Integrated Process Monitoring

General information

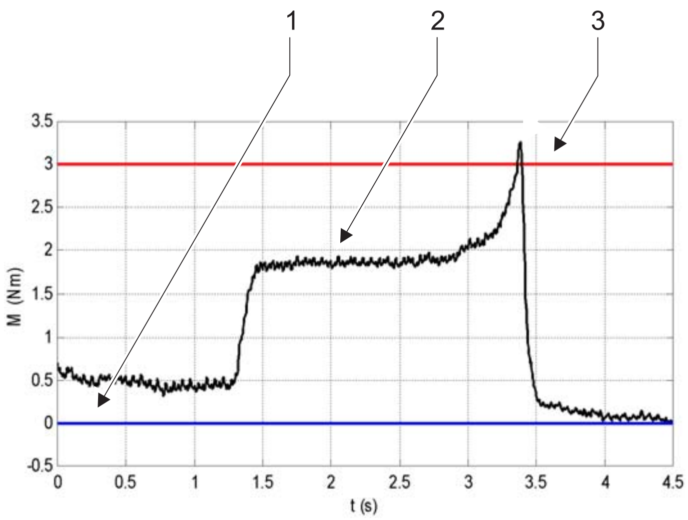
IPM cyclically (in interpolation cycle approx. 2 to 12 ms) evaluates the drive data from the spindle and feed drives and compares it to the limit values defined in a monitoring method.

Details on contents of monitoring methods see Operator Manual.

IPM compensates for any idling power and acceleration components. The performance data can optionally be displayed graphically on the screen in the form of a bar display with a maximum indicator.

Signal flow spindle power or feed force with IPM

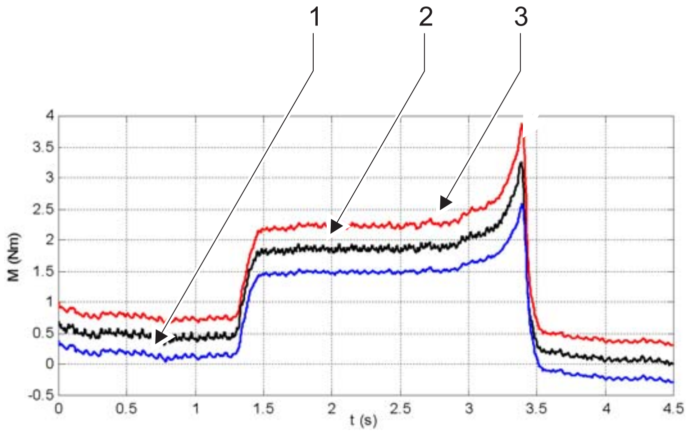
Procedure fixed limit



7.5 - 1 Drilling in aluminium with HSS

1 Lower limit (blue)
2 Signal (black)
3 Upper limit: (red)

Procedure floating threshold



7.5 - 2 Drilling in aluminium with HSS

1 Lower limit (blue)
2 Signal (black)
3 Upper limit: (red)

Wear limit or overload limit (OVERLOAD)

If the wear or overload limit is exceeded, the status of the tool in use is set to "disabled". The active machining process is continued with this tool. At the next tool change for this tool, the machine either stops with a "missing" tool or an available DUPLO tool (EWS) is loaded.

Break limit (BREAK)

If the break limit is exceeded, the process is stopped immediately. Without AWS, the machine is stopped with an error message. With AWS, the NC program is continued from the programmed alternative label CS_ESCLAB. The status of the tool in use is set to "disabled".

Adaptive override OVR

To achieve optimised-power machining time, OVR is able to increase the override by >100% on under-utilisation of the tool and reduce it by <100% on over-utilisation of the tool during machining.

The control parameters are defined in an overload method by the task "OVR control without alarm" or "OVR control with alarm".

Creating monitoring methods

The monitoring methods are normally created in interactive mode on control operating panel.

Exportnig and importing monitoring methods

Monitoring methods can, however, also be exported and imported from one machine to another (file type= *.mtd).

*. mtd files may not be altered externally as the files have a specific format.

7.5.1 IPM Power monitoring LUW (spindle drive)

Sensor location Spindle drive

Example

N10 ...SUPA Z900	Program start
N20 T123 M6	Tool change
N30 CS_IPMON	Break and overload on

Method active due to cutting edge selection D1 in M6 / M46

LUW spindle defined in method

Monitoring limits defined in method

Fixed value or floating threshold in method

N40 G0 Z0	Feed
N50 G1 X400	Mill
N60 G1 Y500	Mill
N70 G1 X-400	Sequential machining
....	
....	
....	
N99 CS_IPMOF	IPM off

7.5.2 IPM Feed force monitoring VUW

Sensor location Feed drive Z-axis

Example

N10 ...SUPA Z900	Program start
N20 T123 M6	Tool change
N30 CS_IPMON	Break and overload on

Method active due to cutting edge selection D1 in M6 / M46

VUW Z-axis defined in method

Monitoring limits defined in method

Fixed value or floating threshold in method

N40 G0 Z2	Feed

`N50 G1 Z-100`	Drilling
`N60 G0 Z200`	
`.......`	
`.......`	
`N999 CS_IPMOF`	IPM off

7.5.3 IPM Supplementary programmable functions

Normal case

Under normal circumstances, the NC program contains no IPM commands other than the CS_IPMON switch-on and the CS_IPMOF switch-off commands.

All the monitoring information is contained in the assigned monitoring methods.

Special cases

For more complex monitoring tasks, the IPM can be modified with further commands in the NC program.

IPM commands – expert knowledge

The following commands should only be used if the operator possesses sufficient expert knowledge of the IPM functionality.

The basic precondition is to carefully read the Operator Information on IPM.

Monitoring method groups

Method groups enable monitoring to be related to certain machining units, spindles or feed axes.

Method group 1

Only method group 1 is available for single-spindle machines.

Method group 2

Method group 2 is used for twin-spindle machines. Monitoring relates to the 2nd spindle 2 and the 2nd quill axis Z2.

Tasks

A maximum of 2 tasks can be defined in one monitoring method.

1. Monitoring, for example, spindle power
2. Monitoring, for example, feed force

Or alternatively:

1. Monitoring, for example, spindle power
2. Adaptively controlled feed override related to spindle power or feed force, with or without alarms

A maximum of 2 cycles can be active at the same time.

7.5.4 Switching IPM on and off

Switch on IMP

IPM is switched on with the CS_IPMON command.

`CS_IPMON`
`CS_IPM()`
`CS_IPM("")`
	Switch on all break and overload monitoring of the active method groups: With single-spindle machines, this is method group 1, with twin-spindle machines, this is dependent upon SOLO1, SOLO2 or TWIN mode.

`CS_IPMON ("BREAK")`	Switch on break monitoring for the active method groups.
`CS_IPMON ("OVERLOAD")`	Switch on overload monitoring for the active method groups.
`CS_IPMON("",2)`	Selection of method group 2. If the user does not specify, method group 1 automatically becomes active. Several method groups can be active at the same time.
`CS_IPMON("",1)` `CS_IPMON("",2)`	Select method groups 1 and 2 at the same time.

Switch-off IPM

IPM is switched off with the CS_IPMOF command.

`CS_IPMOF`
`CS_IPMOF()`
`CS_IPMOF("")`
	Switch off all monitoring of the active method groups.

`CS_IPMOF ("BREAK")`	Switch off break monitoring of the active method groups.
`CS_IPMOF("",2)`	Switch on all monitoring functions of method group 2.
`CS_IPMOF("",2)`	Switch on all monitoring functions of method group 2.
`CS_IPMOF("",1)` `CS_IPMOF("",2)`	Switch off method groups 1 and 2 at the same time.
`CS_IPMOF ("OVERLOAD")`	Switch off overload monitoring of the active method groups.

Path-dependent deactivation of IPM $AC_MARKER[8]

IPM can be deactivated by a synchronous action using the $AC_MARKER[8] variable. This enables path-dependent activation/ deactivation.

`Bit 0=1`	Deactivation of break and overload monitoring
`Bit 1=1`	Deactivation of break monitoring
`Bit 2=1`	Deactivation of overload monitoring (wear)
`Bit0...2 = 0`	Activation of monitoring

Example

- Milling path from X0 to X500
  - X100 to X200 deactivate monitoring
  - from X200 reactivate monitoring

```
G0 G54 X0

WHEN $AA_IW[MA_X]>100 DO $AC_MARKER[8]=1

WHEN $AA_IW[MA_X]>200 DO $AC_MARKER[8]=0

G1 G54 X500 F1000
```

7.5.5 Changing IPM method data

IPM method data

Method data of the IPM define the monitoring parameters for a tool or cutting edge. These can be generated or changed either at the Editor of the control in the IPM area or by NC voice commands. Methods are summarised in an ID code (>=1000) and with descriptive text.

See Operator Manual.

A method defined in the Method Editor contains:
- Method name
- Selection of break or overload monitoring events. This also links the alarm reaction: Overload: Disable tool (worn) break: Trigger machine stop M0 and/or
- Link (OR and/or AND) between tasks
- One or two monitoring jobs

A task defined in the Method Editor contains:
- Task name
- Active flag (for selecting/deselecting the task)
- Sensor (e.g. spindle power, feed force)
- Sensor location (e.g. MA_C1 or MA_Z)
- Monitoring method (e.g. fixed limit, moving threshold, Override control)
- Strategy parameters (e.g. upper limit)

Writing IPM monitoring method data CS_IPMMW("....")

Usually, monitoring method data is automatically activated during the tool change operation. These data can then be specifically adapated afterwards with language commands.

Strategy parameters STRATEGY_PARx

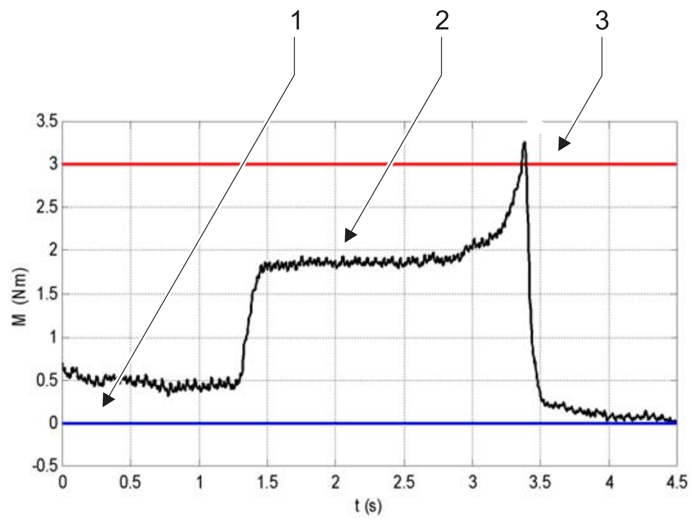
The monitoring method can be adapted using strategy parameters. These are defined as a string with the CS_IPMMW("…") command. The new parameter becomes effective immediately in the current monitoring operation.

“STRATEGY_PAR1 [1,1]“=12

Strategy parameter 1 in method 1, task 1:

With the "fixed limit" monitoring process:

Upper limit, spindle power in kW, feed force in kN



7.5.5 - 1 Drilling in aluminium with HSS

1 Lower limit (blue)
2 Signal (black)
3 Upper limit: (red)

“STRATEGY_PAR1 [1,1]“=34

Strategy parameter 1 in method 1, task 1, method group 1 (no entry):

	With the "moving threshold" monitoring procedure, this is the monitoring tolerance in % to the current threshold value.
("STRATEGY_PAR1 [1,1]“=34,2)	If the strategy parameter is to apply for method group 2 (MCT, for spindle 2), the parameter entry "..." for value 2 is separated by ",".
“STRATEGY_PAR2 [1,1]“=12	Strategy parameter 2 in method 1, task 1: With the "fixed limit" monitoring process: Lower limit, spindle power in kW, feed force in kN
“STRATEGY_PAR2 [1,1]“=2	Strategy parameter 2 in method 1, task 1, method group 1 (no entry): With "moving threshold" monitoring process: Switchover point absolute/percentage monitoring tolerance in kW (spindle power) or kN (feed force). Monitoring values below this threshold practically freeze the percentage monitoring tolerance is kind of frozen, because with values towards "0", the percentage tolerance would also tend toward "0" and hence render monitoring impossible.
("STRATEGY_PAR2 [1,1]“=34,2)	If the strategy parameter is to apply for method group 2 (MCT, spindle 2), the parameter entry "…" for value 2 is separated by ",".
“STRATEGY_PAR3 [1,1]“=1.3	Strategy parameter 3 in method 1, task 1: With the "fixed limit" monitoring process: Average value factor for sensor (no unit, multiple of the position control cycle). If a strong noise is present in the power signal, the factor must be increased to smooth the signal.
„STRATEGY_PAR3 [1,1]“=12	Strategy parameter 2 in method 1, task 1, method group 1 (no entry): With "moving threshold" monitoring process: Average value factor for threshold (no unit, multiple of the position control cycle). 1=performance values in position control cycle (2 ms) 5=usual value in method data (10 ms averaged) If a strong noise is present in the power signal, the factor must be increased to smooth the signal.
("STRATEGY_PAR3 [1,1]“=12,2)	If the strategy parameter is to apply for method group 2 (MCT, spindle 2), the parameter entry "…" for value 2 is separated by ",".

“STRATEGY_PAR4 [1,1]“=1.3	Strategy parameter 4 in method 1, task 1: Defined for the "fixed limit" monitoring process: "Tool missing" -limit in kW (spindle power), kN (feed force) Broken tools are thus detected if the power is underrun.
“STRATEGY_PAR4 [1,1]“=12	Strategy parameter 4 in method 1, task 1, method group 1 (no entry): With "moving threshold" monitoring process: Average value factor for sensor (no unit, multiple of the position control cycle). 1=performance values in position control cycle (2 ms) 5=usual value in method data (10 ms averaged). If a strong noise is present in the power signal, the factor must be increased to smooth the signal.
("STRATEGY_PAR4 [1,1]“=12,2)	If the strategy parameter is to apply for method group 2 (MCT, spindle 2), the parameter entry "…" for value 2 is separated by ",".
Further method data:	In extreme, exceptional cases, further parameters may be used. These parameters may only be modified by IPM experts. They are usually correctly preconfigured in the method templates supplied. Parameterisation upon request: - EVENT - JOB_MASK - JOB_CONNECT - ALARM_RESPONSE - SENSOR_ID - STRATEGY

7.5.6 Reading IPM method data

Reading current IPM monitoring values CS_IPMMR	All active method data (e.g. data as in CS_IPMMW, Chapter 15.5.4.1) can be read with the CS_IPMMR command.
CS_IPMMR (R0,"STRATEGY_PAR1[1,1]")	In the example, the current STRATEGY_PAR1 value from method 1, task 1 is stored in the variable R0.

CS_IPMMR (R0,"STRATEGY_PAR1[1,1]",2)	If values from method group 2 are to be read, the string is followed by: ".....",2)

Read further method data:

STRATEGY_ PAR2[1,1]	Read strategy parameters PAR2…PAR4 from method 1, task 1
STRATEGY_ PAR3[1,1]	
STRATEGY_ PAR4[1,1]	
CS_IPMVR(R0,"UPPER_TOWVAL")	Read upper maximum indicator value
CS_IPMVR(R0,"LOWER_TOWVAL"	Read lower maximum indicator value
CS_IPMVR(R0,"BLOCK_AVERAGEVAL")	Read block average value (filtered performance value over the elapsed time of the monitoring task).
CS_IPMVR(R0,"BLOCK_AVERAGEVAL")	Read block average value (filtered performance value over the elapsed time of the monitoring task)
CS_IPMVR(R0,"EVENT_STATE")	Read code monitoring status.
CS_IPMVR(R0,"DEBUGVAL[n]")	Read DEGUBvariables, for service purposes only.

7.5.7 IPM method selection and method definition

Automatic method selection using M6 / M46	The monitoring method assigned to a tool is automatically activated with the cutting edge D1…D9 activated in M6 / M46.
CS_D(1)	If another cutting edge Dx with a different monitoring method is required, use the CS_D(1)…CS_D(9) command. The monitoring method is not activated with the normal D1…D9 selection.
Direct method selection CS_IPMMS	The monitoring method can also be selected again directly via the CS_IPMMS command. However, this requires the methods to be loaded! The method ID should always be > 1000.
CS_IPMMS(1035)	Method 1035 of method group 1 (default) is activated.
CS_IPMMS(1035,1036,2)	Method 1035 of method group 1 and method 1036 of method group 2 are activated.

CS_IPMMS(1035,1036,2, "TOOL_23")	In addition, the identifier of the tool involved in the monitoring process (in the example TOOL_23) can be indicated. This is output with IPM monitoring messages.
Direct method definition CS_IPMDT (1,2,"STRING1",3,4, "STRING2", "STRING3", 5,6,7,8)	A method is defined with 8 INT variables and 3 STRING variables. Usually, however, methods are generated in the method editor from method templates on the screen.
CS_IPMDT(1,…..)	Array index 0…99 (INT) under which the method is stored in GUDs (SGUD.DEF), in this case=1.
CS_IPMDT(,2,…..)	Method key 1…2147483647 (INT), in this case =2
CS_IPMDT (,,"STRING1",…)	"BREAK" or "OVERLOAD" method event.
CS_IPMDT(,,,2…..)	Task number 1 or 2 corresponds to monitoring task 1 or 2
CS_IPMDT(,,,,3,…)	Machine axis number (INT) 1…31, MA_X=1, MA_Y=2 MA_Z=3
CS_IPMDT (,,,,,"STRING2",…..)	Possible entries for sensor signals are: "CURRENT" "TORQUE" "POWER" "FORCE" "RADIALFORCE" "AXIALFORCE"
CS_IPMDT (,,,,,,"STRING3"…..)	Possible entries for monitoring strategies are: "ACTUALVALUE_FIXLIMIT" "AVERAGEVALUE_FIXLIMIT" "ACTUALVALUE_MOVINGTHRESHOLD" "AVERAGEVALUE_FIXLIMIT_WEARMISSING" "AVERAGEVALUE_MOVINGTHRESHOLD" "FEED_REGULATION" "FEED_REGULATION_ALARM"

CS_IPMDT(,,,,,,,11,..)	Strategy parameters STRATEGY_PAR1
CS_IPMDT(,,,,,,,,22,..)	Strategy parameters STRATEGY_PAR2
CS_IPMDT(,,,,,,,,,33,.)	Strategy parameters STRATEGY_PAR3
CS_IPMDT(,,,,,,,,,,44)	Strategy parameters STRATEGY_PAR4

7.5.8 Executing IPM commands

Executing command CS_IPMCO ("STRING", 123,2)

Individual IPM commands can be executed using CS_IPMCO. Certain "STRING" commands may be accompanied by a value, e.g. (,123) and the command for method group 2 can be specified by entering a further value (,,2).

The following commands can be defined under "STRING":
Single commands: `CS_IPMCO("STRING")`

Example: `CS_IPMCO("CLEARTOWVAL")`

CLEARTOWVAL	Clear all maximum indicator values.
CLEARTOWVAL_ BREAK	Clear maximum indicator values to break.
CLEARTOWVAL_ OVERLOAD	Clear maximum indicator values to overload.
NEXT_NOT_ EXECUTABLE	The next IPM command is not output until the next executable NC block (e.g. G1 X200).
NEXT_ EXECUTABLE	The next IPM command is output immediately.
CLEAR_ALARM_ BREAK	All break alarms are cleared.
CLEAR_ALARM_ OVERLOAD	All overload alarms are cleared.
CLEAR_ALARM	All alarms are deleted.
LOWLIMIT_ DELAYTIME",3)	Set cutting check to 3 seconds, for example. The lower limit is monitored with a 3 second delay.
LOWLIMIT_ DELAYTIME_ REPEAT",3	Set repeatable cutting check to 3 seconds, for example. Cutting monitoring is active in all the following NC blocks with changeover from G0 to G1,G2… .
LOWLIMIT_ DELAYDIST",3)	Set cutting check to 3 mm path distance, for example. The lower limit is monitored with a 3 mm path distance delay.

LOWLIMIT_ DELAYDIST_ REPEAT",4)	Set repeatable cutting monitoring to 4 mm for example. Cutting monitoring is active in all the following NC blocks with changeover from G0 to G1,G2….
ALARM_OFF	Suppress alarm output.
ALARM_ON	Activate alarm output.
SIGNAL_FACTOR[n]“,1.25)	The signal of a machine axis to be monitored (MA_X: n=1, MA_Y: n=2 usw.) is evaluated with a factor, in the example 1.25.
RESET_ METHODGROUPS	All methods of all method groups become ineffective.
CONFIGURE_ METHODGROUPS	All methods and method groups are configured.
CHECK_WEAR	The current power value is compared at overload (tool=blunt). If it is overrun, an alarm is output.
CHECK_MISSING	The current power value is compared at overload (tool=missing, no power draw). If it is underrun, an alarm is output.
CHECK_WEAR_ AND_MISSING	The current performance value is compared at overload and when the power is too low (see above).
CLEAR_BLOCK_ AVERAGEVAL	The current block average value (averaged performance value, moving threshold) is reset.
UPPER_OVERRIDE_CHECK",120)	Overload (blunt) monitoring is switched off from an override position of 120%, for example. Feasible values: 100%….200%
LOWER_OVERRIDE_CHECK",70)	Cut and missing check monitoring monitoring is switched off from an override position of 70%, for example. Feasible values: 0%...99%
SET_EVENT_STATE",’b100’	Switch on/ off individual monitoring processes. - `Bit 0=1`: All monitoring processes are active - `Bit 1=1`: only break monitoring processes active - `Bit 2=1`: only overload monitoring active - `All bits=0`:
FEEDRATE_AVERAGE_FACTOR",10)	The current power value for the "controlled override OVR" process is filtered. The factor represents a multiple of the position control cycle (approx. 2 ms).

FEEDRATE_AIR_ CUT_LIMIT",2)	With the "controlled override" process, the override is increased to > 100 % if the power is underrun (air cut, empty cut). The limit in kW or kN (e.g. 2 kW) indicates the power level from which an air cut is detected.
FEED_RATE_AIR_CUT_FEED_FACTOR", 130)	When an air cut is detected, this factor becomes effective in % of the programmed feed (e.g. 130%). The default value is 150%. Feasible values: 100%…250%
FEEDRATE_ GAIN",1.2)	The feed control with the "controlled override OVR" process is designed as a PI control. This factor can be used to determine the P-component of the control (in the e.g. 1.2). The default value of FEEDRATE_GAIN is = 0, hence only the I part of the controller works with the integration time (see below FEEDRATE_TIME). Feasible values: 0…2, optimum value 1.2
FEEDRATE_TIME",5)	This is the time constant of the I controller in seconds (e.g. 5 s) for the feed determination in the "controlled OVR" process. The default value is 4 s, i.e. the override is regulated within 4 s to the higher OVR value in the case of air cutting or to the lower value in the case of a power increase.

7.5.9 Logging IPM values to file

Writing current IPM values to file CS_IPMWL ("LOGFILE")

CS_IPMWL writes IPM values, usually maximum tow values to a File.

This is useful, for example, if the machining process is being run in for the first time.

"STRING2"

Name of logfile.

If no name is specified, the file IPMLOG_CHAN1_1.MPF is automatically written in the current *.WPD.

When this file is full, the next one is created up to a max. of IPMLOG_CHAN1_9.MPF. When this is also full, the next file will be deleted beforehand.

"STRING3"

Text as headline in the log file created with STRING2.

7.5.10 Setting IPM values for special cases

Defining the signal source for IPM CS_IPMSS(1,2,3,4)

CS_IPMSS defines the signal source as an axis number from which the performance values are read.

(1,….)	Axis table `$MC_IPM_METHODGROUP_AXTAB[0..3]`
(,2,…)	Machine axis number 1...31, sensor location MA_X=1, MA_Y=2, MA_Z=3, MA_C1=
(,,3,..)	Machine axis number 1…31 sensor location adjustment
(,,,2)	Method group number 1 or 2

Specifying spindle tool orientation CS_IPMTO(1,2,3)

CS_IPMTO specifies the spindle tool orientation in vector components of the GEOAX direction and is used to calculate axial and radial force components.

The spindle orientation is usually set automatically via the plane selection. For special cases, however, this can be modified with CS_IPMTO.

G17: Z direction

G18: Y direction

G19: X direction

ExamplesCS_IPMTO:

Normal case: Tool axis in MA_Z direction: `CS_IPMTO(0,0,1)`

WZ axis in X direction, e.g. angular head: `CS_IPMTO(1,0,0)`

WZ axis spatially inclined:

```
CS_IPMTO(0.2345,0.4567,0.85815);XYZ component
```

Setting IPM commands CS_IPMWL ("COMMAND", "STRING2")

Individual IPM commands can be executed using `CS_IPMWL("COMMAND","STRING2")`.

The following commands can be indicated under "STRING2":

CLEARLOGFILES	Clear all logfiles
START_LEARNING_MONITORING	Start teach-in/monitoring run
ALL_ON	Switch on all logging processes

ALL_OFF	Switch off all logging processes
LOWERTOW_ON	Lower maximum indicator value on
LOWERTOW_OFF	Lower maximum indicator value off
UPPERTOW_ON	Upper maximum value indicator on
UPPERTOW_OFF	Upper maximum value indicator off
BLOCK_AVERAGE_ON	Block mean value ON
BLOCK_AVERAGE_OFF	Block average value OFF
TEXTFORMAT_ON	Text format ON, output in ASCII format, spaces=separators Output EXCEL format (*.CSV) "," = separator
TEXTFORMAT_OFF	Text format OFF, output in ASCII format, spaces=separators Output EXCEL format (*.CSV) "," = separator
HEADERLOGFILELINE_ON	Show header line
HEADERLOGFILELINE_OFF	Hide header line
HEADERLOGFILELINE_ON	Show header line
HEADERLOGFILELINE_OFF	Hide header line
LASTLOGFILELINE_ON	Show final line
LASTLOGFILELINE_OFF	Hide final line

Set machining section command

```
CS_IPMWL ("COMMAND", "SETMONITORINGCOUNTER","3")
```

The machining sections of a tool are fitted with a counter (MONITORING_COUNTER). This counter can be indicated with the 3rd STRING variable. This is only required if machining sections are skipped within a machining program (e.g. resume).

7.5.11 IPM mode, teach-in, monitoring run

Teach-in "LEARNING" Monitoring run "MONITORING"

Before a machining process can be monitored, the nominal power values must be determined with new and/or sharp tool. This can be done either by reading the maximum indicator values from the "IPM basic screen" HMI operating panel and entering them into the method editor or through a learning run, which involves the nominal power values being automatically determined and entered into the method data.

For the time being, a learning run is not possible with the "moving threshold" process.

Activating a teach-in run

This command activates a teach-in run whereby the learned data are automatically written to a file IPMLEARN_CHAN1.MPF in the active *.WPD.

Start of teach-in run:

```
CS_IPMWL ("MONITORING", "LEARNING")
```

End of teach-in run:

```
CS_IPMWL ("LEARNING", "LEARNING")
```

All recorded values are written to a so-called "learn file" IPMLEARN_CHAN1.MPF ,not however to the assigned monitoring method.

Activating subsequent monitoring run

Toggling the 2nd string when entering CS_IPMWL from LEARNING to MONITORING, activates the monitoring run with the data previously recorded during the teach-in run. Preferably this should be done by changing the assignment of a STRING variable in the program sequence.

Start of monitoring run:

```
CS_IPMWL ("MONITORING", "MONITORING ")
```

End of monitoring run:

```
CS_IPMWL ("LEARNING ", "MONITORING ")
```

Programming example

Complete programming example for a "simple" teach-in run.
The following commands are required to complete a teach-in run.

DEF STRING[12]	LV_MODE	
N100 LV_MODE	="LEARNING"	For teach-in run
N110 ;LV_MODE	="MONITORING"	For monitoring run, after 1st run N100 ;LV_MODE= comment out, activate N110 LV_MODE
CS_IPMWL	("COMMAND","ALL_ON")	Logging function all on.
CS_IPMWL	("COMMAND","START_ LEARNING_MONITORING")	Start command for teach-in run
T123 M6		Load tool
CS_IPMWL	("MONITORING","LV_MODE")	Monitoring with very wide boundaries, no alarms
G1 X123 X234		Operational sequence

CS_IPMWL	("LEARNING","LV_MODE")	Read learned values, max. indicator value
T234 M6		Load next tool.

7.6 Replacement tool strategy EWS

Function of EWS

EWS allows worn ("disabled") tools to be automatically replaced for the next use by a tool that is normally of the same type.

Several tools of similar type have the same tool name but have different replacement tool numbers (DUPLO numbers in Siemens terminology).

EWS activates the tools in order of ascending DUPLOnumber.

EWS is carried out automatically by the tool management and during tool change M6 / M46.

No replacement tool present

If no replacement tool can be found for the tool change, without AWS or with AWS deselected, the machine stops with an error message.

If AWS is selected, the alternative strategy is started.

7.7 AWS alternative strategy (pallet change strategy)

Purpose of AWS

The part program sequence normally runs to the end without a fault.

There can, however, be faults in the machining process due to wear, material fault, etc.

If an operator is present, such faults can be detected audibly or visually. Undetectable faults or faults when the operator is not present, e.g. during unmanned shifts, must be detected automatically using suitable sensors (e.g. SBBK).

AWS enables a planned response to fault events of this kind.

This means that the machine does not simply remain stopped (unproductive), (e.g. following SBBK in unmanned shift), but continues to operate in a suitably programmable manner (e.g. change pallet, machine next pallet).

Function of dynamic AWS

An AWS label is programmed before every program section that is intended for monitoring with IPM, SBBK or other methods. If, for example, the IPM breakage limit is triggered at any point in the machining process, the NC program is stopped immediately, aborted, a RESET is carried out and an automatic skip to the AWS label (marker) is executed. The NC program is automatically continued from that point.

The programmer must ensure that it is possible to continue without collisions in all possible abort situations in the AWS program branch.

Example see chapter:
"Activating AWS, alternative strategy" **page 86**

CHAPTER 8

HELLER Technology Cycles

8 HELLER Technology Cycles

8.1 Overview

8.1.1 General information

Scope

This collection of sub-programs includes all the machining and measurement cycles for SINUMERIK 840D sl that are supplied with the machine as options on suitable data media. They are normally also stored in the controller memory (under "Manufacturer cycles" in the "Programs" main menu). The sub-programs are written "axis-independent" and can be used in both the XY plane G17 and the ZX plane G18. From Version V7.1 onwards, the sub-programs are supported by graphics in the program editor of the 840D.

Graphics in this description

The sequence of the sub-programs and necessary geometric parameters are illustrated in diagrams.

The input parameters and their significance are listed below the diagram. With complex sub-programs, a sequence description is added if necessary. A programming example is also given.

Note

The term "Sub-program" is also abbreviated to "UP" in the following descriptions.

8.1.2 Classifying the sub-programs in groups

Group division

The sub-programs are divided into the following main groups, with the L numbering also corresponding to the main groups.

1. Sub-programs for machining
   L0xxx Drilling sub-programs
   L1xxx Milling sub-programs
2. L2xxx Geometry sub-programs
3. L3xxx Measurement sub-programs
4. L4xxx Machine handling sub-programs

5. L5xxx Calculation sub-programs
6. L6xxx User-specific sub-programs

8.1.2.1 Determining L numbering

Allocation to groups

Specific L numbers are defined to permit better allocation of the subprograms to the sub-program groups.

8.1.3 Dividing the input parameters into groups

Input parameters

The sub-programs can be supplied either by R parameters R0…R89 or by transfer parameters.

As R parameters

The sub-program call up must not contain (…) when R parameters are supplied.

Example of R parameters

```
N10 R0=0 R1=-20 R5=10 R6=2
N20 L61
```

Transfer parameter

With transfer parameters, the sequence in the brackets (1,2,3,…) decides on how the numerical value is assigned as an input parameter. Therefore, either all transfer parameters must always be stated or a space entered separated by …,_,… commas.

Individual parameters that are not entered are accepted with the value "0".

If several sub-programs are called up in sequence with the same parameterisation, only the first call up needs to contain all transfer parameters. The following call ups without brackets automatically transfer the values from the first call up (corresponding R parameters are automatically loaded).

The transfer parameter sequence corresponds to the sequence of the described R parameters shown in the graphics below.

Example of transfer parameters

```
N10 L61(0,20,10,2)
```

R parameters in groups

For reasons of clarity, the input parameters as R parameters are divided into the following groups. It also makes sense to use these groups when creating your own sub-programs.

R group	Group designation
R0.....R9	Details of drilling axis
R10...R29	Details of machining plane axes
R30...R39	Details of the axis vertical to the plane
R40...R59	General dimension details
R60...R69	Sequence influences/settings
R70...R79	Technological details
R80...R89	Measuring result parameters
R90...R99	Freely available for user
R100...R399	Internal calculation parameters for the user
R500...R599	Reserved for HELLER technology applications.
R600...R699	Freely available for the user
R700...R799	Freely available for the user
R800...R899	Freely available for the user

8.1.3.1 Overview of parameter assignment

Significance

The supply of individual sub-programs takes place maintaining the above group allocation with R parameters or transfer parameters. The significance of these can be found in the corresponding description of the sub-programs in these programming instructions.

Local parameters are used for internal calculations.

8.1.4 Handling the sub-programs

General

All the machining sub-programs developed by HELLER are described in these programming instructions. The assignment of the parameters is explained for every sub-program and is shown with the help of an application example. The significance of the individual parameters is also shown graphically.

First of all, if necessary, a plan view of the machining plane is shown with the parameters for this plane. A side view is then shown with the remainder of the parameters. The side view normally shows the situation with reference to the drilling axis.

With simple sub-programs, only one view is shown; with complex sub-prorgams there can be several views, for example to show the details more clearly.

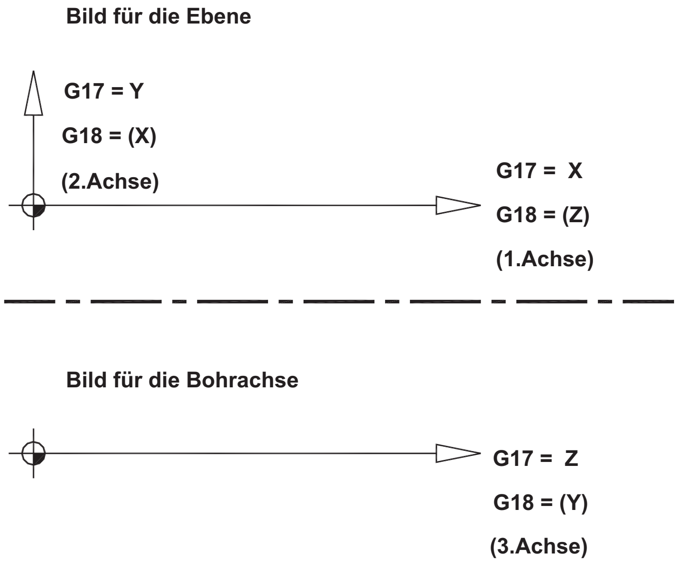
Input parameters and graphical illustrations

The appropriate graphic is illustrated for each sub-program and the associated input parameters are listed under each diagram.

The following conventions apply to all diagrams that show different machining planes:

Coordinate details X, Y and Z usually applies to machines with fixed work spindle.

Details (X), (Y) and (Z) apply to the G18-(ZX) plane for machines with swivelling work spindle or for angled heads, if fitted.



8.1.4 - 1 Figure for the plane and the boring axis

8.1.5 Using the sub-programs

Sub-program call up

The sub-programs are called up directly from the parts program or from MDA mode.

For the reasons shown in the diagram printed in this sub-program and the listing of the input parameters as value assignments, the input parameters need to be written before the sub-program call up.

Executing the programs

Execution of the sub-program takes place in travel mode either in automatic sequence by calling up from the part program or by calling in MDA mode.

In automatic operation, executable NC blocks are shown on the screen and the current block is specifically marked. The Single block, hide block, etc. functions are generally available.

Machines with swivel head

The sub-programs are written "axis-independent" and can be used in both the XY plane and the ZX plane.

Tool data

The precondition for using the standard cycles is correct tool data input.

The current tool radius and not only wearing must be entered into most programs.

The tool length is calculated in the drilling axis.

For drilling cycles, the drill radius is to be entered in $TC_DP6 and the point angle in $TC_DP24.

HELLER

8.2 Sub-programs for machining

8.2.1 Drilling sub-programs

Standard drilling cycles

The tool tip is calculated in the sub-program using the data. It is thus possible to program the effective drilling depth.

Tool data pre-requisites

To be able to calculate the tool tips of the drills, tool radius and point angle must be specified in the tool data.

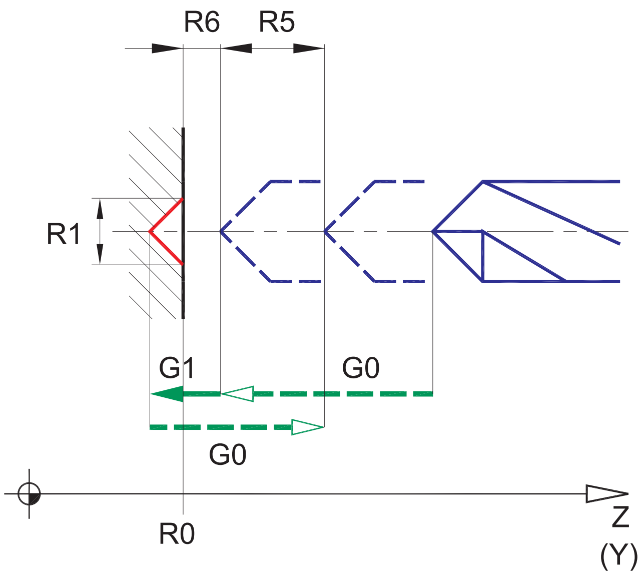
WZ-Typ drilling or milling tools

Tool radius: $TC_DP6 [mm]

Point angle: $TC_DP24 [0.500° to 180.000°]

8.2.1.1 L0xxx Drilling sub-programs

L60 F and S matched centring



8.2.1 - 1 Cycle diagram L60

Input parameters

R0 Surface area

R1 Centring diameter (Point angle from tool cutting user data)
R5 Retraction
R6 Safety distance

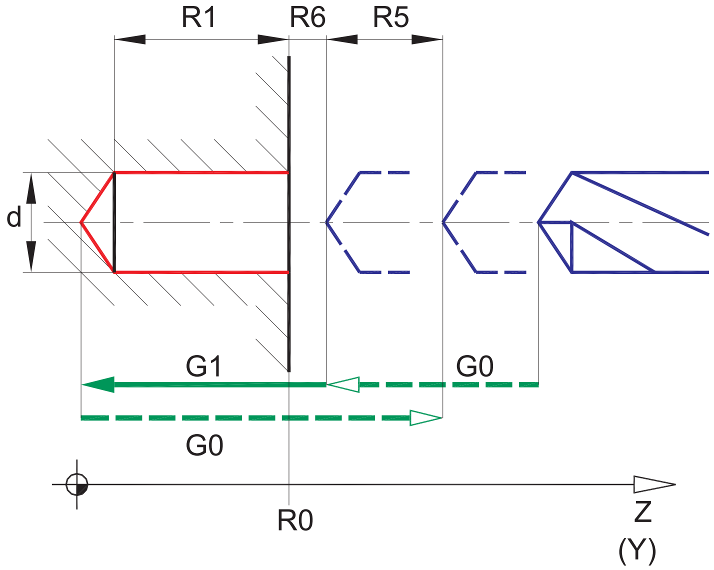
Application example

```
N100 X100 Y100
N110 R0=0 R1=10 R5=50 R6=2
N120 L60
```

i

A change of gear when matching the speed can be prevented by programming M42. The maximum permissible speed (machine, tool) is observed.

L61 Drilling with effective drilling depth



8.2.1 - 2 Cycle diagram L61

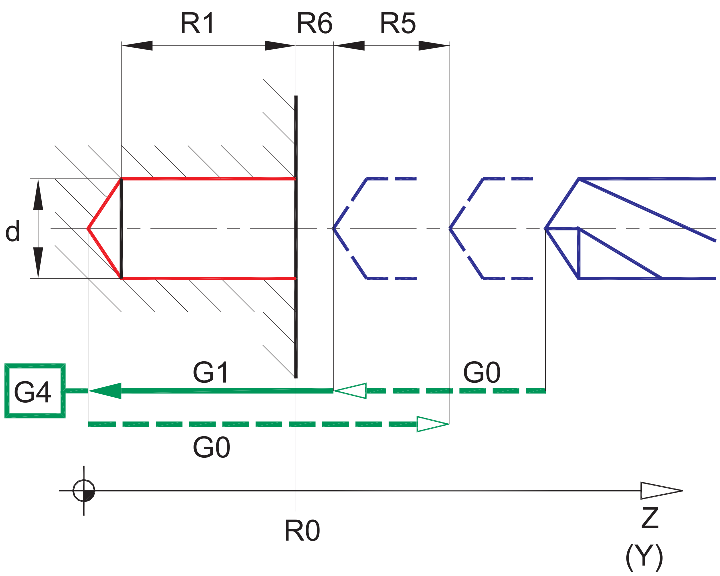
Input parameters

R0 Surface area
R1 Effective drilling depth (point angle from cutting edge user data)
R5 Retraction
R6 Safety distance

Application example

```
N100 X100 Y100
N110 R0=0 R1=20 R5=200 R6=1
N120 L61
N130 X200
N140 L61
```

L62 Drilling with chip clearance



8.2.1 - 3 Cycle diagram L62

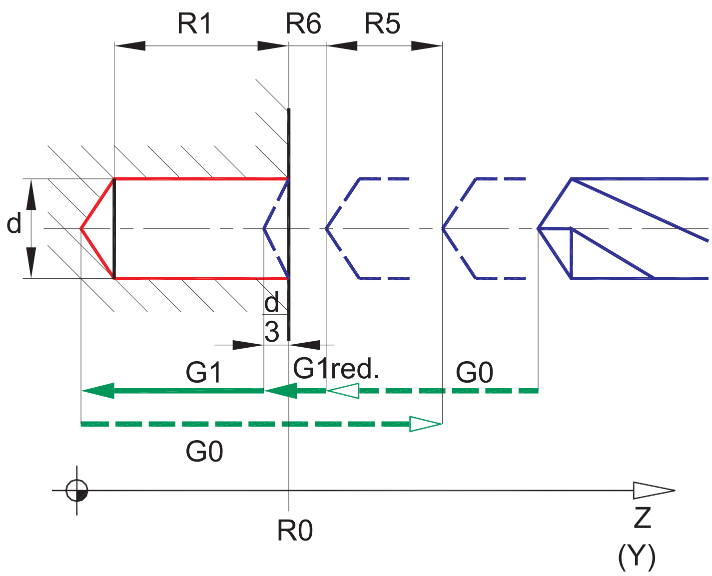
Input parameters

R0 Surface area
R1 Effective drilling (Point angle from tool cutting user data)
R2 Dwell time in seconds [s]
R5 Retraction
R6 Safety distance

Application example

```
N100 X100 Y100
N110 R0=100 R1=20 R2=0.2 R5=200 R6=1
N120 L62
N130 X200
N140 L62
```

L63 Drilling with tapping



8.2.1 - 4 Cycle diagram L63

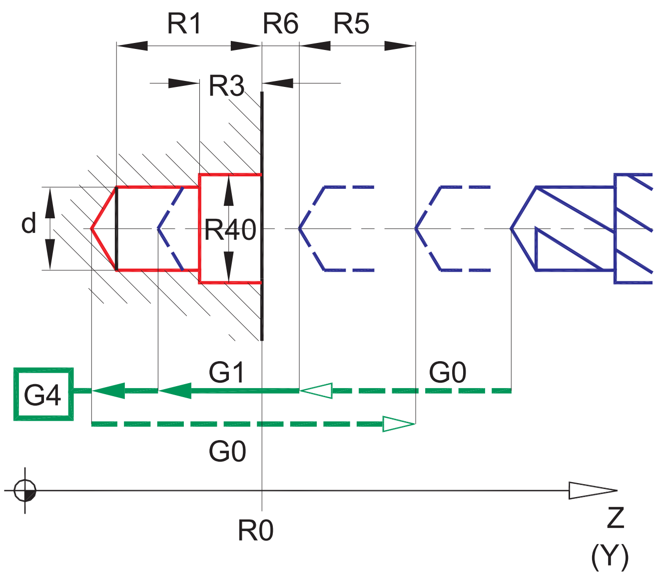
Input parameters

R0 Surface area
R1 Effective drilling (Point angle from tool cutting user data)
R5 Retraction
R6 Safety distance
R70 Feed rate reduction in % when tapping

Application example

```
N100 X100 Y100
N110 R0=100 R1=20 R5=200 R6=1 R70=50
N120 L63
N130 X200
N140 L63
```

L64 Drilling with counterboring and chip clearance



8.2.1 - 5 Cycle diagram L64

Input parameters

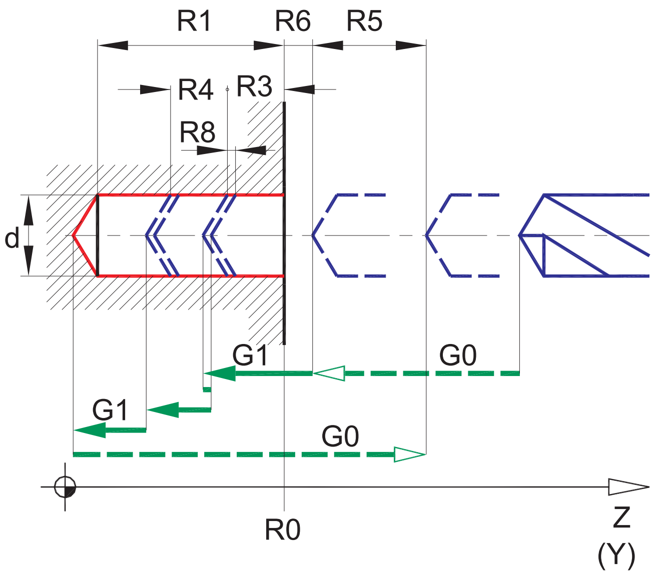
R0 Surface area
R1 Effective drilling (Point angle from tool cutting user data)
R2 Dwell time in seconds [s]
R3 Counterboring depth or chamfer
R5 Retraction
R6 Safety distance
R40 Counterboring diameter

Application example

```
N100 X100 Y100 M42
N110 R0=100 R1=40 R2=0.2 R3=15 R5=200 R6=1 R40=25
N120 L64
N130 X200
N140 L64
```

i Feed and speed rate are reduced by a ratio of d/R40 according to the counterboring. Gear changing on machines with automatic gear changing can be prevented by programming the appropriate M function (e.g. "M42").

L65 Drilling with chip breaking



8.2.1 - 6 Cycle diagram L65

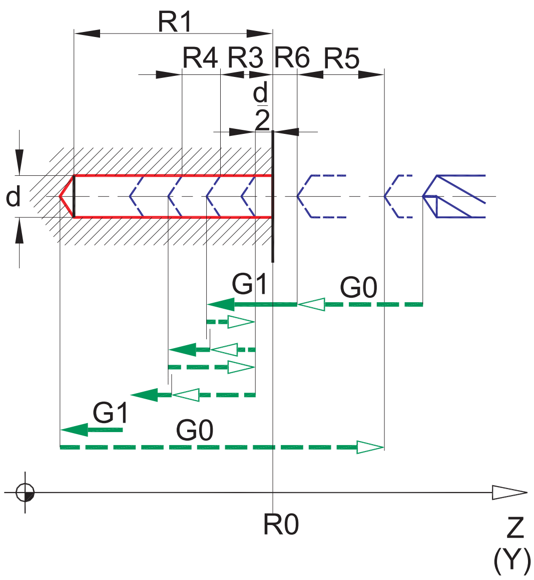
Input parameters

R0 Surface area
R1 Effective drilling (Point angle from tool cutting user data)
R3 1. Drilling depth
R4 Depth growth
R5 Retraction
R6 Safety distance
R8 Chip breaker stroke

Application example

```
N100 X100 Y100
N110 R0=100 R1=60 R3=20 R4=5 R5=200 R6=1 R8=1
N120 L65
N130 X200
N140 L65
```

L66 Deep hole drilling with reduction



8.2.1 - 7 Cycle diagram L66

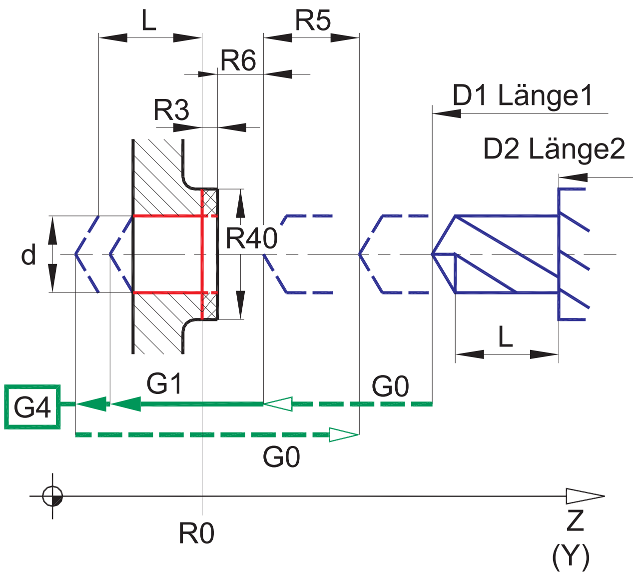
Input parameters

R0 Surface area
R1 Effective drilling (Point angle from tool cutting user data)
R3 1. Drilling depth
R4 Depth growth
R5 Retraction
R6 Safety distance
R70 Feed rate reduction in % (for depth growth in each case)
R71 Depth growth reduction in %

Application example

```
N100 X100 Y100
N110 R0=100 R1=60 R3=20 R4=5 R5=200 R6=1 R70=10 R71=20
N120 L66
N130 X200
N140 R0=120
N150 L66
```

L67 Drilling with facing and chip clearance



8.2.1 - 8 Cycle diagram L67

Input parameters

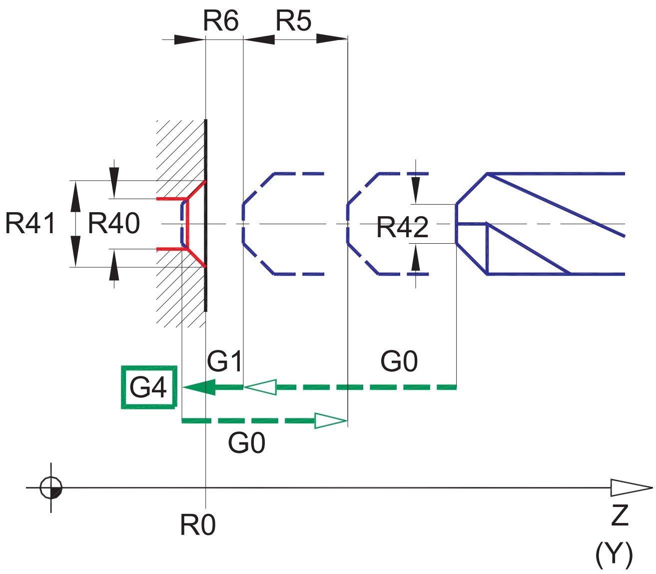
R0 Surface area
R2 Dwell time in seconds [s]
R3 Facing height
R5 Retraction
R6 Safety distance
R40 Facing diameter
R70 Feed rate factor in % for facing
R71 Speed factor in % when facing

HELLER

Application example

```
N100 X100 Y100
N110 R0=0 R2=1 R3=5 R5=200 R6=2 R40=33 R70=70
N120 R71=45
N130 L67
N140 X200
N150 R0=120
N160 L67
```

L68 Chamfering with countersink



8.2.1 - 9 Cycle diagram L68

Input parameters

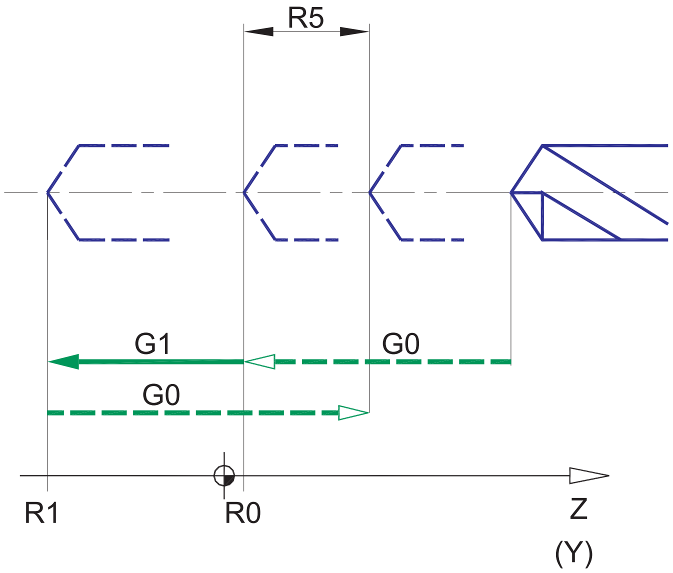
R0 Surface area
R2 Dwell time in seconds [s]
R5 Retraction
R6 Safety distance
R40 Drilling diameter
R41 Chamfer diameter
R42 Tool flattening

Application example

```
N100 X100 Y100
N110 R0=0 R2=1 R5=200 R6=2 R40=10 R41=15 R42=2
N120 L68
N130 X200
N140 R0=120
N150 L68
```

8.2.1.2 Drilling cycles according to DIN 66025

L81 Drilling



8.2.1 - 10 Cycle diagram L81

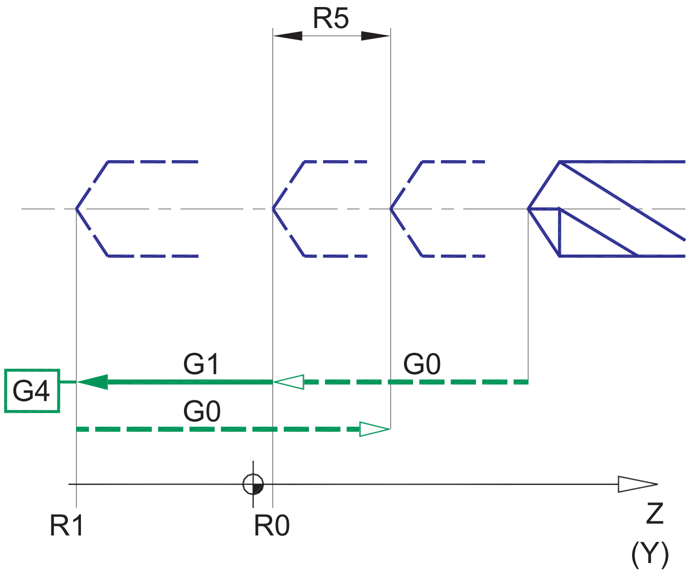
Input parameters

R0 Reference plane
R1 Drilling depth
R5 Retraction, incremental dimension

Application example

```
N100 X100 Y100
N110 R0=2 R1=-30 R5=100
N120 L81
N130 X200
N140 R5=0
N150 L81
```

L82 Drilling with chip clearance



8.2.1 - 11 Cycle diagram L82

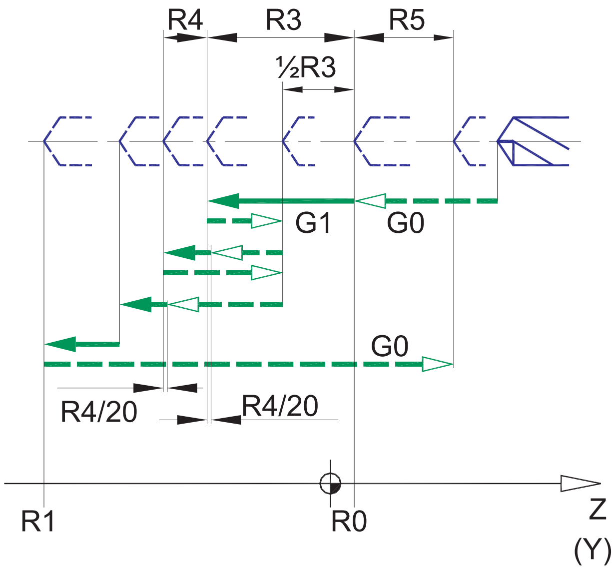
Input parameters

R0 Reference plane
R1 Drilling depth
R2 Dwell time in seconds [s]
R5 Retraction, incremental dimension

Application example

```
N100 X100 Y100
N110 R0=0 R1=-30 R5=100 R2=0.2
N120 L82
N130 X200
N140 R5=0
N150 L82
```

L83 Deep hole drilling



8.2.1 - 12 Cycle diagram L83

Input parameters

R0 Reference plane

R1 Final drilling depth

R3 1. Drilling depth, incremental dimension

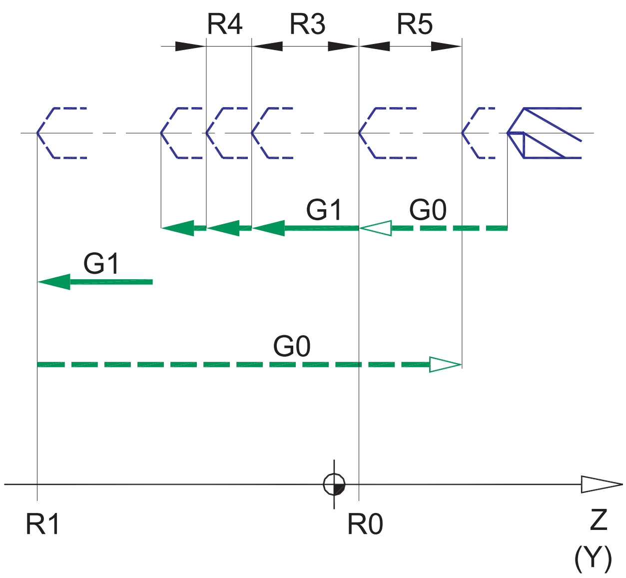
R4 Depth growth

R5 Retraction, incremental dimension

Application example

```
N080 X100 Y100
N090 R0=0 R1=-120 R3=30 R4=10 R5=50
N100 MCALL L83
N110 X100 Y100
N120 X200
N130 MCALL
```

L88 Drilling with chip breaking



8.2.1 - 13 Cycle diagram L88

Input parameters

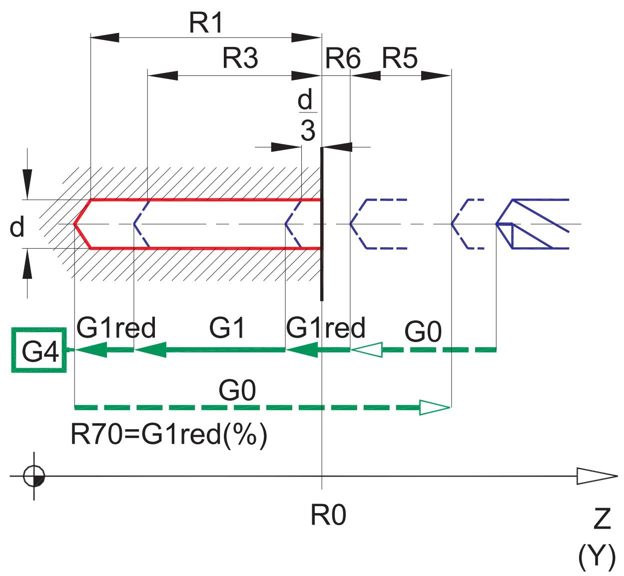
R0 Reference plane
R1 Final drilling depth
R2 Dwell time in seconds [s]
R3 1. Drilling depth, incremental dimension
R4 Depth growth
R5 Retraction, incremental dimension

Application example

```
N100 R0=0 R1=-120 R2=0.5 R3=-30 R4=10 R5=50
N110 MCALL L88
N120 X100 Y100
N130 X200
N140 MCALL
```

8.2.1.3 L01xx Other drilling cycles

L101 Drilling, F reduced for tapping and withdrawal



8.2.1 - 14 Cycle diagram L101

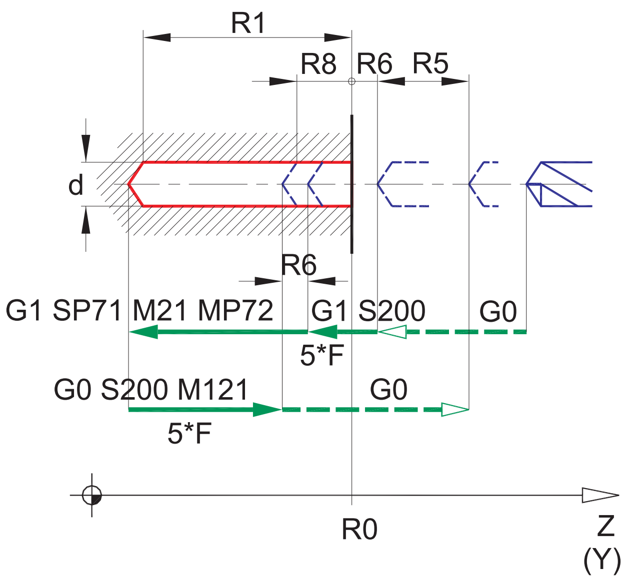
Input parameters

R0 Absolute surface
R1 Effective drilling depth
R2 Dwell time in seconds [s]
R3 Effective drill depth with feed 100%
R5 Retraction
R6 Safety distance
R70 Feed rate reduction in %

Application example

```
N100 R0=100 R1=60 R2=1 R3=50 R5=20 R6=2 R70=50
N110 MCALL L101
N120 X100 Y100
N130 X200
N140 MCALL
```

L102 Deep hole drilling, pilot drilling and high pressure



8.2.1 - 15 Cycle diagram L102

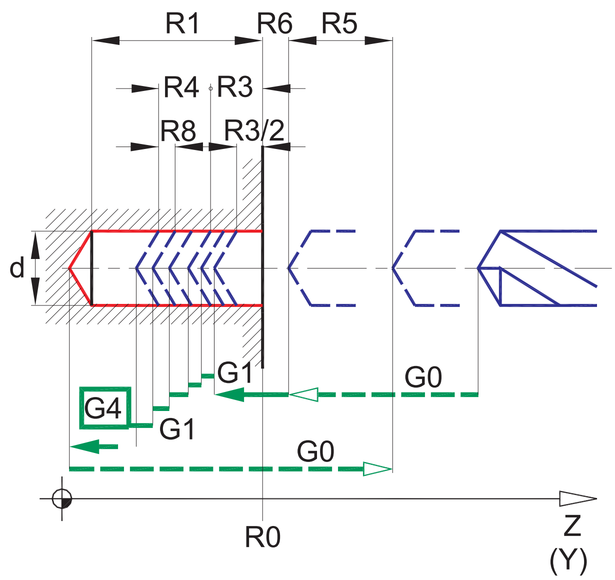
Input parameters

R0 Absolute surface
R1 Effective drilling depth
R5 Retraction
R6 Safety distance
R8 Pre-drilled drilling depth
R71 Machining speed [1/min] is activated in the sub-program
R72 Coolant pressure, M function M121...M127

Application example

```
N100 X100 Y200
N110 R0=100 R1=60 R5=20 R6=2 R8=20 R71=1000 R72=127
N120 L102
N130 X200
N140 L102
```

L103 Deep hole drilling with chip breaking



8.2.1 - 16 Cycle diagram L103

Input parameters

R0 Absolute surface

R1 Final drilling depth

R2 Dwell time in seconds [s]

R3 1. Drilling depth

R4 Depth growth

R5 Retraction

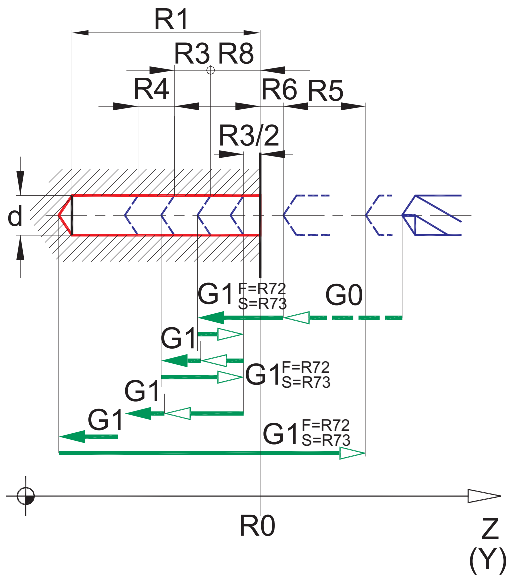
R6 Safety distance

R8 Depth increase for chip breaking

Application example

```
N110 R0=100 R1=120 R2=1 R3=20 R4=15 R5=20 R6=2 R8=3
     MCALL L103
N120 X200 Y100
N130 Y200
N140 MCALL
N150
```

L104 Deep hole drilling with reduction, pre-drilled



8.2.1 - 17 Cycle diagram L104

Input parameters

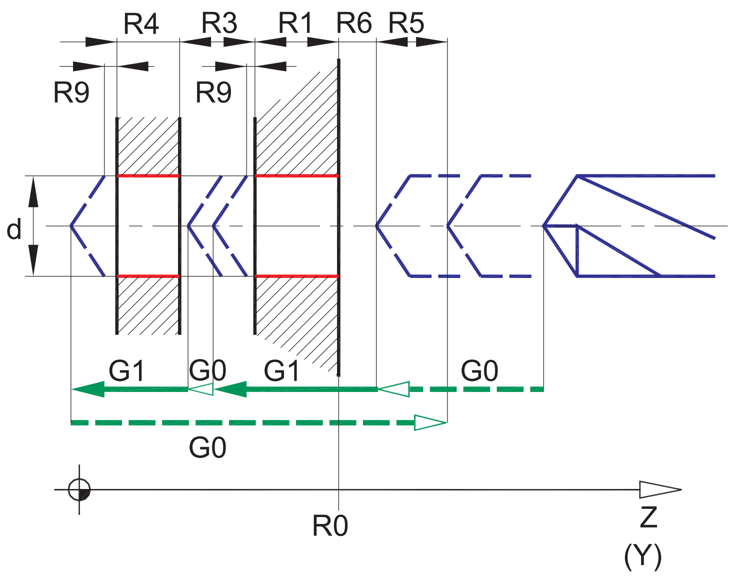
R0 Absolute surface
R1 Effective drilling depth
R3 1. Drilling depth
R4 Depth growth
R5 Retraction
R6 Safety distance
R8 Pre-drilled drilling depth
R70 Feed rate reduction in % (for depth growth in each case)
R71 Depth growth reduction in %
R72 Feedrate for pre-drilled hole

R73 Speed for pre-drilled hole

Application example

```
N100 X100 Y200
R0=100 R1=60 R3=30 R4=5 R5=200 R6=1 R8=20 R70=10 R71=20 R72=3
N110 L104
```

L105 Double-bottom drilling



8.2.1 - 18 Cycle diagram L105

Input parameters

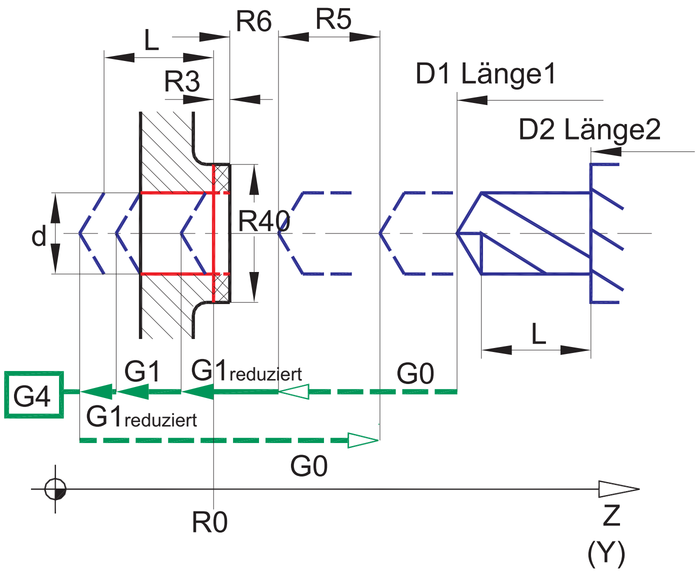
R0 Absolute surface
R1 1. Drilling depth
R3 Clearance
R4 2. Drilling depth
R5 Retraction
R6 Safety distance
R9 Overflow
R70 Feed rate reduction for tapping in %

HELLER

Application example

```
N100 X100 Y200
N110 R0=100 R1=20 R3=30 R4=25 R5=120 R6=2 R9=3
N120 R70=30
N130 L105
N140 X200
N150 L105
```

L106 Drilling with tapping, facing



8.2.1 - 19 Cycle diagram L106

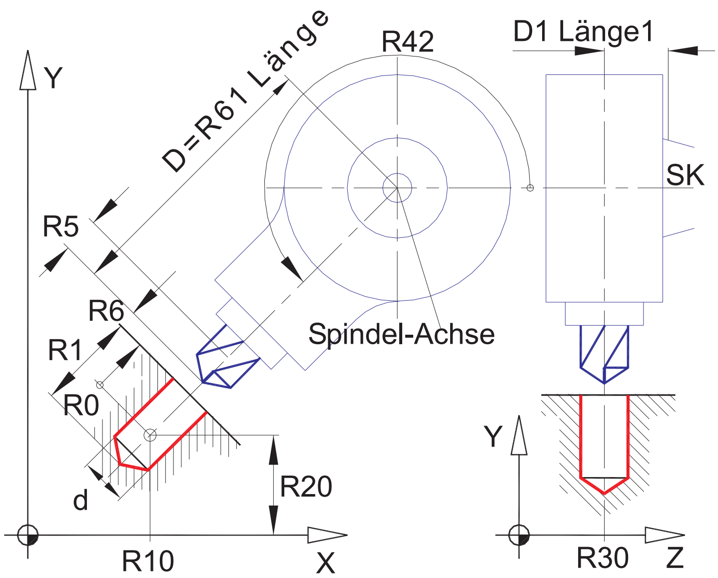
Input parameters

R0 Absolute surface
R2 Dwell time in seconds [s]
R3 Facing height
R5 Retraction
R6 Safety distance
R40 Facing diameter
R70 Feed rate reduction for tapping in %

Application example

```
N100 X100 Y200
N110 R0=100 R3=5 R5=120 R6=2 R40=25 R70=30
N120 L106
N130 X200
N140 L106
```

L107 Drilling in XY plane using angle drilling head



8.2.1 - 20 Cycle diagram L107

Input parameters

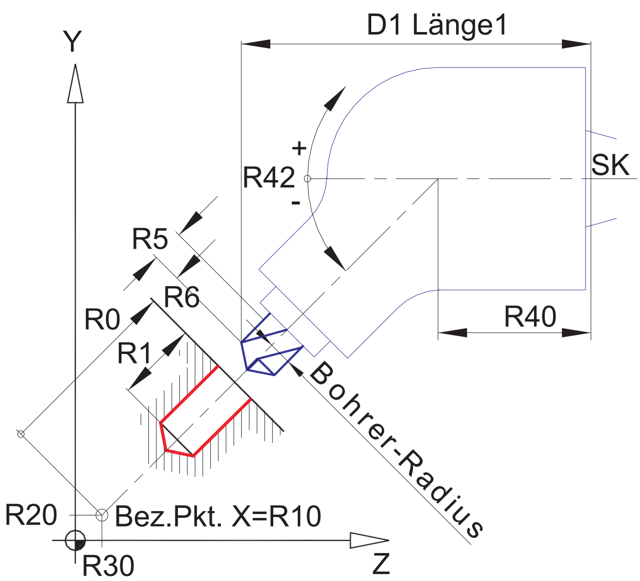
R0 Reference plane with respect to reference point
R1 Drilling depth
R2 Dwell time in seconds [s]
R5 Retraction
R6 Safety distance
R10 X coordinate
R20 Y coordinate
R30 Z coordinate
R42 Drilling angle to +X
R61 Cutting edge 1...9

HELLER

Application example

```
N100 X100 Y200 Z400 D1
N110 R0=100 R1=20 R2=0.2 R5=2 R6=200 R10=300 R20=150 R30=225 I
N120 L107
```

L108 Drilling in YZ plane using angle drilling head



8.2.1 - 21 Cycle diagram L108

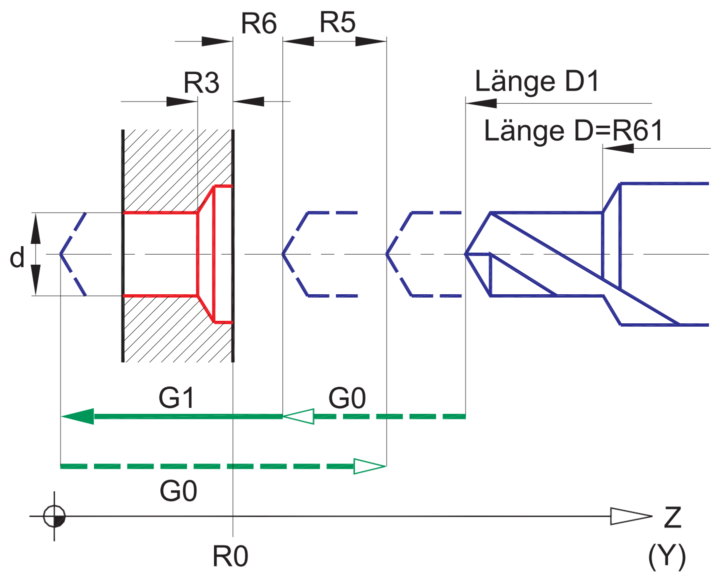
Input parameters

R0 Reference plane with respect to reference point
R1 Drilling depth
R5 Retraction
R6 Safety distance
R10 X coordinate reference point
R20 Y coordinate reference point
R30 Z coordinate reference point
R40 Length to tool axis intersection
R42 YZ angle of inclined tool axis

Application example

```
N2850 G0 G56 X-85 Y-273 F223 S1393 M4 M8
N2860 Z118
N2870 R0=5 R1=35 R5=0 R6=2 R10=-85 R20=-182 R30=18 R40=193.68
N2880 L108
```

L109 Drilling with step drill



8.2.1 - 22 Cycle diagram L109

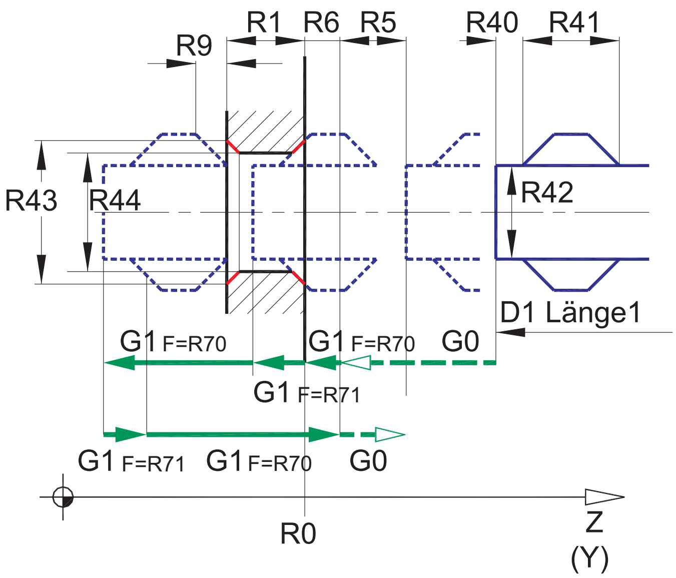
Input parameters

R0 Absolute surface
R3 Facing height
R5 Retraction
R6 Safety distance
R61 Cutting edge 1...9

Application example

```
N100 X100 Y200
N110 R0=100 R3=10 R5=0 R6=2 R61=2
N120 L109
N130 X200
N140 L109
```

L110 Deburring with Heule tool, through bore



8.2.1 - 23 Cycle diagram L110

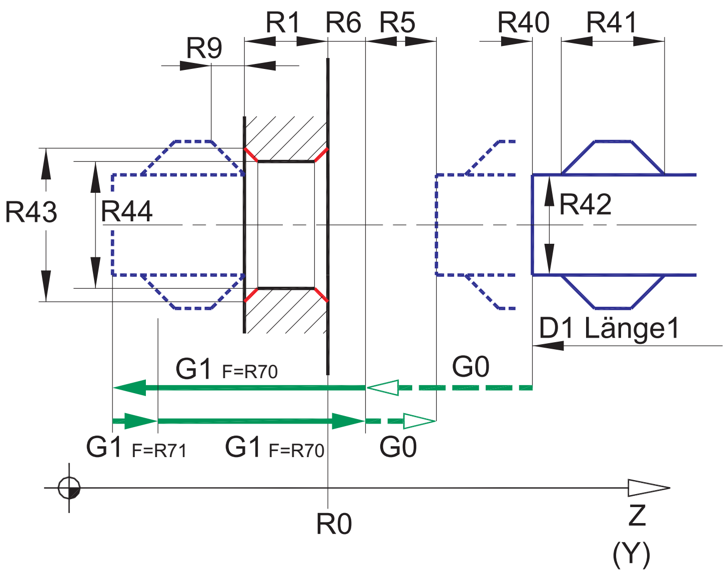
Input parameters

R0 Absolute surface
R1 Wall thickness, R1=0 only deburr at front
R2 Dwell time in seconds [s]
R5 Retraction
R6 Safety distance
R9 Passage injection
R40 Heule dimension A
R41 Heule dimension B
R42 Heule dimension D1
R43 Countersink diameter
R44 Drilling diameter
R70 Approach or traversing feedrate
R71 Machining feed

Application example

```
N100 X100 Y200
N110 R0=0 R1=20 R2=3 R5=0 R6=3 R9=5 R40=2 R41=6 R42=7.5 R43=10
N120 L110
```

L111 Heule deburr, hole exit



8.2.1 - 24 Cycle diagram L111

Input parameters

R0 Absolute surface
R1 Wall thickness, R1=0 only deburr at front
R2 Dwell time in seconds [s]
R5 Retraction
R6 Safety distance
R9 Passage injection
R40 Heule dimension A
R41 Heule dimension B
R42 Heule dimension D1
R43 Countersink diameter
R44 Drilling diameter
R70 Approach or traversing feedrate
R71 Machining feed

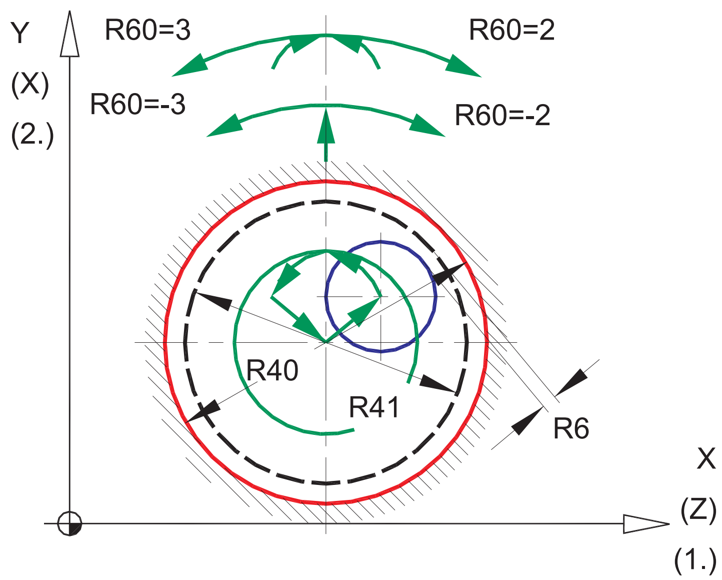
Application example

```
N100 X100 Y200
N110 R0=0 R1=20 R2=3 R5=0 R6=3 R9=10 R40=5 R41=12 R42=22 R43=
N120 L111
```

8.2.2 L1xxx Milling sub-programs

8.2.2.1 L10xx Milling inner contours

L1001 Milling bore



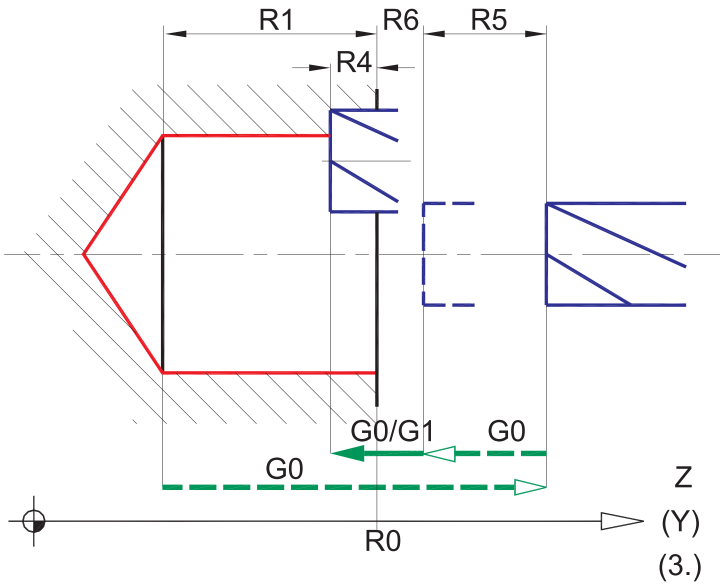
8.2.2 - 1 Machining plane cycle diagram L1001

Input parameters for machining plane

R40 Drilling diameter

R41 Unfinished diameter

R60 Up-milling =2, down-milling =3, straight move in -2, -3



8.2.2 - 2 Feed plane cycle diagram L1001

Feed plane input parameters

R0 Surface area
R1 Final depth
R4 Depth growth
R5 Retraction
R6 Radial and axial safety clearance

Sequence description

The drilling axis moves in rapid traverse to the safety clearance from the centre of the drilling that has already been approached before calling up L1001. If the unfinished diameter is less than the tool diameter with safety clearance, reduced feed is applied, otherwise the feed proceeds to the depth at rapid traverse. If the hole is machined in a single cut, R4=R1 is specified, otherwise machining should be in the steps specified by R4 up to the final depth R1. After the final depth R1 has been reached, the tool is positioned at the centre of the bored hole and retracted along the boring axis to the withdrawn position.

R60 is used to select up cut or down cut (provided clockwise tool is used, G2 orG3). If straight approach to the drilling contour is required instead of entry/exit quarter circles to save time, this is done by specifying negative values of R60.

HELLER

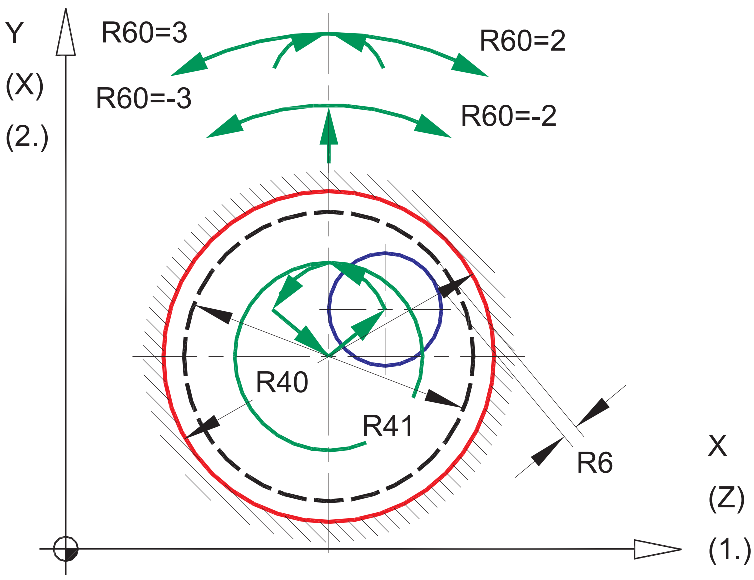
The circle contour is approached taking into account the safety distance R6. If the difference between the tool diameter and unfinished diameter is less than the safety distance R6, entry takes place with G1 instead of G0. If the unfinished diameter is greater than the drilling diameter, an error message is issued.

To prevent marking of flat surfaces, the drilling axis is moved a few 1/100 mm when entering and extending.

Application example

```
N100 X100 Y100
N110 R0=0 R1=30 R4=10 R5=50 R6=2 R40=100 R41=98 R60=2
     L1001
N120
```

L1002 Milling internal threads



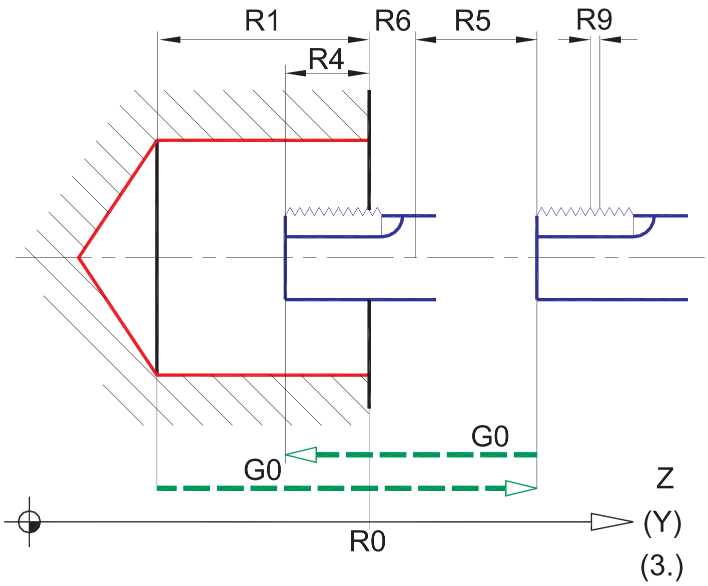
8.2.2 - 3 Machining plane cycle diagram L1002

Input parameters for machining plane

R40 Nominal diameter of thread

R41 Core diameter

R60 Up-milling =2, down-milling =3, straight move in -2, -3



8.2.2 - 4 Feed plane cycle diagram L1002

Feed plane input parameters

R0 Surface area
R1 Final depth
R4 Depth growth
R5 Retraction
R6 Radial and axial safety clearance
R9 Thread pitch, r.h. thread positive, l.h. thread negative

Sequence description

The drilling axis moves in rapid traverse to the required depth along the centre of the drilling that was already approached before calling up L1002.

If the hole is machined in a single cut, R4=R1 is specified, otherwise machining is carried out in the steps specified by R4 up to the final depth R1, with the depth growth always being rounded off to a multiple of the thread pitch. If the thread is to be cut using a single-tooth milling tool (whirling), the depth growth R4 is specified equal to the pitch R9 and milling then takes place in a spiral form until the final depth is reached. When the final depth R1 is reached, the tool is positioned to the centre of the drilling and the drilling axis withdraws.

Milling of the thread takes place with entry and exit quadrants that are determined automatically from the core diameter. To prevent back cutting, the proportionate thread pitch is taken into account for the entry circle. The circle contour is approached taking into account the safety distance R6. If the distance between the tool and the core diameter is less than the safety distance R6, an error message is issued.

Up cut or down cut is selected using R60. (G2 or G3, clockwise tool) If straight approach to the drilling contour is required instead of entry/ exit quarter circles to save time, this is done by specifying negative values ofR60.

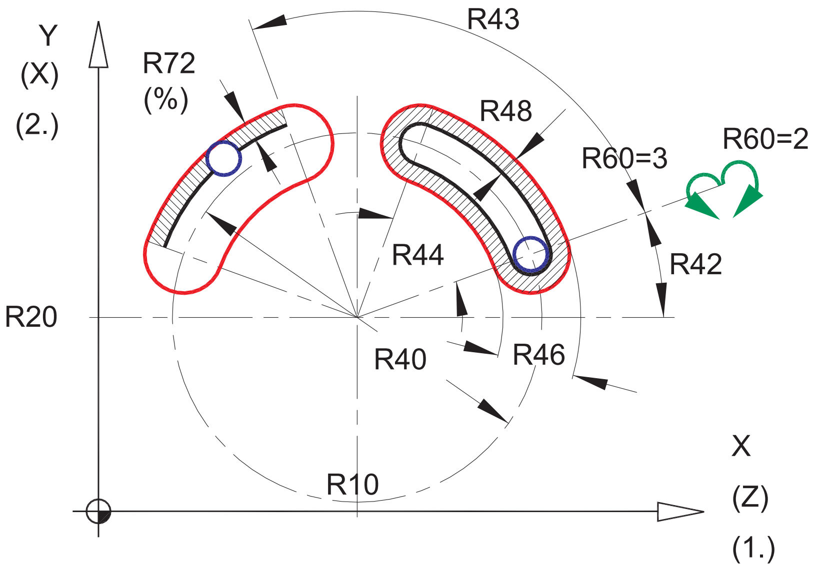
A r.h. thread is programmed with positive pitch, a l.h. thread with negative pitch R9.

Care must be taken that the core hole depth is sufficient for the proportionate pitch of the exit quadrant. It is a maximum 1/8 pitch.

Application example

```
N100 X100 Y100;left-hand thread in 2 passes;
N110 R0=100 R1=50 R4=26 R5=0 R6=2 R9=-2 R40=100 R41=98R60=2
     L1002
N120 X200
N130 R5=50
N140 L1002
N150
```

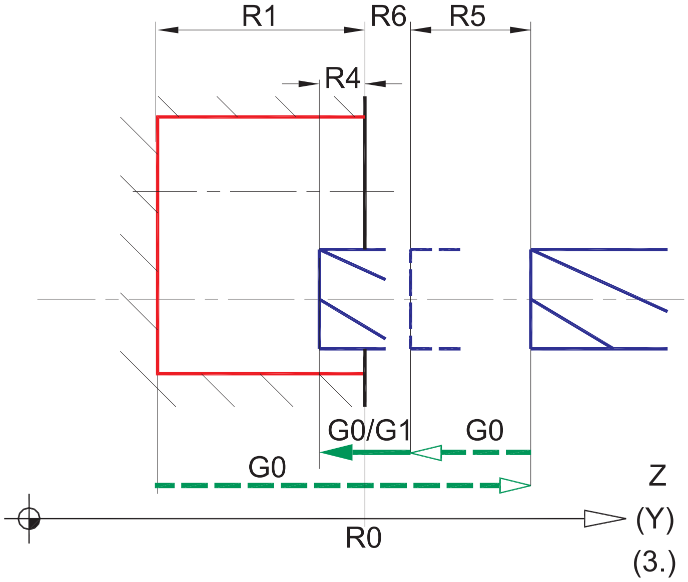
L1003 Milling circular grooves



8.2.2 - 5 Machining plane cycle diagram L1003

Input parameters for machining plane

R10 Reference circle centre point (G17 X, G18Z)
R20 Reference circle centre point (G17 Y, G18 X)
R40 Pitch circle diameter
R42 Initial angle
R43 Incremental angle
R44 Groove angle
R46 Groove width
R48 Allowance, from the solid >=1/2 groove width
R65 Number of circular grooves
R60 Up-milling =2, down-milling =3
R72 Cut width in % of tool diameter



8.2.2 - 6 Feed plane cycle diagram L1003

Feed plane input parameters

R0	Surface area
R1	Final depth
R4	Depth growth
R5	Retraction
R6	Safety distance
R70	Plunge feed in % of machining feed

Sequence description

Traverse takes place in the plane to the start point above the first circular grooves. Rapid traverse then takes place to the safety clearance.

The groove is milled in the steps specified by R4. If the groove is to be milled in one feed, R4=R1 is specified.

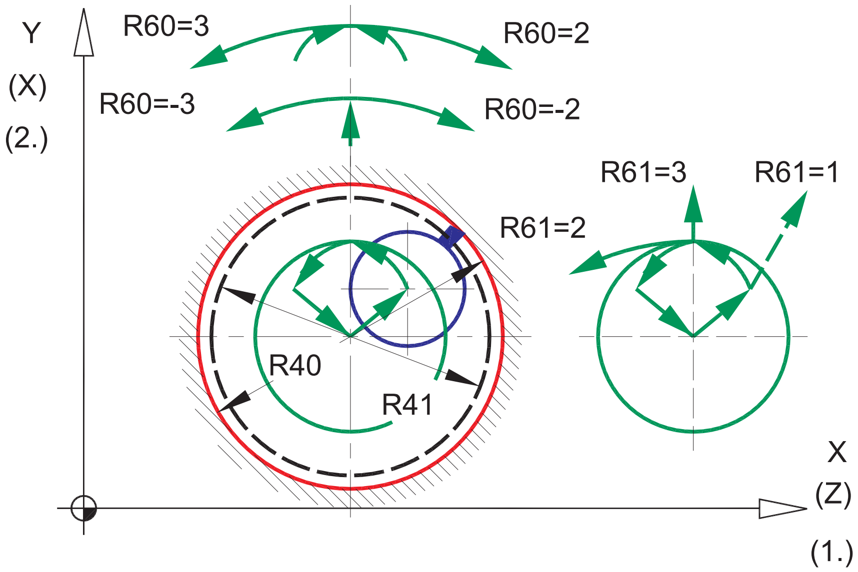
Milling takes place with the specified maximum cut width in % of the milling tool diameter. The generation of the milling path until the finished contour takes place according to the specified allowance R48. The contour is approached in a semicircle in each case.

Movement takes place to the withdrawn position when the final depth is reached. All the above sequences are repeated until the specified number of circular grooves have been machined.

Application example

```
N100 R10=100 R20=200 R40=150 R42=30 R43=45 R44=20 R46=20
     R48=6 R65=7 R60=2 R72=50
N110 R0=300 R1=20 R4=8 R5=100 R6=2 R8=15 R70=30
N120 L1003
N130
```

L1004 Cutting the internal circlip groove (Seeger circlip)



8.2.2 - 7 Machining plane cycle diagram L1004

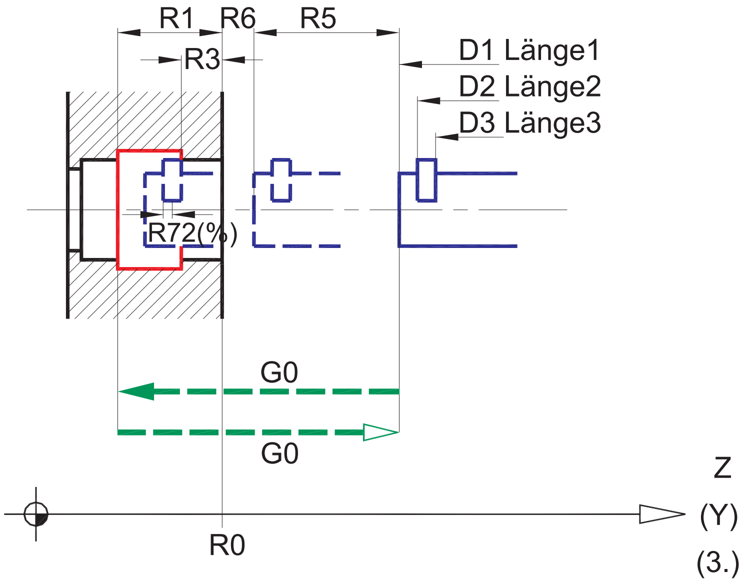
Input parameters for machining plane

R40 Outer diameter of groove

R41 Inner diameter of groove

R60 Up-milling =2, down-milling =3, straight move in -2, -3

R61 Depth feed, with extending=1, spiral form=2, without extending=3



8.2.2 - 8 Feed plane cycle diagram L1004

Feed plane input parameters

R0 Surface area
R1 Final depth
R3 1st depth
R5 Retraction
R6 Safety distance
R72 Depth increase in % of cutting edge width

i

The front cutting edge of the tool is specified with D2, the rear cutting edge with D3. The total length of the tool is given under D1 in the tool data.

Sequence description

The drilling axis moves in rapid traverse to the 1st depth along the centre of the drilling that was already approached before calling up L1004. (rear tool cutting edge to depth R3).

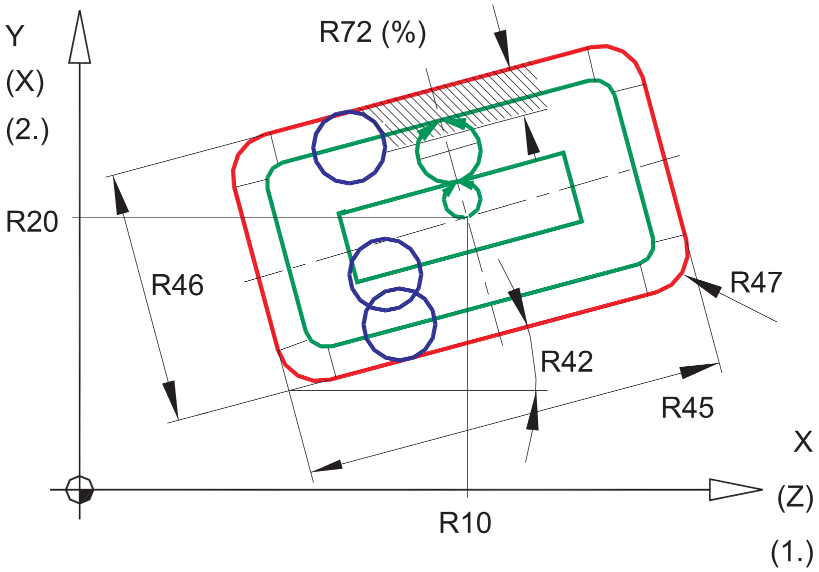
The plunge into the groove usually takes place in a quadrant. If straight traverse is preferred, a negative up-milling or down-milling R60 is specified. Feed takes place in a full circle in the drilling axis in accordance with the required type of depth feed R61 until the final depth is reached. After the final feed, the cutter returns to the centre of the drilling and the withdrawn position.

1. R61=1 – Extending with exit quadrant, feed, entry quadrant
2. R61=2 – Spiral form feed
3. R61=3 – Feed without extending

Application example

```
N100 X100 Y100
N110 R40=60 R41=55 R60=2 R61=1
N120 R0=100 R1=40 R3=10 R72=50 R5=200 R6=2
N130 L1004
```

L1005 Milling rectangular pocket

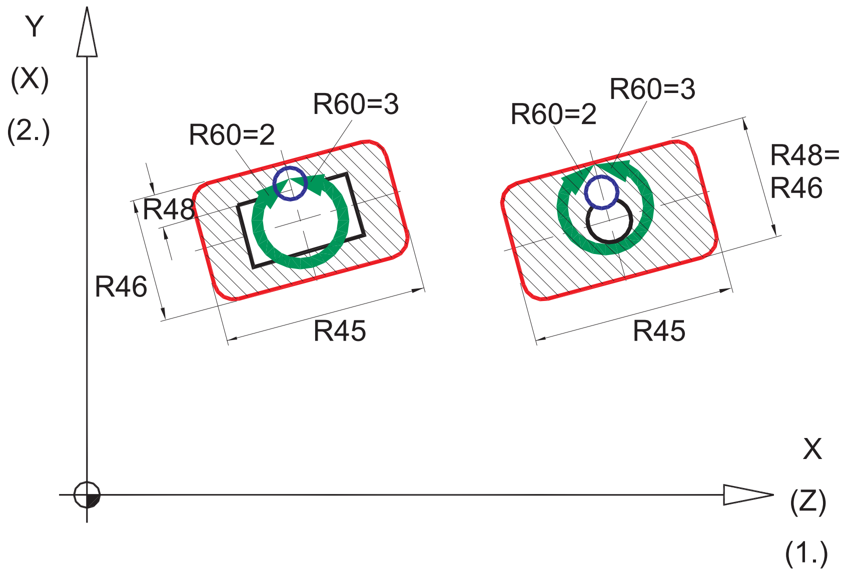


8.2.2 - 9 Machining plane cycle diagram L1005

Input parameters for machining plane

R10 Centre of pocket
R20 Centre of pocket
R42 Inclination angle of pocket
R45 Length of pocket
R46 Width of pocket
R47 Corner radius

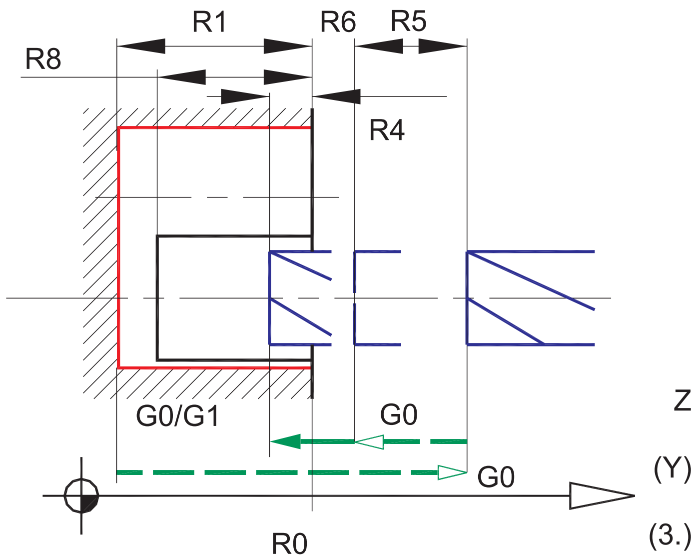
R70 Feed rate at pocket corners in % of machining feedrate
R72 Max. width of cut in % of tool diameter



8.2.2 - 10 Milling mode cycle diagram L1005

Milling mode - input parameters

R48 Allowance, from the solid R48 >= 1/2 width
R60 Up-milling =2, down-milling =3



8.2.2 - 11 Feed plane cycle diagram L1005

Feed plane input parameters

R0 Surface area
R1 Final depth
R4 Depth growth
R5 Retraction
R6 Safety distance
R8 Plunge drilling depth
R71 Plunge feed rate in % of machining feed rate, no bore R8=0

Sequence description

Rapid traverse first takes place to the centre of the pocket and then rapid traverse up to the safety clearance from the surface. There may already be a plunge drilling at the centre of the pocket. If there is no plunge drilling, its depth is specified as R8=0 and feed then always takes place at the plunge feed specified. Otherwise, the plunge drilling is entered at rapid traverse until its depth is reached, then at plunge speed. The pocket is milled in the steps specified by R4 until the final depth R1 is reached. If the pocket is to be milled in one feed,R4=R1 is specified.

Broaching takes place taking into account the allowance R48, up cut/ down cut and the specified max. cut width with equally wide milling paths. If the pocket is milled from the solid, the allowance R48 is specified as greater than or equal to half of the width R46. If only finishing is required, a milling path for finishing the pocket contour results automatically from the small allowance of finishing cut.

Feed into the corners of the pocket takes place with reduced feed rate (R70) because of the enlarged wrap angle at the milling tool. If no reduction is preferred, R70=100 is specified.

When the final depth is reached, positioning takes place to the centre of the slot and the withdrawn position.

i

L1005 can also be used to mill a circular pocket, both the length and the width being specified as equal to twice the radius of the corner.

HELLER

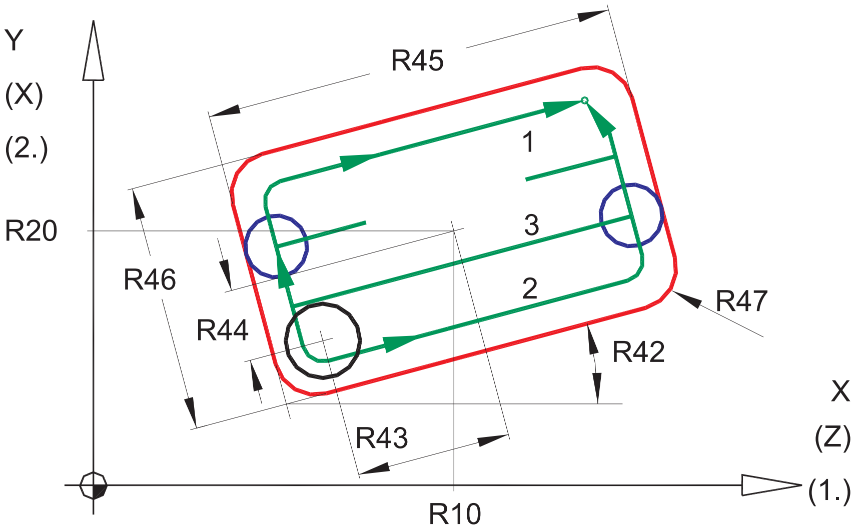
Application example

```
N100 R10=100 R20=200 R42=30 R45=100 R46=80 R47=20 R48=40
     R72=65 R70=50 R60=2
N110 R0=200 R1=30 R8=25 R4=10 R5=50 R6=2 R71=50
N120 L1005
N130
```

L1006 Milling a rectangular pocket horizontally

i

This special milling procedure when a horizontal spindle is used permits better clearance of chips, especially when “hedgehog” millers are used and prevents the trapping of chips.



8.2.2 - 12 Machining plane cycle diagram L1006

Input parameters for machining plane

R10 Centre of pocket (G17 X, G18 Z)

R20 Centre of pocket (G17 Y, G18 X)

R42 Inclination angle of pocket

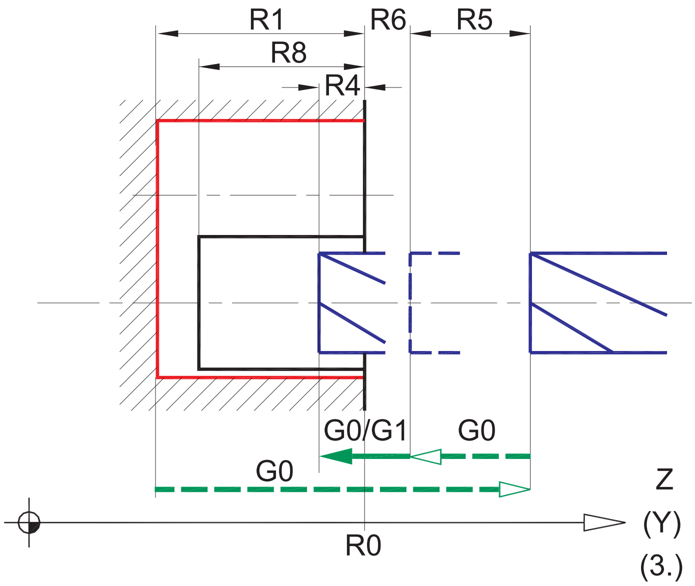
R43 Plunge drilling distance

R44 Plunge drilling distance

R45 Length of pocket

R46 Width of pocket

R47 Corner radius



8.2.2 - 13 Feed plane cycle diagram L1006

Feed plane input parameters

R0 Surface area

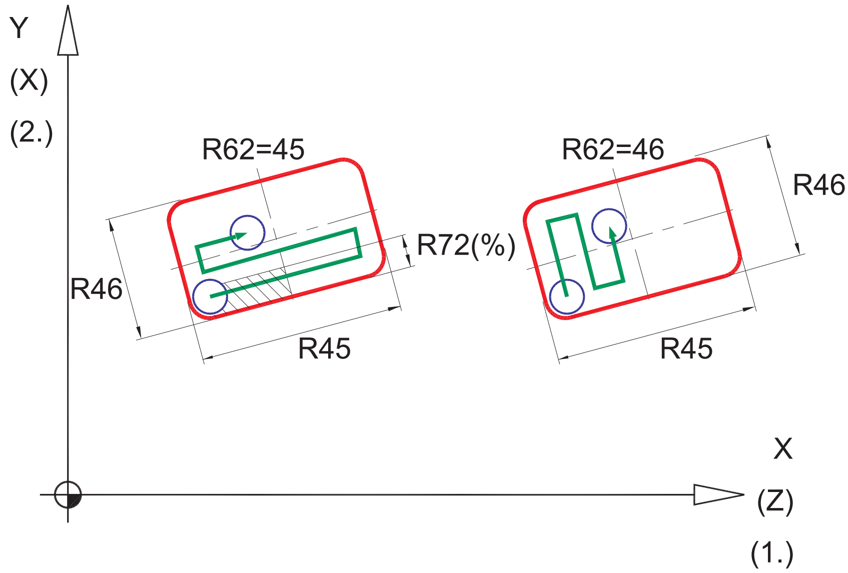
R1 Final depth

R4 Depth growth

R5 Retraction

R6 Safety distance

R8 Plunge drilling depth



8.2.2 - 14 Milling mode cycle diagram L1006

Milling mode - input parameters

R62 Milling parallel to length R62=45, parallel to width R62=46

R72 Max. cutting width in % of tool diameter

Sequence description

The first travel is to the plunge drilling position. This must be of a greater diameter than the tool and at least the depth of the pocket deep. The position of the plunge drilling must be close to the bottom left hand corner of the pocket as the first milling path starts from there. The plunge into the drilling takes place at 3 times the feed rate. The pocket is milled in the steps specified by R4 until the final depth R1 has been reached. If the pocket is to be milled in one feed,R4=R1 is specified.

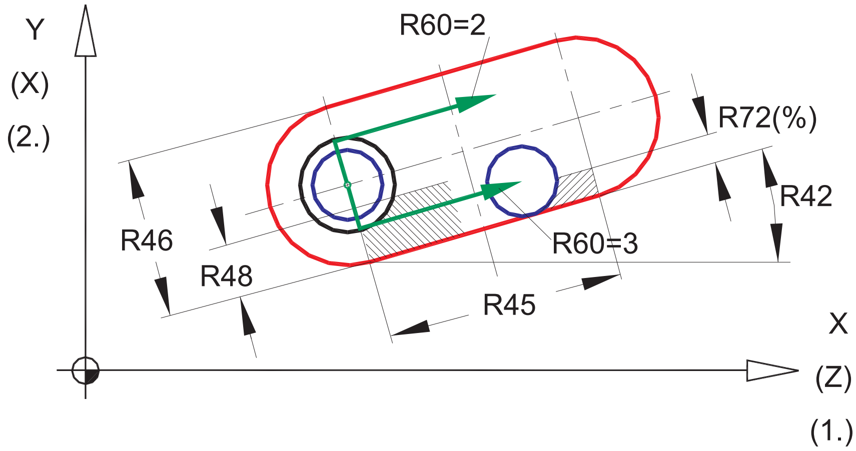
Broaching takes place taking into account the best possible chip removal with horizontal spindle, the external contour being broached first. A choice can then be made whether the milling path is to run parallel to the pocket length or width.

When the final depth is reached, positioning takes place to the centre of the slot and the withdrawn position.

Application example

```
N100 R10=200 R20=150 R42=45 R43=-40 R44=-40
N110 R45=140 R46=120 R47=20
N120 R0=0 R1=40 R4=20 R5=50 R6=2 R8=45
N130 R62=45 R72=70
N140 L1006
```

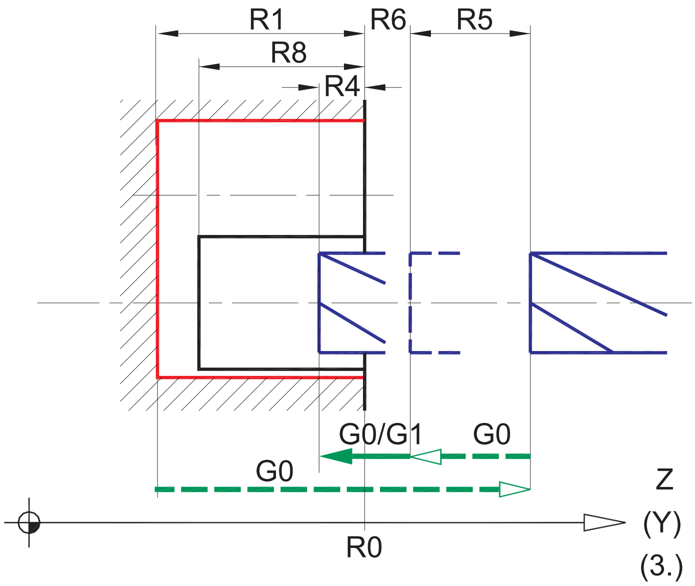
L1007 Groove milling



8.2.2 - 15 Machining plane cycle diagram L1007

Input parameters for machining plane

R42 Inclination angle of groove
R45 Groove length
R46 Groove width
R48 Allowance, (from the solid R48 >= 1/2 width)
R60 Up-milling =2, down-milling =3
R72 Maximum cut width in % of tool diameter



8.2.2 - 16 Feed plane cycle diagram L1007

Feed plane input parameters

R0 Surface area
R1 Final depth
R4 Depth growth
R5 Retraction
R6 Safety distance
R8 Plunge drilling depth
R70 Plunge feed in % of machining feed

Sequence description

Before calling up L1007, rapid traverse takes place along the approached position until the safety distance to the surface is reached. This represents the starting position (centre point of slot radius), any plunge drilling provided must be at this point. If there is no plunge drilling, its depth is specified as R8=0 and feed then always takes place at the plunge feed specified. Otherwise, the plunge drilling is entered at rapid traverse until its depth is reached, then at plunge speed. The groove is milled in the steps specified by R4 until the final depth R1 has been reached. If the groove is to be milled in one feed, R4=R1 is specified.

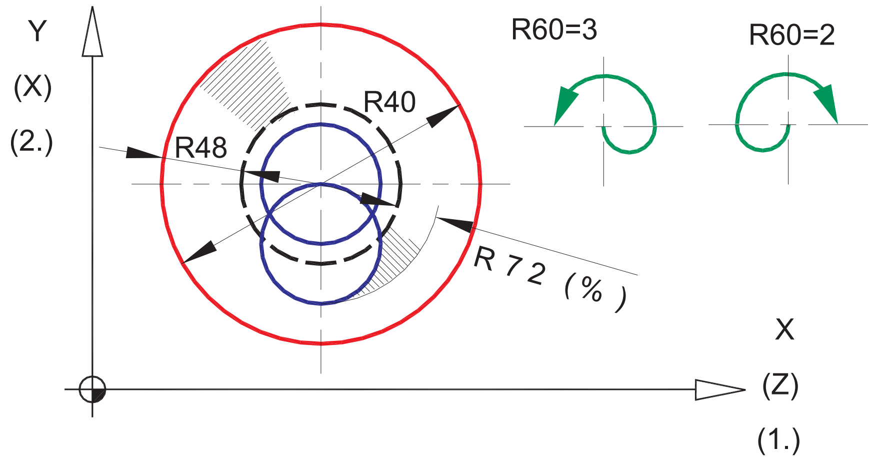
Broaching takes place taking into account the allowance, up cut/ down cut and the specified max. cut width with equally wide milling paths. If the groove is milled from the solid, the allowance R48 is specified as greater than or equal to (half of) the width R46. If only finishing is required, a milling path for finishing the slot contour results automatically from the small allowance of finishing cut.

When the final depth is reached, positioning takes place to the centre of the slot and the withdrawn position.

Application example

```
N100 R42=30 R45=100 R46=20 R48=20 R60=2 R72=75
N110 R0=0 R1=30 R4=10 R5=200 R6=2 R8=28 R70=50
N120 L1007
```

L1008 Spiral form surface milling of circular pocket



8.2.2 - 17 Machining plane cycle diagram L1008

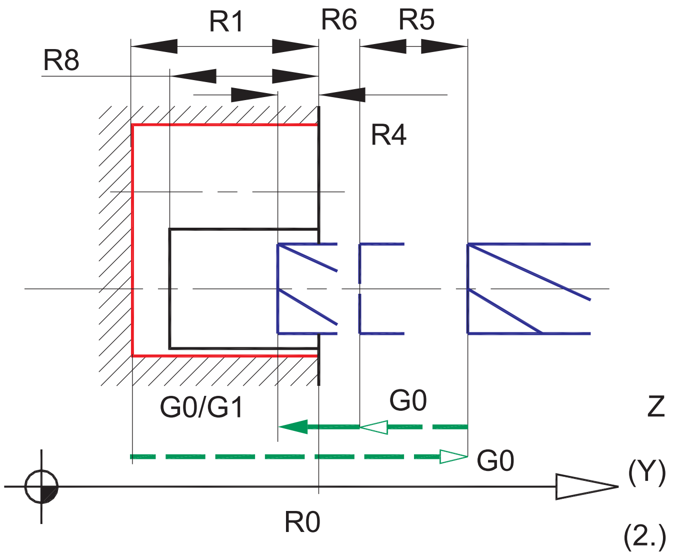
Input parameters for machining plane

R40 Diameter of circular area

R48 Radial allowance

R60 Up-milling =2, down-milling =3

R72 Width of cut in % of milling tool diameter



8.2.2 - 18 Feed plane cycle diagram L1008

Feed plane input parameters

R0 Surface area
R1 Final depth
R4 Depth growth
R5 Retraction
R6 Safety distance
R8 Plunge drilling depth
R70 Plunge feed in % of machining feed

Sequence description

Before calling up L1008, rapid traverse takes place along the approached position until the safety distance is reached. This represents the centre point of the circular pocket, any plunge hole provided must be at this point.

If there is no plunge drilling, its depth is specified as R8=0 and feed then always takes place at the plunge feed specified. Otherwise, the plunge drilling is entered at rapid traverse until its depth is reached, then at plunge speed. The surface is milled in the steps specified by R4 until the final depth R1 is reached. If the pocket is to be milled in one feed,R4=R1 is specified.

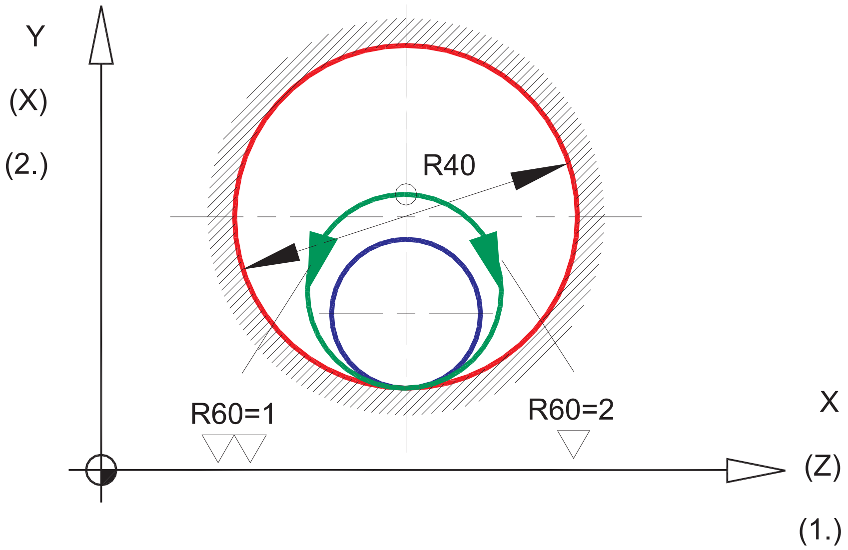
Broaching takes place taking into account the amount of cut, up cut/ down cut and the specified maximum cut width with equally wide spirally increasing milling paths. If the surface is milled from the solid, the allowance R48 is specified as greater than or equal to half of the diameter R40. If only finishing is required, a milling path for finishing the pocket contour results automatically from the small allowance of finishing cut.

When the final depth is reached, positioning takes place to the centre of the slot and the withdrawn position.

Application example

```
N100 X100 Y200
N110 R40=200 R48=40 R60=3 R72=80
     R0=200 R1=30 R4=20 R5=100 R6=1 R8=28 R70=80
     L1008
```

L1009 Thread milling with boring thread miller

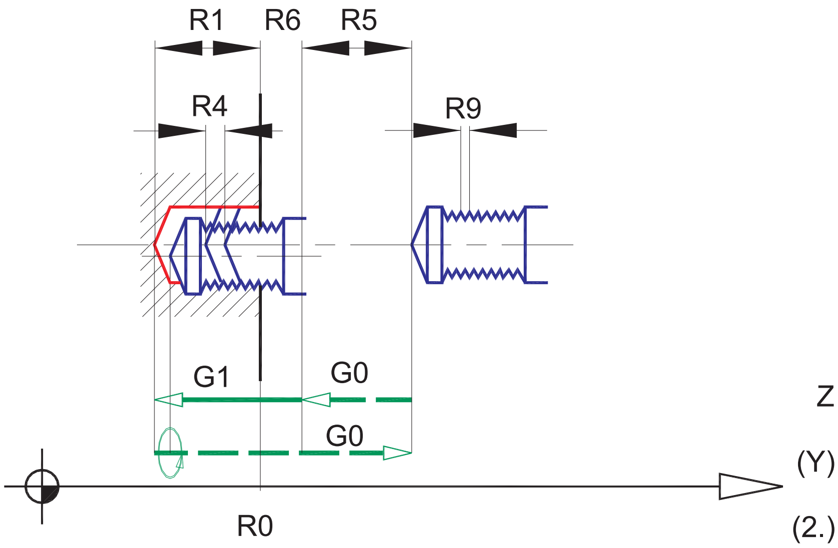


8.2.2 - 19 Machining plane cycle diagram L1009

Input parameters for machining plane

R40 Nominal diameter of thread

R60 One-step milling =1, with pre-milling = 2



8.2.2 - 20 Feed plane cycle diagram L1009

Feed plane input parameters

R0 Surface area
R1 Final depth
R4 Depth growth for drilling
R5 Retraction
R6 Safety distance
R9 thread pitch, right-handed=+, left-handed=-
R70 Drilling feedrate in % of finish milling
R71 Pre-milling feedrate in % of finish milling

Sequence description

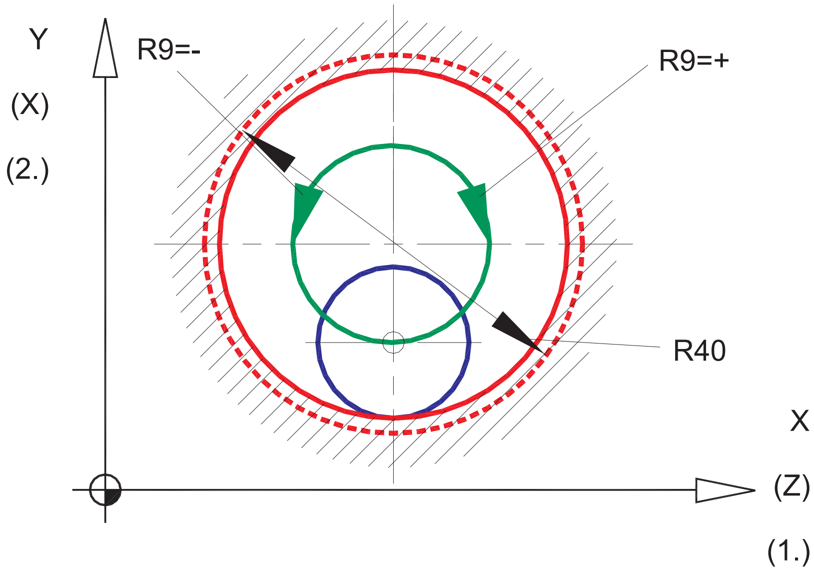
Travel takes place along the boring centre present before L1009 was called up to the safety clearance and boring then takes place to the final depth, with chip removal. If no chip removal is preferred, R4=R1 is specified.

After reaching the final depth, the tool is lifted off and the thread is cut in one pass. Approach to the thread is with a semicircle taking the proportionate thread pitch into account.

Application example

```
N100 X100 Y100
     R40=24 R60=2 R0=100 R1=30 R4=10 R5=50 R6=1 R9=2 R70=50 R
N110 L1009
N120 X200
N130 R5=0
N140 L1009
N150
```

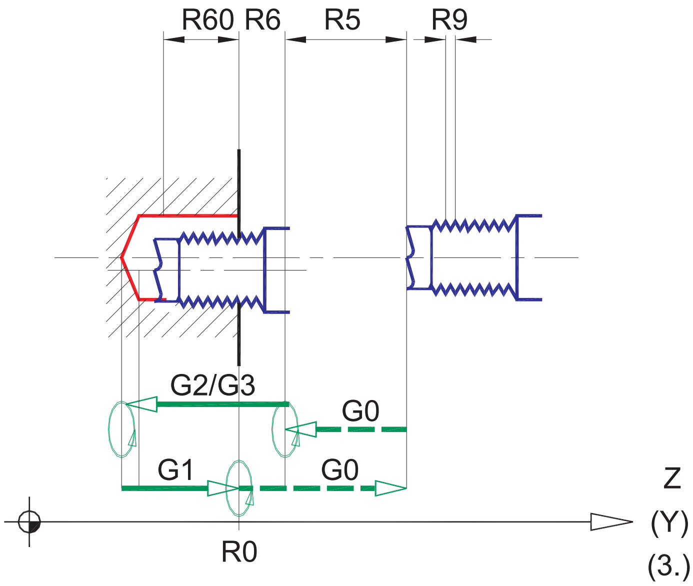
L1010 Thread milling with end milling thread miller



8.2.2 - 21 Machining plane cycle diagram L1010

Input parameters for machining plane

R40 Milling diameter of thread



8.2.2 - 22 Feed plane cycle diagram L1010

Feed plane input parameters

R0 Surface area

R5 Retraction

R6 Safety distance

R9 Thread pitch, r.h. thread =+, l.h. thread = -

R60 Number of pitches

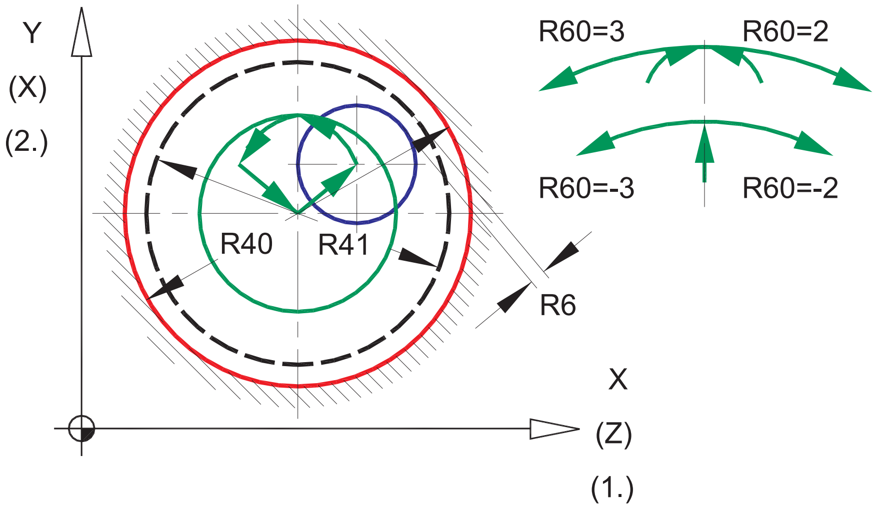
Sequence description

Use of a NORIS universal end milling thread miller allows a thread to be milled without first drilling a hole. The depth is specified by the number of pitches R60. Machining is completed by counterboring to a depth of 0,2*R9 with circular interpolation.

Application example

```
N100 X100 Y100
N110 R40=10.2 R60=18 R0=100 R5=50 R6=2 R9=+1.5
N120 L1010
N130 X200
N140 L1010
```

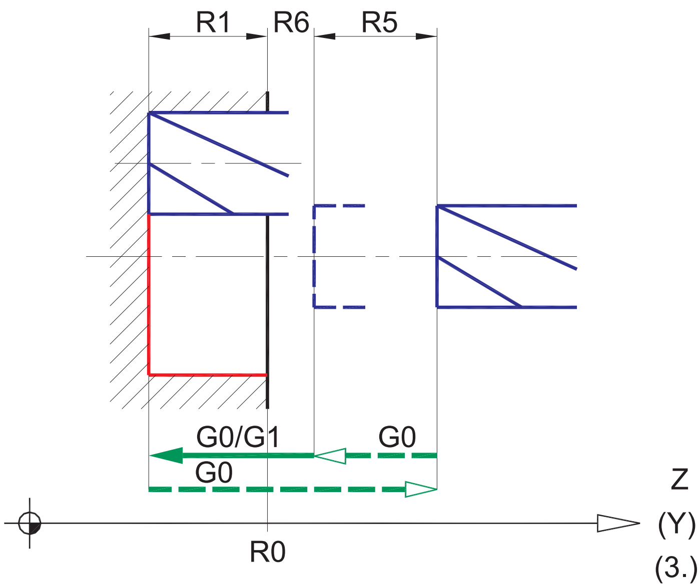
L1011 Milling bored hole without depth growth



8.2.2 - 23 Machining plane cycle diagram L1011

Input parameters for machining plane

R40 Drilling diameter
R41 Unfinished diameter
R60 Up-milling =2, down-milling =3, straight move in -2, -3



8.2.2 - 24 Feed plane cycle diagram L1011

HELLER

Feed plane input parameters

R0 Surface area
R1 Tiller cutting depth
R5 Retraction
R6 Radial and axial safety clearance

Sequence description

The drilling axis moves in rapid traverse to the safety clearance from the centre of the drilling that has already been approached before calling up L1011. If the unfinished diameter is less than the tool diameter with safety clearance, reduced feed is applied, otherwise the feed proceeds to the depth at rapid traverse. The boring is machined in one pass. After the circle is machined, the tool is positioned at the centre of the bored hole and withdrawn along the boring axis.

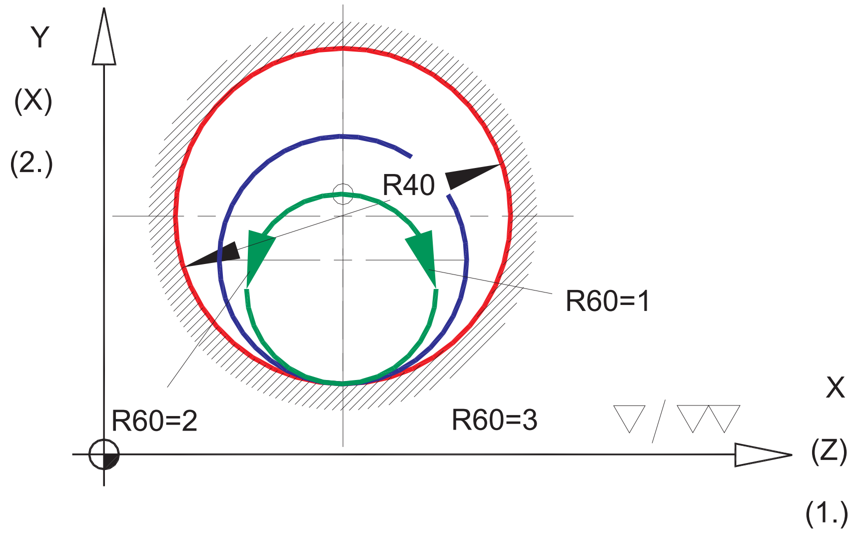
R60 is used to select up cut or down cut (provided clockwise tool is used, G2 orG3). If straight approach to the drilling contour is required instead of entry/exit quarter circles to save time, this is done by specifying negative values of R60.

The circle contour is approached taking into account the safety distance R6. If the difference between the tool diameter and unfinished diameter is less than the safety distance R6, entry takes place with G1 instead of G0. If the unfinished diameter is greater than the drilling diameter, an error message is issued.

Application example

```
N100 X100 Y100
N110 R40=100 R41=98 R60=2 R0=0 R1=30 R5=50 R6=2
N120 L1011
```

L1012 Thread milling with JEL boring thread miller BGF

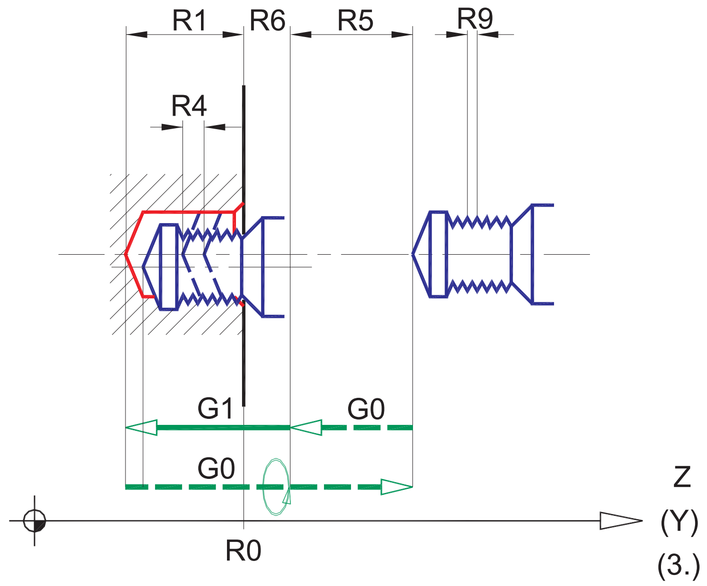


8.2.2 - 25 Machining plane cycle diagram L1012

Input parameters for machining plane

R40 Nominal diameter of thread

R60 Up-milling =1, down-milling =2, up-milling and down-milling (roughing and finishing) = 3



8.2.2 - 26 Feed plane cycle diagram L1012

Feed plane input parameters

R0 Surface area
R1 Depth (=dimension L2.7 from JEL catalogue)
R4 Depth growth for drilling
R5 Retraction
R6 Safety distance
R9 thread pitch, right-handed=+, left-handed=-
R70 Drilling feed rate
R71 Milling feed rate

Sequence description

Travel takes place along the boring centre present before L1012 was called up to the safety clearance and boring then takes place to the final depth, with chip removal. If no chip removal is preferred, R4=R1 is specified.

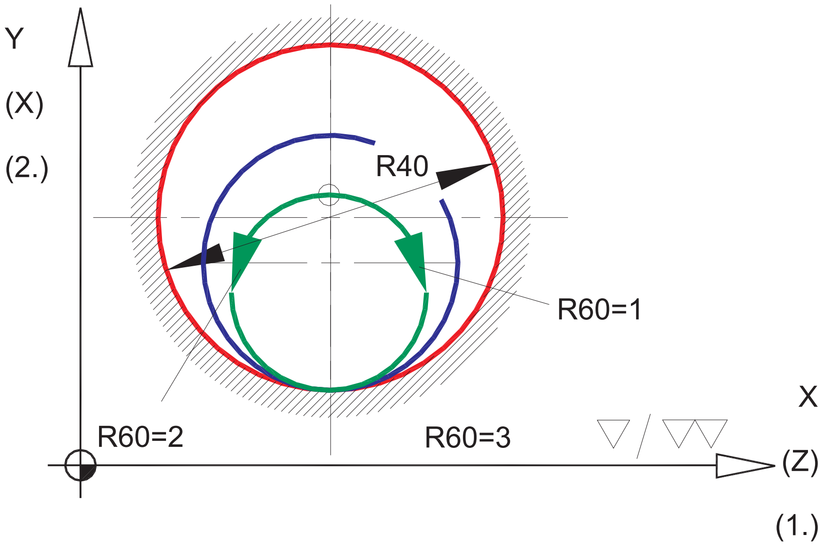
After the final depth is reached, the tool is lifted off by dimension "g" (JEL) and the thread is milled as specified by R60. Approach to and extend from the thread is with a semicircle taking the proportionate thread pitch into account.

Withdrawal from the bored hole then takes place at rapid traverse.

Application example

```
N100 MSG ("right-hand thread M8 with up-milling")
N110 X100 Y100
N120 R40=8 R60=1 R0=100 R1=18.16 R4=10 R5=50 R6=2
N130 R9=1.25 R70=2250 R71=1050
N140 L1012
N150 X200
N160 R5=0
N170 L1012
```

L1013 Thread milling with JEL mini thread millerMGF

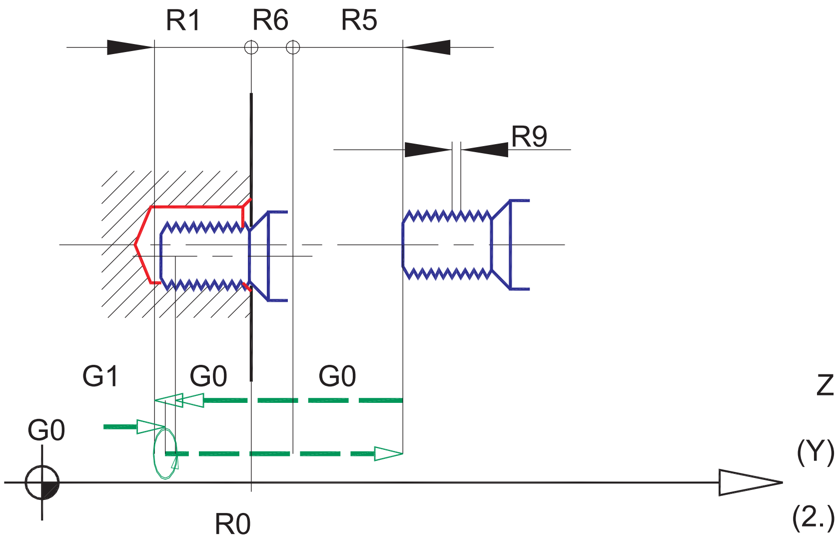


8.2.2 - 27 Machining plane cycle diagram L1013

Input parameters for machining plane

R40 Nominal diameter of thread

R60 Up-milling =1, down-milling =2, up-milling and down-milling (roughing and finishing) = 3



8.2.2 - 28 Feed plane cycle diagram L1013

Feed plane input parameters

R0 Surface area
R1 Depth (=dimension L2.7 from JEL catalogue)
R5 Retraction
R6 Safety distance
R9 thread pitch, right-handed=+, left-handed=-
R70 Drilling feed rate
R71 Milling feed rate

Sequence description

Positioning of the boring axis takes place at rapid traverse along the core hole bore centre, pre-bored before L1013 was called up, to the specified depth with 1mm safety clearance between the chamfering blade and the surface of the bore. Traverse than continues at countersink feed rate to the final depth and the chamfering is thus produced.

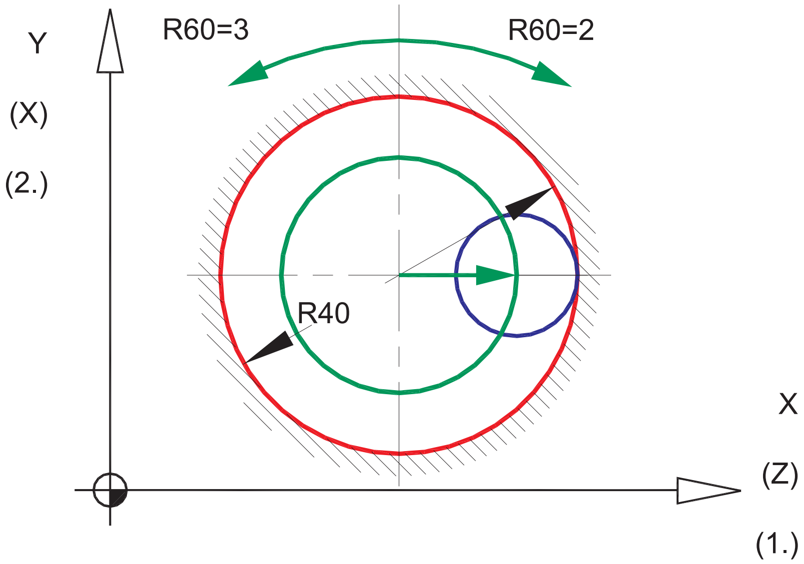
After the final depth is reached, the tool is lifted off by dimension "g" (JEL) and the thread is milled as specified by R60. Approach to and extend from the thread is with a semicircle taking the proportionate thread pitch into account.

Withdrawal from the bored hole then takes place at rapid traverse.

Application example

```
N100 X100 Y100
N110 R40=8 R60=1 R0=100 R1=18.16 R5=50 R6=2
N120 R9=1.25 R70=2250 R71=1050
N130 L1013
N140 X200
N150 R5=0
N160 L1013
```

L1014 Circular bore milling

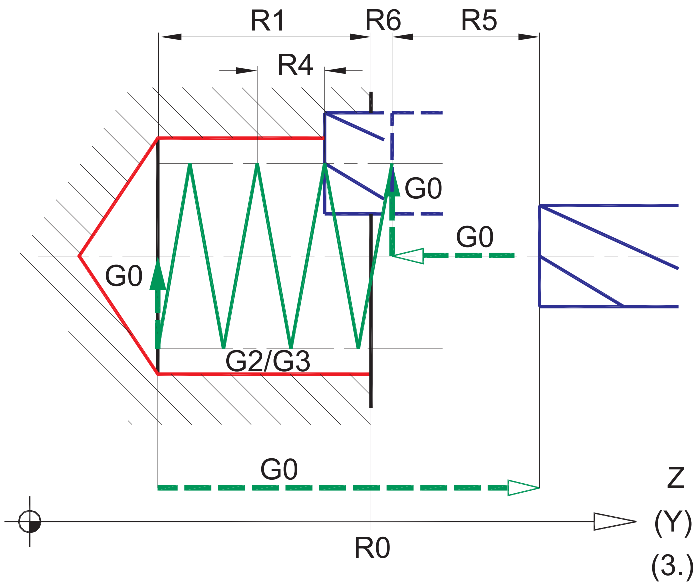


8.2.2 - 29 Machining plane cycle diagram L1014

Input parameters for machining plane

R40 Nominal diameter of thread

R60 Milling up-milling =2, down-milling =3



8.2.2 - 30 Feed plane cycle diagram L1014

Feed plane input parameters

R0 Surface area
R1 Final depth
R4 Depth growth
R5 Retraction
R6 Safety distance

Sequence description

Traverse must take place in the main program to the centre of the boring before L1014 is called up.

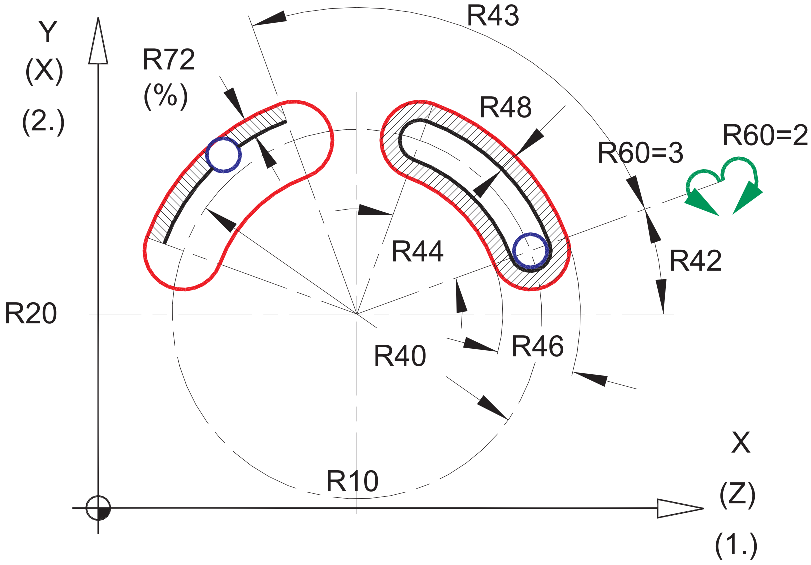
In the L1014 sub-program, a rapid traverse movement to the safety distance takes place in the spindle axis. Subsequently, the tool is moved to the programmed diameter in the positive 1st X/Z-axis. Bore milling is carried out with a circular movement in the programmed plane and, at the same time, a spindle axis feed in R4 steps. The programmed feed is held to the circular path. The remaining depth is divided up appropriately into quadrants. After the final depth is reached, a full circle is described to machine the cylindrical surface completely. Finally, rapid traverse takes place along the bore centre, the spindle axis being slightly lifted. Rapid traverse then takes place out of the bore to safety clearance and withdrawn position.

Application example

```
N100 X100 Y100
R40=100 R60=2 R0=100 R1=60 R4=15 R5=50 R6=5
N110 L1014
```

8.2.2.2 L1xxx Sub-programs for plunge milling

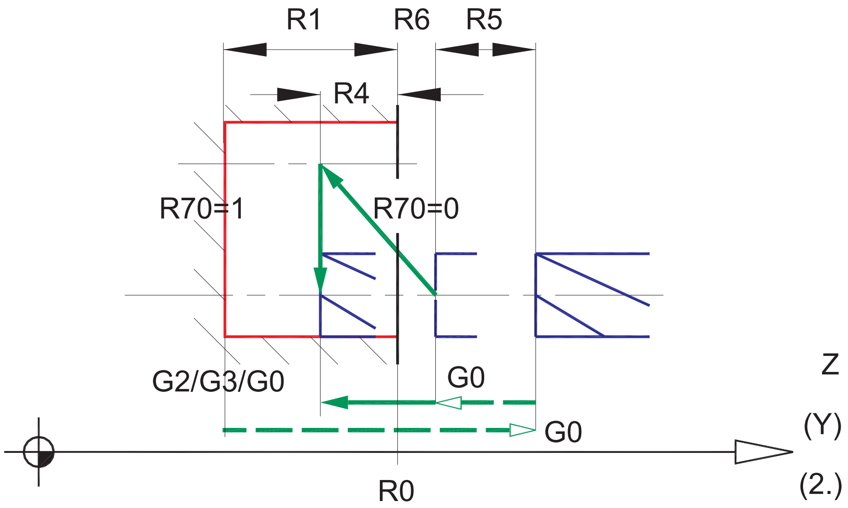
L1053 Milling circular grooves with plunge milling



8.2.2 - 31 Machining plane cycle diagram L1053

Input parameters for machining plane

R10 Reference circle centre point (G17 X, G18 Z)
R20 Reference circle centre point (G17 Y, G18 X)
R40 Pitch circle diameter
R42 Initial angle
R43 Incremental angle
R44 Groove angle
R46 Groove width
R48 Allowance, from the solid >=0,5*groove width
R60 Up-milling =2, down-milling =3
R65 Number of circular grooves
R70 Plunge, =0 without return traverse to depth
R70 Plunge, =1 with return traverse to depth
R71 Plunge feed in % of machining feed
R72 Cut width in % of tool diameter



8.2.2 - 32 Feed plane cycle diagram L1053

Feed plane input parameters

R0 Surface area

R1 Final depth

R4 Depth growth

R5 Retraction

R6 Safety distance

Sequence description

Traverse takes place in the XY plane to the start point above the first circular grooves. Rapid traverse then takes place to the safety clearance.

If R70=0 is programmed, the tool plunges to depth on 3 axes and then mills the groove. If R70=1 is programmed, the tool first plunges to depth on 3 axes and then returns to the start position depth.

However, if the allowance of the groove (R48) is so small that the milling tool can plunge without cutting, plunging takes place at rapid traverse in Z.

Milling takes place with the specified maximum cut width in % of the milling tool diameter. The generation of the milling path until the finished contour takes place according to the specified allowance R48. The contour is approached in a semicircle in each case.

The tool moves back to the retracted position when the depth is reached.

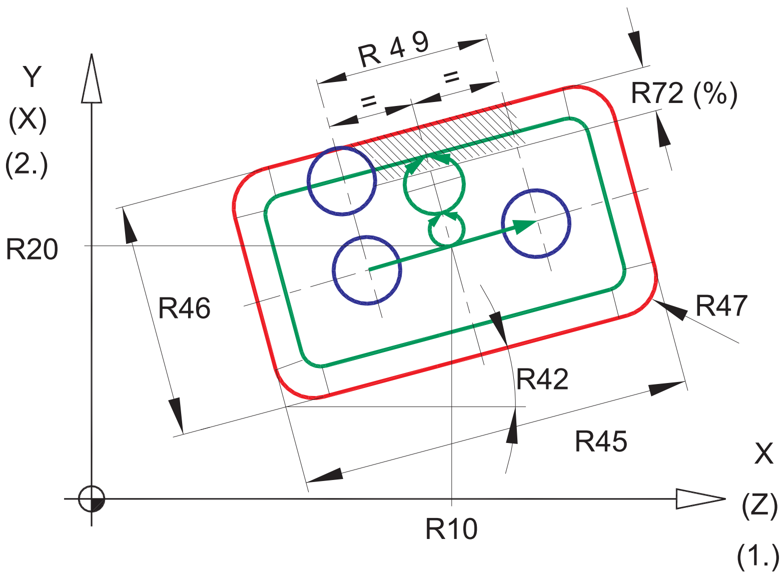
The groove is milled in the steps specified by R4. If the groove is to be milled in one feed, R4=R1 is specified.

All the above sequences are repeated until the specified number of circular grooves have been machined.

Application example

```
N100 R10=100 R20=200 R40=150 R42=30 R43=45 R44=20
N110 R46=20 R48=6 R60=2 R65=4 R70=1 R71=75 R72=50
N120 R0=300 R1=20 R4=10 R5=100 R6=2
N130 L1053
```

L1055 Milling a rectangular pocket with plunge milling



8.2.2 - 33 Machining plane cycle diagram L1055

Input parameters for machining plane

R10 Centre of pocket
R20 Centre of pocket
R42 Inclination angle of pocket
R45 Length of pocket

HELLER

R46 Width of pocket

R47 Corner radius

R48 Allowance, from the solid R48 >= 1/2width (see milling mode diagram)

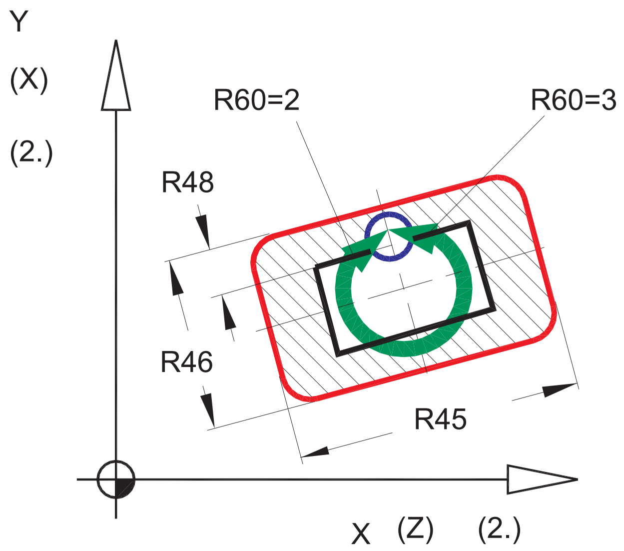
R49 Plunge length

R60 Up-milling =2, down-milling =3 (see milling mode diagram)

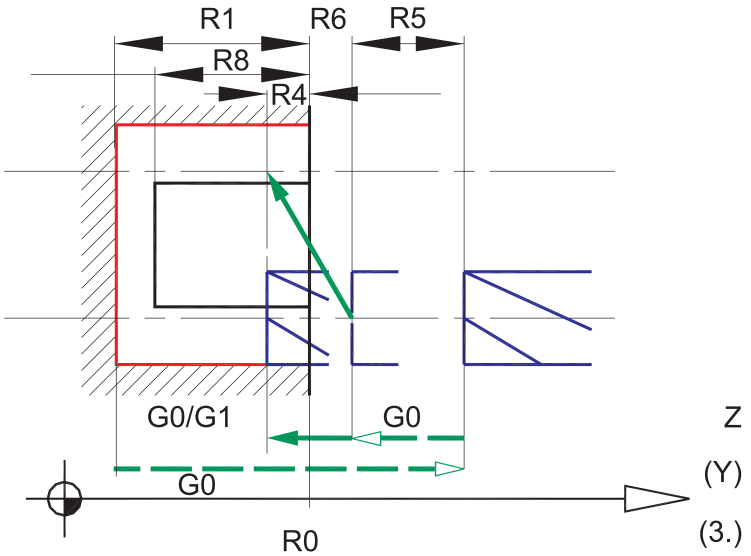
R70 Feed rate at pocket corners in % of machining feedrate

R71 Plunge feed in % of machining feed

R72 Maximum cut width in % of tool diameter



8.2.2 - 34 Milling mode cycle diagram L1055



8.2.2 - 35 Feed plane cycle diagram L1055

Feed plane input parameters

R0 Surface area
R1 Final depth
R4 Depth growth
R5 Retraction
R6 Safety distance
R8 Overall depth

Sequence description

The tool moves in rapid traverse to the plunge position and then in rapid traverse up to the safety clearance from the surface.

The plunge position is specified by length R49. The depth is reached on the R49 1.5 times path.

If allowance R48 provides sufficient space for the milling tool, it approaches the depth in rapid traverse. When raw depth R8 is reached, it then plunges in advanced feed.

The pocket is milled in the steps specified by R4 until the final depth R1 has been reached. If the pocket is to be milled in one feed,R4=R1 is specified.

Broaching takes place taking into account the allowance R48, up-milling / down-milling as well as the specified maximum cut width with equally wide milling paths.

If the pocket is milled from the solid, the allowance R48 is specified as greater than or equal to half of the width R46. If only finishing is required, a milling path for finishing the pocket contour results automatically from the small allowance of finishing cut.

Feed into the corners of the pocket takes place with reduced feed rate ((R70)) because of the enlarged wrap angle at the milling tool. If no reduction is preferred, R70=100 is specified.

When the final depth is reached, positioning takes place to the centre of the slot and the withdrawn position.

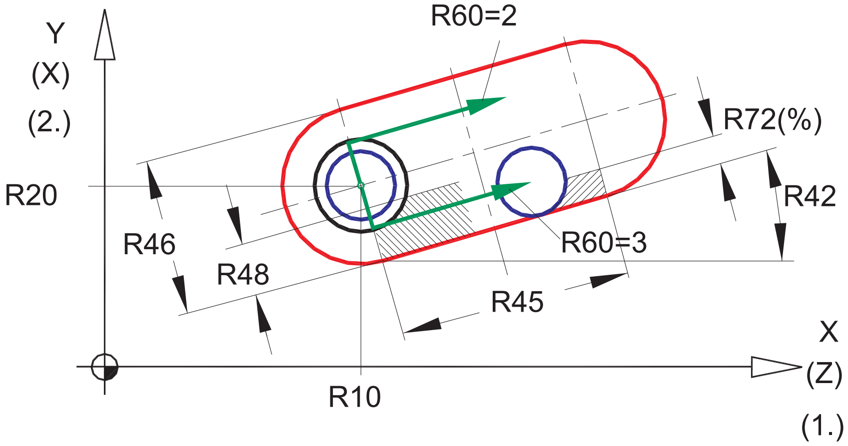
i

L1055 can also be used to mill a circular pocket, both the length and the width being specified as equal to twice the radius of the corner.

Application example

```
N100 R10=100 R20=200 R42=30 R45=100 R46=80 R47=20
N110 R48=40 R49=50 R60=3 R70=50 R71=50 R72=75
N120 R0=200 R1=30 R4=10 R5=50 R6=2 R8=0
N130 L1055
```

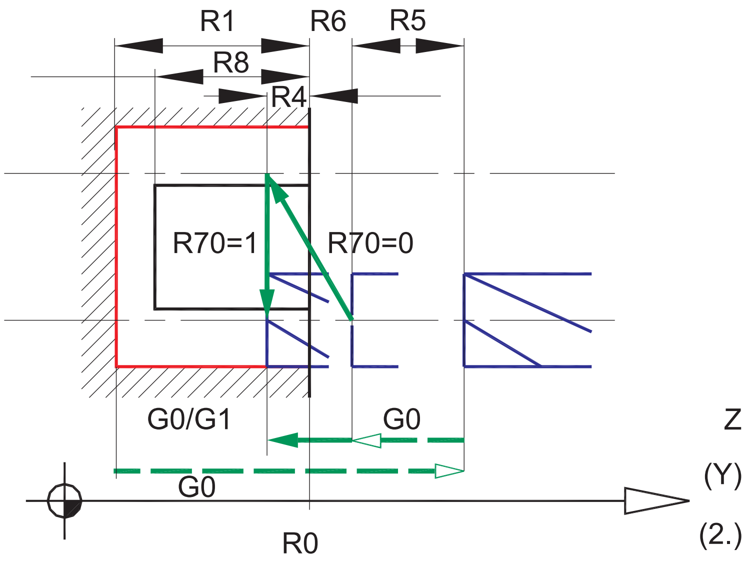
L1057 Milling grooves with plunge milling



8.2.2 - 36 Machining plane cycle diagram L1057

Input parameters for machining plane

R10 Centre of groove radius in X centre pocket
R20 Centre of groove radius in Y centre pocket
R42 Inclination angle of groove
R45 Groove length
R46 Groove width
R48 Allowance, (from the solid R48 >= 1/2 width)
R60 Up-milling =2, down-milling =3
R70 Plunge, =0 without return traverse to depth
R70 Plunge, =1 with return traverse to depth
R71 Plunge feed in % of machining feed
R72 Maximum cut width in % of tool diameter



8.2.2 - 37 Feed plane cycle diagram L1057

Feed plane input parameters

R0 Surface area
R1 Final depth
R4 Depth growth
R5 Retraction
R6 Safety distance
R8 Overall depth

Sequence description

The tool moves in rapid traverse to the plunge position and then in rapid traverse up to the safety clearance from the surface.

The plunge position is specified by length R45. The depth is reached directly with R70=0. With R70=1, the tool also retracts to the start point at the feed depth so that the entire depth is milled.

If allowance R48 provides sufficient space for the milling tool, it approaches the depth in rapid traverse. When raw depth R8 is reached, it then plunges in advanced feed.

The groove is milled in the steps specified by R4 until the final depth R1 has been reached. If the groove is to be milled in one feed, R4=R1 is specified.

Broaching takes place taking into account the allowance , up cut/ down cut as well as the specified maximum cut width with equally wide milling paths.

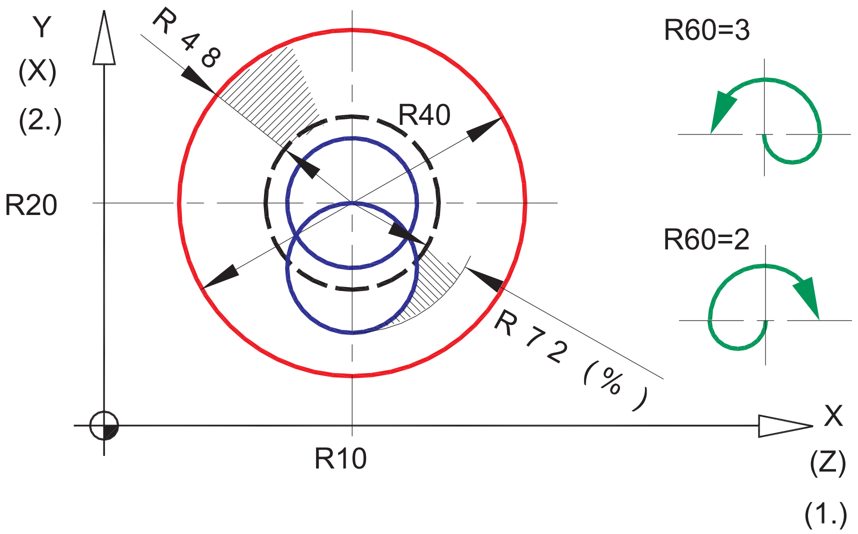
If the groove is milled from the solid, the allowance R48 is specified as greater than or equal to (half of) the width R46. If only finishing is required, a milling path for finishing the slot contour results automatically from the small allowance of finishing cut.

The tool retracts in Z when the final depth is reached.

Application example

```
N100 R10=150 R20=250 R42=30 R45=100 R46=20 R48=20
N110 R60=2 R70=1 R71=80 R72=80 R0=0 R1=30 R4=10
N120 R5=100 R6=2 R8=28
N130 L1057
```

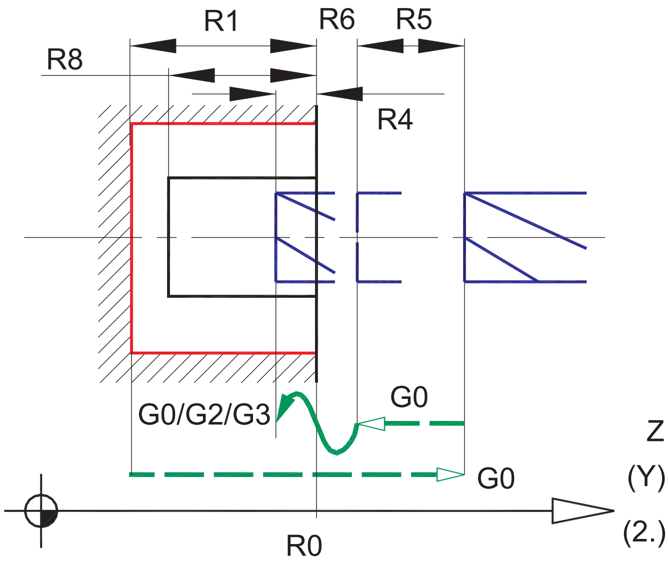
L1058 Spiral form broaching of circular pocket with plunge miller



8.2.2 - 38 Machining plane cycle diagram L1058

Input parameters for machining plane

R10 Centre of circular pocket in X
R20 Centre of circular pocket in Y
R40 Diameter of circular area
R48 Radial allowance
R60 Up-milling =2, down-milling =3
R70 Plunge feed in % of machining feed
R72 Width of cut in % of milling tool diameter



8.2.2 - 39 Feed plane cycle diagram L1058

Feed plane input parameters

R0 Surface area
R1 Final depth
R4 Depth growth
R5 Retraction
R6 Safety distance
R8 Overall depth

Sequence description

The start positions R10 and R20 are approached in the XY plane. Rapid traverse then takes place to the safety clearance.

The plunging movement is carried out with a full circle in X, Y and Z at the plunging feed calculated from R70. The run-in radius is calculated from the cut width R72. At the milling depth, a further full circle is travelled to remove the residual material.

However, if the allowance of the circular pocket R48 is so small that the milling tool can plunge without cutting, plunging takes place in Z in the centre of the pocket. When raw depth R8 is reached, it then plunges in advanced feed.

Broaching takes place taking into account the amount of cut, up cut/ down cut and the specified maximum cut width with equally wide spirally increasing milling paths. When the circular diameter has been reached with the spirals, a full circle is added for finish machining.

If the surface is milled from the solid, the allowance R48 is specified as greater than or equal to half of the diameter R40. If only finishing is required, a milling path for finishing the pocket contour results automatically from the small allowance of finishing cut.

Movement takes place to the withdrawn position when the intermediate depth is reached, and the next depth is processed.

The groove is milled in the steps specified by R4. If the pocket is to be milled in one feed,R4=R1 is specified.

When the final depth is reached, positioning takes place to the centre of the slot and the withdrawn position.

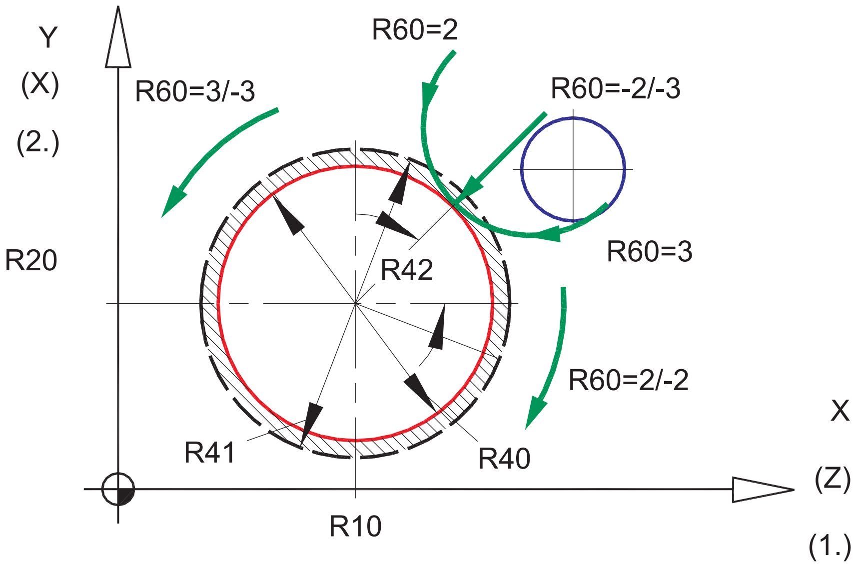
Application example

```
N100 R10=100 R20=150 R40=60 R48=30 R60=2 R70=80
N110 R72=75 R0=200 R1=30 R4=15 R5=100 R6=2 R8=0
N120 L1058
```

HELLER

8.2.2.3 L11xx Milling external contours

L1101 Milling journals



8.2.2 - 40 Machining plane cycle diagram L1101

Input parameters for machining plane

R10 Centre point (G17 X, G18 Z)

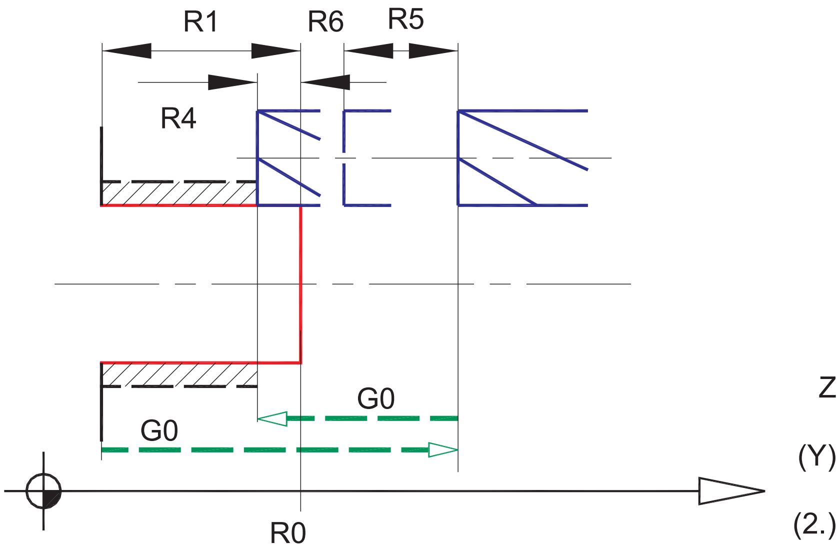
R20 Centre point (G17 Y, G18 X)

R40 Diameter

R41 Unfinished diameter

R42 Contour entry angular position

R60 Milling mode, clockwise = 2, anticlockwise = 3, straight move in = -2 / -3



8.2.2 - 41 Feed plane cycle diagram L1101

Feed plane input parameters

R0 Surface area
R1 Final depth
R4 Depth growth
R5 Retraction
R6 Safety distance

Sequence description

Traverse first takes place to the start position on the entry quadrant, the start position being determined from the unfinished diameter, safety clearance and angular position of contour entry.

If the journal is machined in a single cut, R4=R1 is specified, otherwise machining should be in the steps specified by R4 up to the final depth R1. When the final depth R1 is reached, the tool exits in an exit quadrant and the boring axis withdraws.

R60 is used to select up-milling or down-milling (clockwise tool). If a straight approach to the contour of the hole is preferred to entry and exit quadrants to save time, this is achieved by specifying a negative R60. Approach to the unfinished diameter takes into account the safety distance R6.

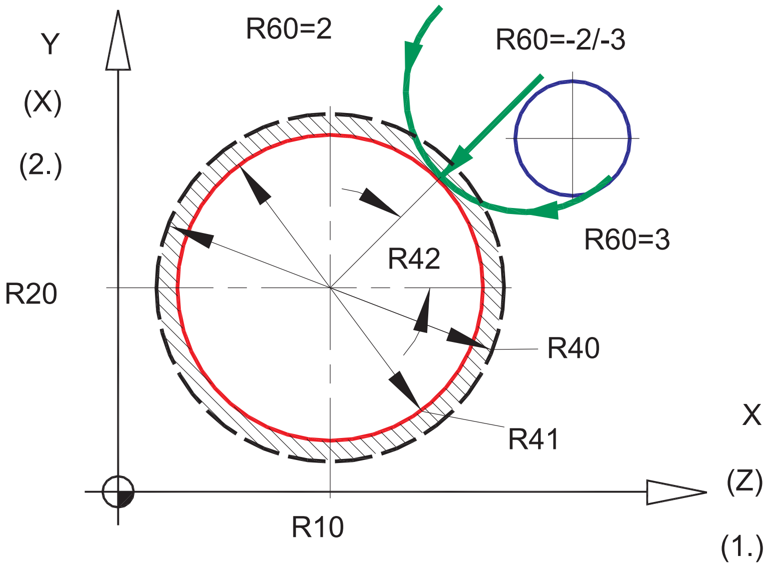
To prevent marking of flat surfaces, the drilling axis is moved a few 1/100 mm when entering and extending.

HELLER

Application example

```
N100 R10=200 R20=300 R40=100 R41=120 R42=45 R60=2
N110 R0=200 R1=40 R4=10 R5=50 R6=1
N120 L1101
```

L1102 Milling external threads



8.2.2 - 42 Machining plane cycle diagram L1102

Input parameters for machining plane

R10 Centre point (G17 X, G18 Z)

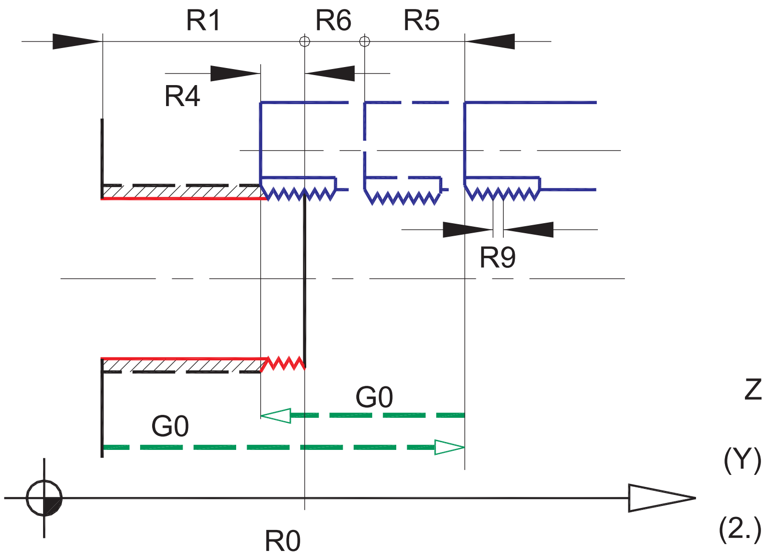
R20 Centre point (G17 Y, G18 X)

R40 Nominal diameter

R41 Core diameter

R42 Contour entry angular position

R60 Milling mode, clockwise = 2, anticlockwise = 3, straight move in = - 2,-3



8.2.2 - 43 Feed plane cycle diagram L1102

Feed plane input parameters

R0	Surface area
R1	Final depth
R4	Depth growth
R5	Retraction
R6	Safety distance
R9	Pitch, r.h. thread + and l.h. thread -

Sequence description

Traverse first takes place to the start position on the entry quadrant, the start position being determined from the unfinished diameter, safety clearance and angular position of contour entry. The boring axis then moves at rapid traverse to the required depth.

Milling of the thread usually takes place with entry and exit quadrants that are determined automatically from the core diameter. Here, to prevent back cutting, the proportionate thread pitch is taken into account.

If the thread is to be cut using a single-tooth milling tool (whirling), the depth growth R4 is specified equal to the pitch R9 and milling then takes place in a spiral form until the final depth is reached.

If the thread is to be machined in a single cut, R4=R1 is specified. Otherwise, machining takes place to the final depth R1 with the steps specified in R4.

It must be noted that the depth growth is always rounded to a multiple of the thread pitch.

If R4=R1 is programmed, the thread is always milled in a single pass.

Up-milling or down-milling is selected using R60 (clockwise tool). If straight approach to the drilling contour is required instead of entry/exit quarter circles to save time, this is done by specifying negative values of R60.

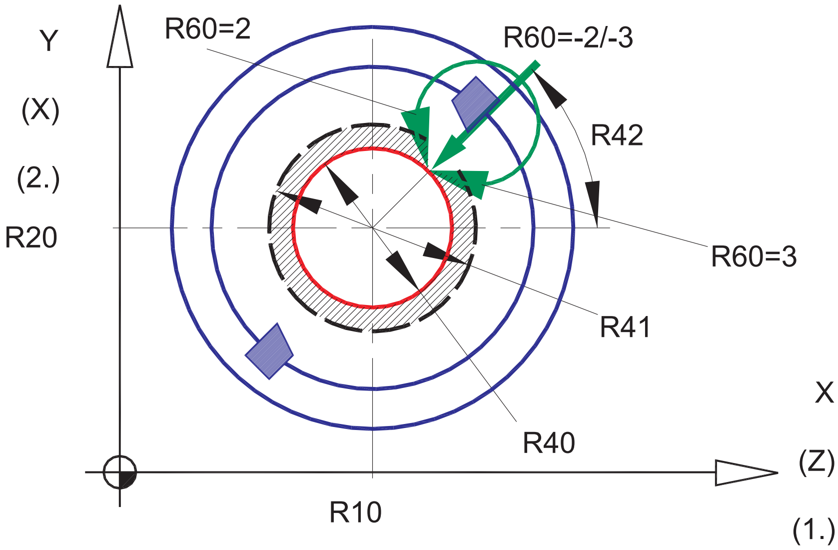
A r.h. thread is programmed with positive pitch, a l.h. thread with negative pitch R9.

Care must be taken that the depth of any end face that may be present is sufficient for the proportionate pitch of the exit quadrant. It has a maximum 1/8* pitch.

Application example

```
N100 R10=100 R20=200 R40=50 R41=48.5 R42=45 R60=3
N110 R0=100 R1=30 R4=15 R5=50 R6=2 R9=1.5
N120 L1102
```

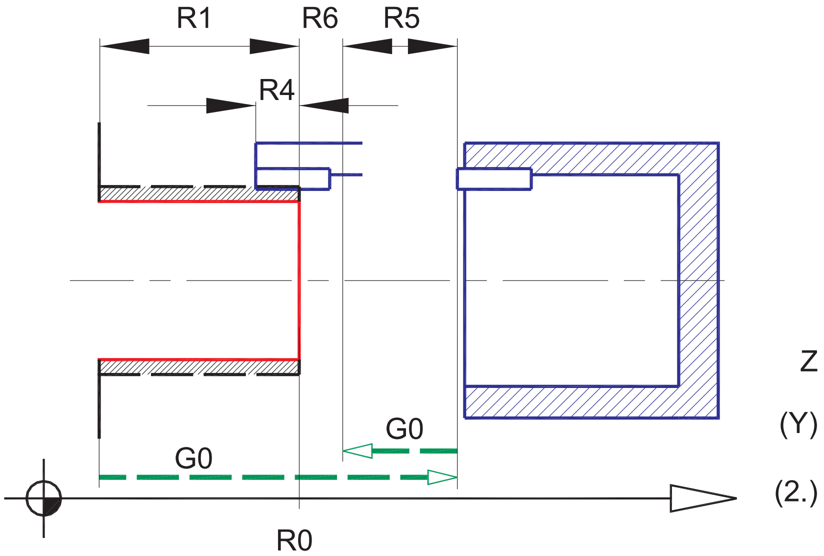
L1103 Milling journals with bell tool



8.2.2 - 44 Machining plane cycle diagram L1103

Input parameters for machining plane

R10 Centre point (G17 X, G18 Z)
R20 Centre point (G17 Y, G18 X)
R40 Spigot diameter
R41 Unfinished diameter
R42 Contour entry angular position
R60 Up-milling = 2, down-milling = 3, straight move in = -2, -3



8.2.2 - 45 Feed plane cycle diagram L1103

Feed plane input parameters

R0 Surface area
R1 Final depth
R4 Depth growth
R5 Retraction
R6 Safety distance
R71 Plunge feed in % of machining feed

Sequence description

The centre point is first approached at rapid traverse and positioning then takes place to the safety clearance.

If the unfinished diameter is greater than the tool diameter with safety distance, positioning takes place at reduced feed speed (R71), otherwise at rapid traverse.

If the journal is machined in a single cut, R4=R1 is specified, otherwise machining should be in the steps specified by R4 up to the final depth R1. When the final depth R1 is reached, the tool moves out in an exit semicircle and the boring axis withdraws.

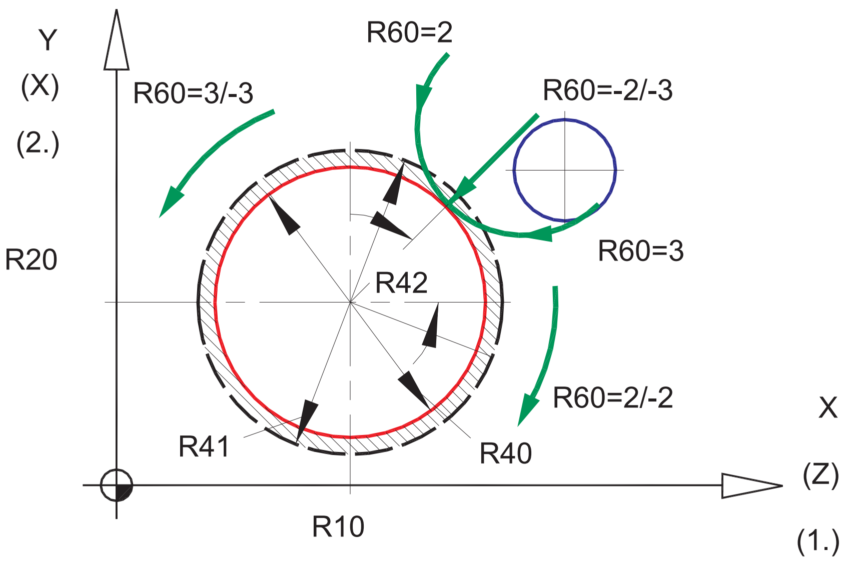
Entry and extending takes place in semicircles.

Up-milling or down-milling is selected using R60 (clockwise tool). If straight approach to the journal contour is required instead of entry/exit semicircles to save time, this is done by specifying negative values of R60. Approach to the unfinished diameter takes into account the safety distance R6.

Application example

```
N090 R10=100 R20=200 R40=50 R41=45 R42=45 R60=2 R0=200 R1=40
N100 L1103
```

L1104 Milling journals, spiral form



8.2.2 - 46 Machining plane cycle diagram L1104

Input parameters for machining plane

R10 Centre point (G17 X, G18 Z)

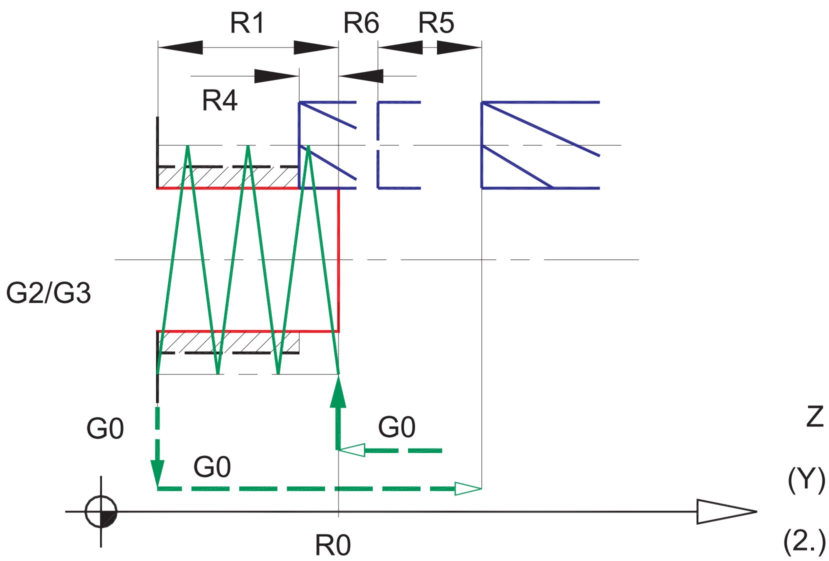
R20 Centre point (G17 Y, G18 X)

R40 Diameter

R41 Unfinished diameter

R42 Contour entry angular position

R60 Milling mode, clockwise = 2, anticlockwise = 3, straight move in = -2 / -3



8.2.2 - 47 Feed plane cycle diagram L1104

Feed plane input parameters

R0	Surface area
R1	Final depth
R4	Depth growth
R5	Retraction
R6	Safety distance

Sequence description

Traverse first takes place to the start position on the entry quadrant, the start position being determined from the unfinished diameter, safety clearance and angular position of contour entry.

If the journal is machined in a single cut, R4=R1 is specified, otherwise machining should be in the steps specified by R4 up to the final depth R1. When the final depth R1 is reached, the tool exits in an exit quadrant and the boring axis withdraws.

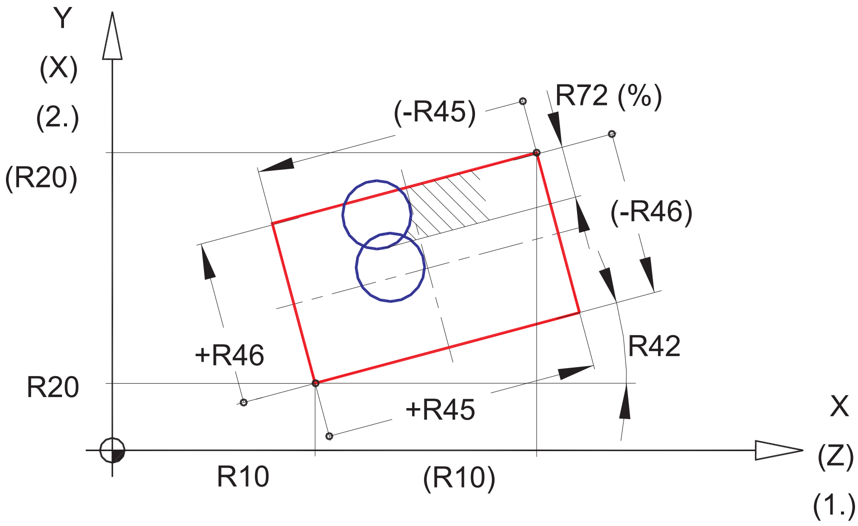
R60 is used to select up-milling or down-milling (clockwise tool). If a straight approach to the contour of the hole is preferred to entry and exit quadrants to save time, this is achieved by specifying a negative R60. Approach takes into account the safety clearance R6 to the unfinished diameter.

Application example

```
N090 R10=200 R20=300 R40=100 R41=120 R42=45 R60=2 R0=200 R1=4
N100 L1104
```

8.2.2.4 L13xx Surface milling

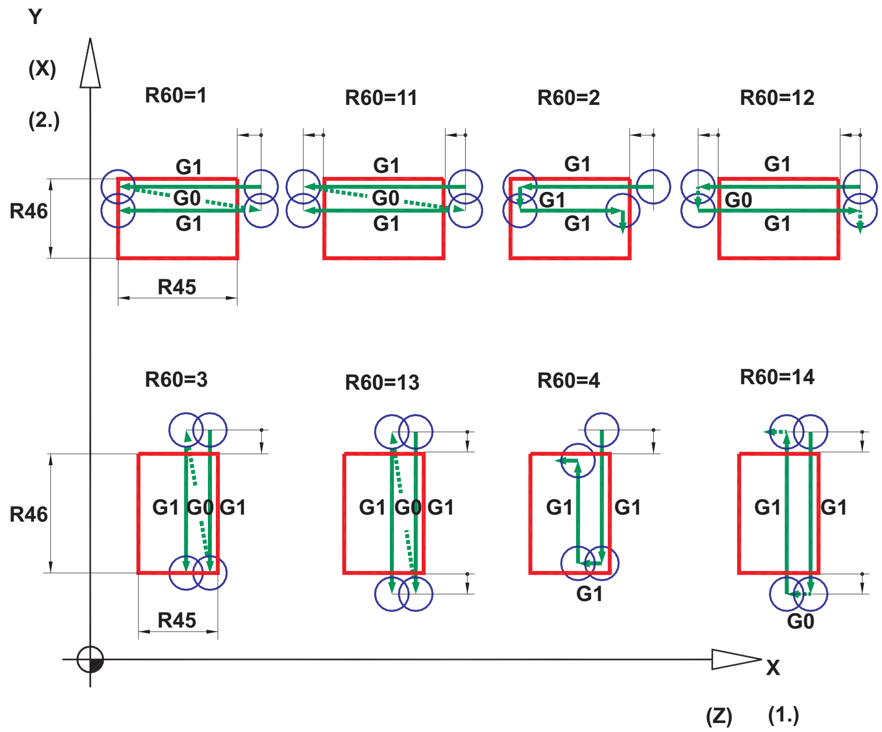
L1301 Milling rectangular surfaces



8.2.2 - 48 Machining plane cycle diagram L1301

Input parameters for machining plane

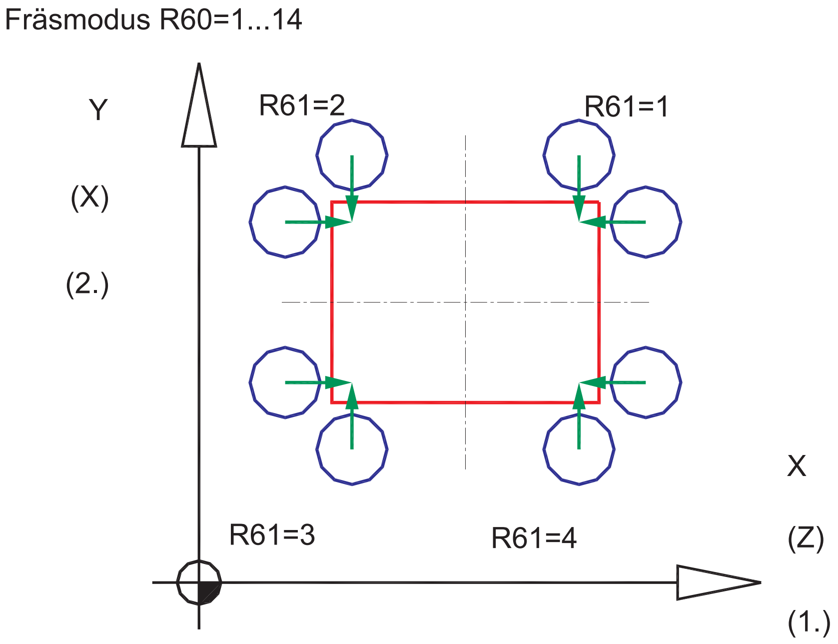
R10 Position of a corner (G17 X, G18 Z)
R20 Position of a corner (G17 Y, G18 X)
R42 Inclination angle of surface
R45 Length of surface
R46 Width of surface
R72 Width of cut in % of milling tool diameter



8.2.2 - 49 Milling mode cycle diagram L1301

Milling mode - input parameters

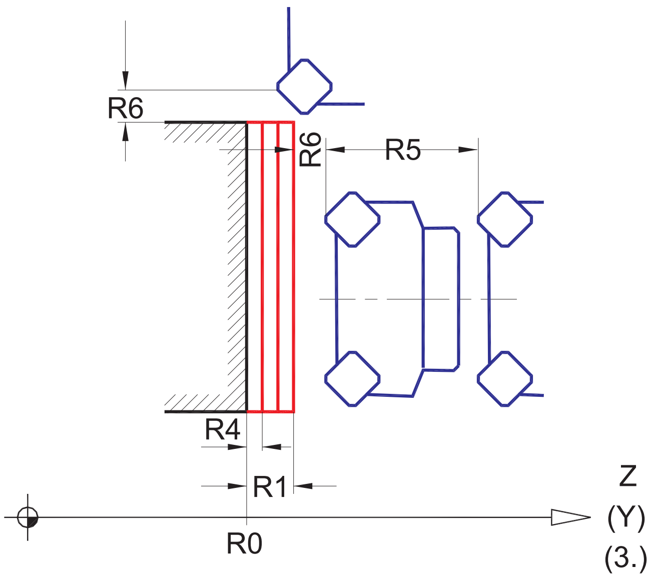
R60 Milling mode according to graphic 1, 2, 3, 4, 11, 12, 13, 14



8.2.2 - 50 Starting point - cycle diagram L1301

Starting point input parameters

R61 Starting point of machining according to graphic 1, 2, 3, 4 (quadrants)



8.2.2 - 51 Feed plane cycle diagram L1301

Feed plane input parameters

R0 Surface, finished size
R1 Allowance
R4 Depth growth
R5 Retraction
R6 Safety distance

Sequence description

Movement first takes place to the starting position determined by the choice of milling mode and starting point.

Possible milling modes are:
- Parallel to length with or without lifting off, with or without run off.
- Parallel to width with or without lifting off, with or without run off.
- Parallel to length, meander-shaped
- Parallel to width, meander-shaped

The division of the milling paths is done with equal path widths. The last path has a sideways overlap amounting to the safety distanceR6.

If the surface is to be machined in one feed, R4=R1 is specified, otherwise machining is executed in the steps specified by R4 up to the final depth R1 and machining proceeds according to the selected milling mode. Once the final depth R1 is reached, the boring axis is moved to the withdrawn position.

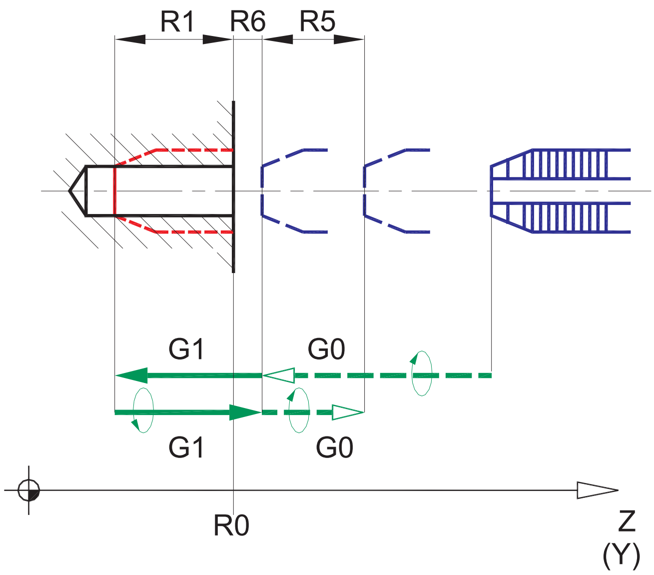
Application examples

```
N100 R10=100 R20=300 R42=30 R45=200 R46=150
N110 R72=80 R60=3 R61=2
N120 R0=-50 R1=20 R4=5 R5=100 R6=2
N130 L1301
```

8.2.3 L0xxx Sub-programs for tapping

8.2.3.1 L007x Standard tapping cycles

L70 Right hand or left hand tapping



8.2.3 - 1 Cycle diagram L70

Input parameters

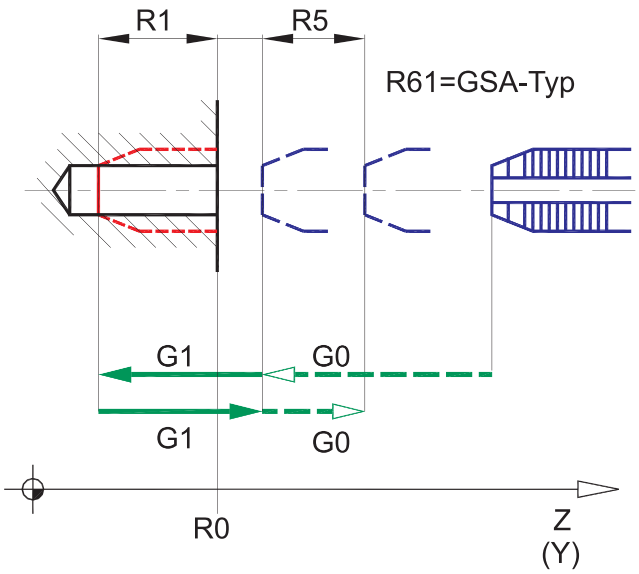
R0 Surface area
R1 Thread depth
R5 Retraction
R6 Safety distance
R60 Direction of rotation, right=3, left=4

Application examples

```
N100 X100 Y100
N110 R0=0 R1=30 R5=50 R6=5 R60=3
N120 L70
N130 X200
N140 R5=100
N150 L70
```

HELLER

L71 Tapping with GSA reversing chuck



8.2.3 - 2 Cycle diagram L71

Input parameters

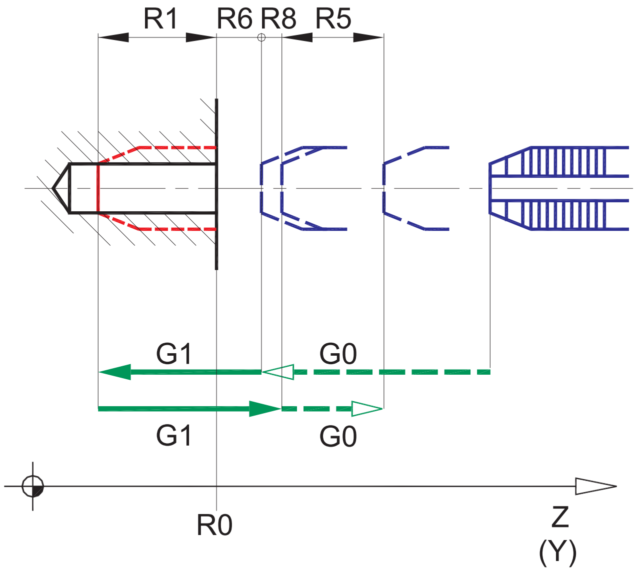
R0 Surface area
R1 Thread depth
R5 Retraction
R61 GSA-type 8/ 12/ 20 (Benz)

The tapping chuck used here automatically reverses the direction of the tap when the bottom of the thread is reached. As an example, L71 is described in accordance with the manufacturer's details for the Benz type GSA 8/ 12/ 20 chucks. L71 must be modified if necessary for chucks from other manufacturers. This is particularly important for the values for the extraction path, for which the rotation direction reverses, and for the necessary safety clearance for leaving the thread.

Application examples

```
N100 X100 Y100
N110 R0=0 R1=30 R5=50 R61=8
N120 L71
```

L72 Tapping with self-release chuck



8.2.3 - 3 Cycle diagram L72

Input parameters

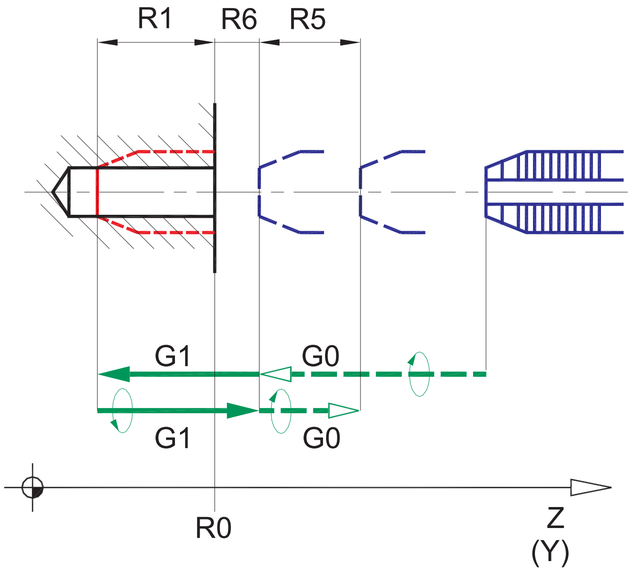
R0 Surface area
R1 Thread depth
R2 Dwell time in seconds [s]
R5 Retraction
R6 Safety distance
R8 Chuck self-release distance

The tapping chuck used here releases the drive at the bottom of the thread. This permits relatively exact depths to be reached.

Application examples

```
R0=0 R1=30 R2=1 R5=50 R6=2 R8=2
N100 MCALL L72
N110 X100 Y100
N120 X200
N130 MCALL
```

L78 tapping with GNCN+GNCK reversing chuck



8.2.3 - 4 Cycle diagram L78

Input parameters

R0 Surface area

R1 Thread depth

R5 Retraction

R6 Safety distance

R60 Direction of rotation, right=3, left=4

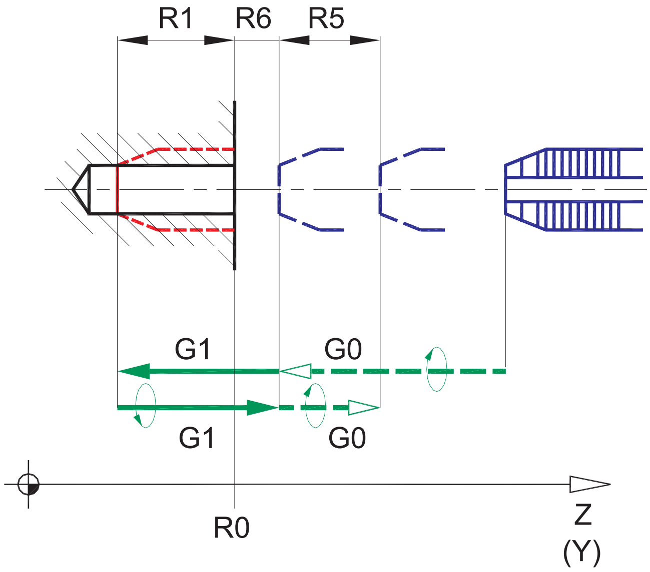
i

The spindle does not change its directional direction, as such change is effected by the thread cutter. Right or left threads can be cut.

Application example

```
N100 X100 Y100
N110 R0=0 R1=30 R5=50 R6=5 R60=3
N120 L78
N130 X200
N140 R5=100
N150 L78
```

L79 Tapping without compensating chuck, direct



8.2.3 - 5 Cycle diagram L79

Input parameters

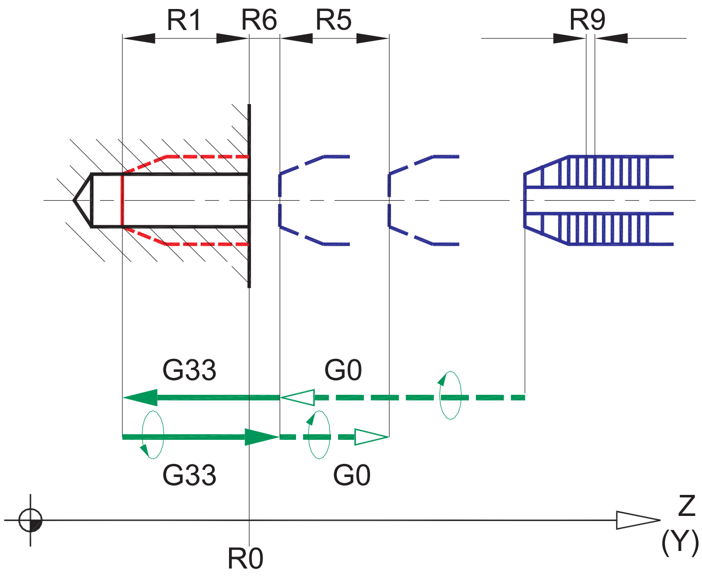
R0 Surface area
R1 Thread depth
R5 Retraction
R6 Safety distance
R60 Direction of rotation, right=3, left=4

Starting takes place with spindle stationary to synchronise the spindle and feed. It is therefore advisable not to switch speed M3/ M4 after tool change.

Application example

```
N100 X100 Y100 M5
N110 R0=0 R1=30 R5=50 R6=5 R60=3
N120 L79
N130 X200
N140 R5=100
N150 L79
```

L170 Tapping with compensating chuck, G33



8.2.3 - 6 Cycle diagram L170

Input parameters

R0 Absolute surface
R1 Relative thread depth
R5 Retraction
R6 Safety distance
R9 Threaded pitch, right=+, left=-
R70 Speed factor for return traverse

i

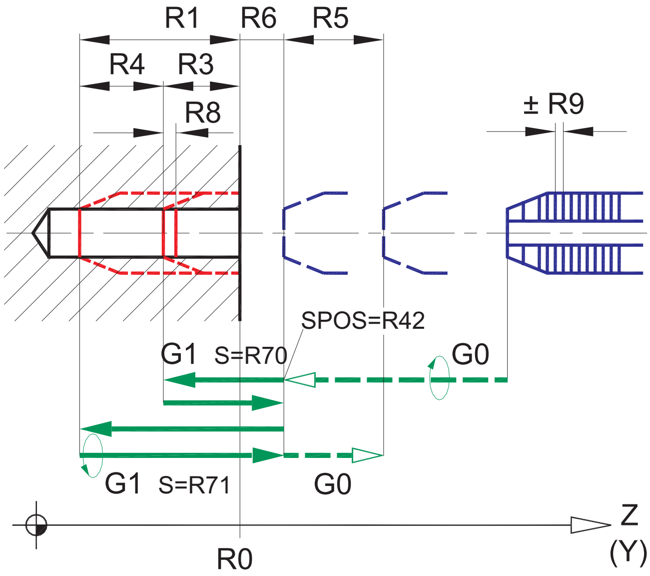
The Z-axis is coupled to the rotating spindle by G33 ("floating connection"). G33 possible only after version 6.2.

R70 can be used to accelerate the retraction.

Application example

```
N100 X100 Y100 S800 M3
N110 R0=0 R1=30 R5=50 R6=3 R9=1.5 R70=2
N120 L170
N130 X200
N140 L170
```

L179 Deep-hole tapping without compensating chuck



8.2.3 - 7 Cycle diagram L179

Input parameters

R0 Absolute surface
R1 Relative thread depth
R3 1. Thread depth
R4 Depth growth
R5 Retraction
R6 Safety distance
R8 Chip breaker stroke
R9 Threaded pitch, right=+, left=-
R40 Spindle alignment position
R70 Speed fortapping
R71 Speed for return traverse

This tapping cycle can be used for gun tapping with repeated chip removal.

R8 programs the chip breaker stroke.

If R8=0, no chip removal.

If R8>0 and R8<R1, chip removal occurs to these values.

If R8>=R1, chip removal is always to the safety level.

If R3=R1 is programmed, the thread depth is created all at once.

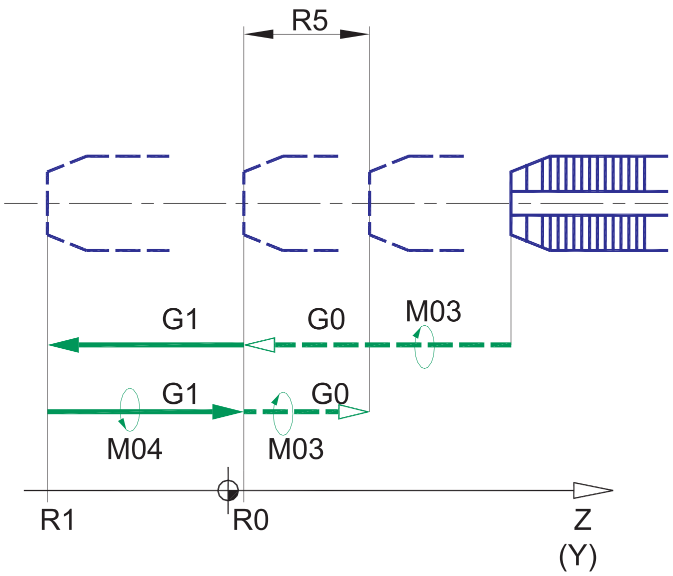
The speed for retraction R71 enables much faster retraction than for tapping. If too high a speed is programmed, the maximum speed is used in line with the selected gear step.

Application example

```
N100 X100 Y100
N110 R0=100 R1=30 R3=10 R4=10 R5=0 R6=3 R8=1 R9=1.25 R40=0 R7
N120 L179
N130 X200
N140 L179
```

8.2.3.2 L008x Tapping cycles according to DIN 66025

L84 Right hand tapping



8.2.3 - 8 Cycle diagram L84

Input parameters

R0 Absolute surface

R1 Thread depth

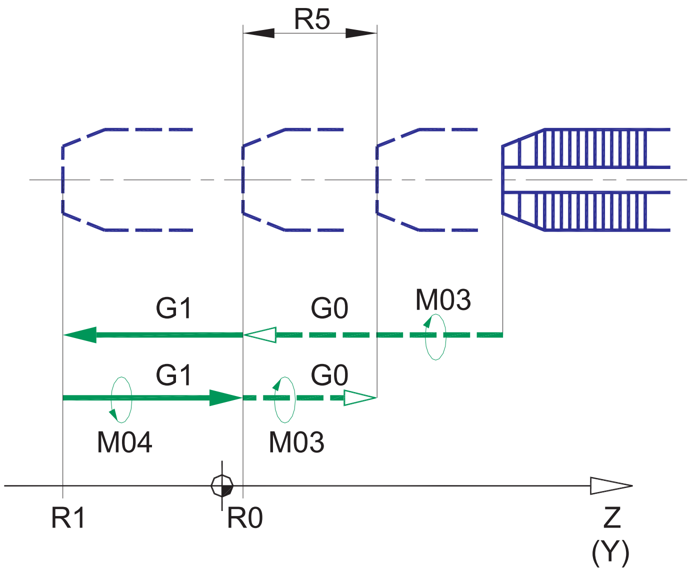
R5 Retraction, incremental dimension

Application example

without cycle support

```
N100 X100 Y100
N110 R0=0 R1=-40 R5=50
N120 L84
N130 X200
N140 R5=0
N150 L84
```

L89 Tapping without compensating chuck, direct



8.2.3 - 9 Cycle diagram L89

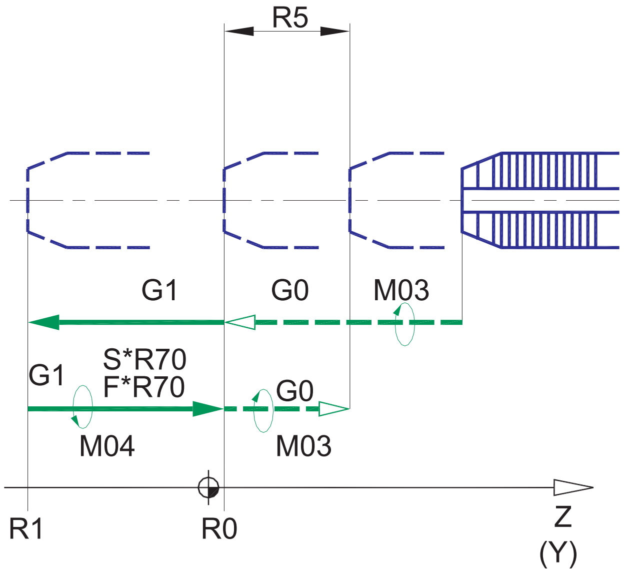
Input parameters

R0 Reference plane
R1 Thread depth
R5 Retraction, incremental dimension

Application example

```
N100 X100 Y100
N110 R0=0 R1=-40 R5=50
N120 L89
N130 X200
N140 R5=0
N150 L89
```

L184 Tapping, right hand rotation, faster retraction



8.2.3 - 10 Cycle diagram L184

Input parameters

R0 Absolute reference plane
R1 Absolute thread depth
R5 Retraction, incremental dimension
R70 Factor for retraction

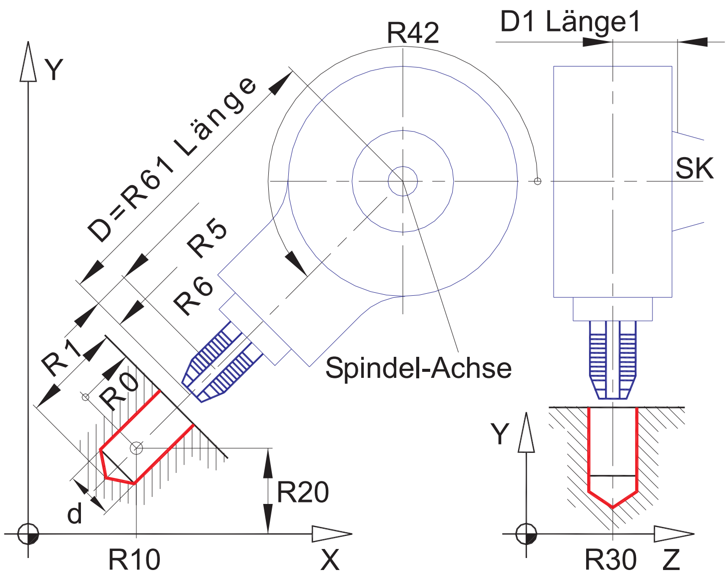
The factor R70 for speed and feed enables a much faster retraction to be programmed. If too high a speed is programmed, the max. speed is used in line with the selected gear step.

Application example

```
N100 X100 Y100 S900 F900 M3
N110 R0=200 R1=180 R5=0 R70=2
N110 L184
N120 X200
N130 L184
```

8.2.3.3 L03xx Other tapping cycles

L301 Tapping in XY plane using angular drilling head



8.2.3 - 11 Cycle diagram L301

Input parameters

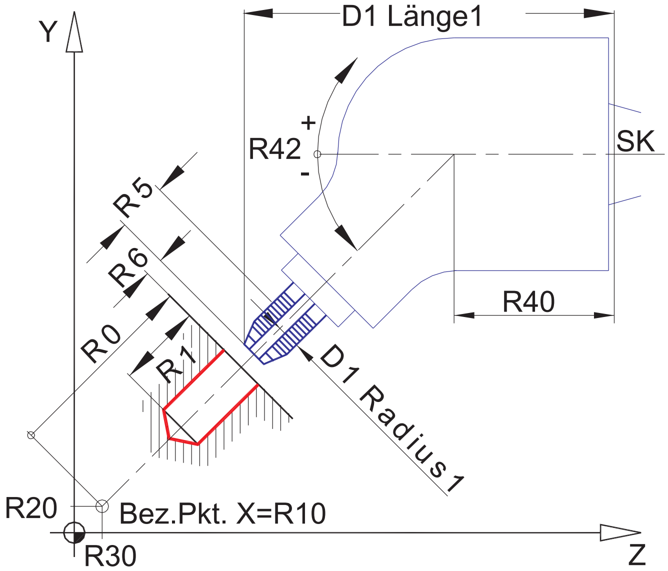
R0 Reference plane with respect to reference point
R1 Thread depth
R5 Retraction
R6 Safety distance
R10 X coordinate
R20 Y coordinate
R30 Z coordinate
R42 Drilling angle to +X
R61 Cutting edge 1...9
R63 Direction of rotation, right=3, left=4

i The thread is tapped using a compensating chuck.

Application example

```
N100 X100 Y200 Z400 D1
N110 R0=100 R1=20 R5=120 R6=2 R10=200 R20=300
N120 R30=150 R42=135 R61=2 R63=3
N130 L301
```

L302 Tapping in YZ plane using angular drilling head, direct



8.2.3 - 12 Cycle diagram L302

Input parameters

R0 Reference plane with respect to reference point
R1 Thread depth
R5 Retraction
R6 Safety distance
R10 X coordinate
R20 Y coordinate
R30 Z coordinate
R40 Length SK50 to tool axis intersection
R42 YZ angle of inclined tool axis
R60 Direction of rotation, right=3, left=4

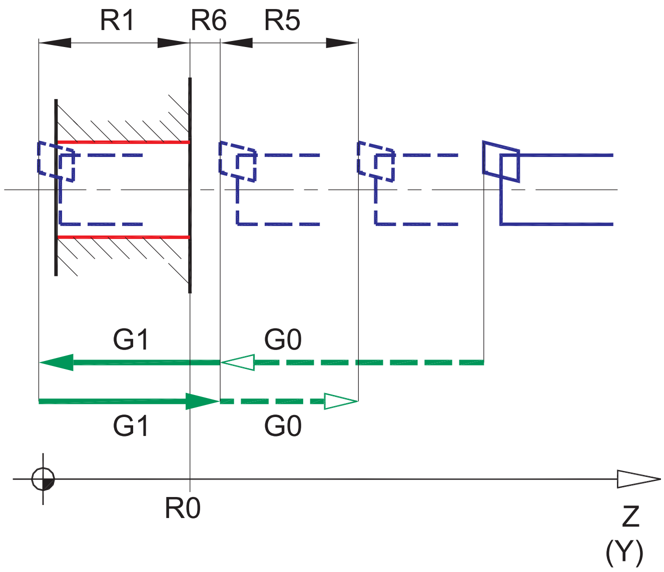
Application example

```
N100 X100 Y200 Z400 D1
N110 R0=100 R1=20 R5=120 R6=2 R10=200 R20=300
N120 R30=150 R40=115 R42=-45 R60=3
N130 L302
```

8.2.4 L0xxx Reverse boring sub-programs

8.2.4.1 L007x Standard reverse boring cycles

L73 Reverse boring 1, withdrawal with feed



8.2.4 - 1 Cycle diagram L73

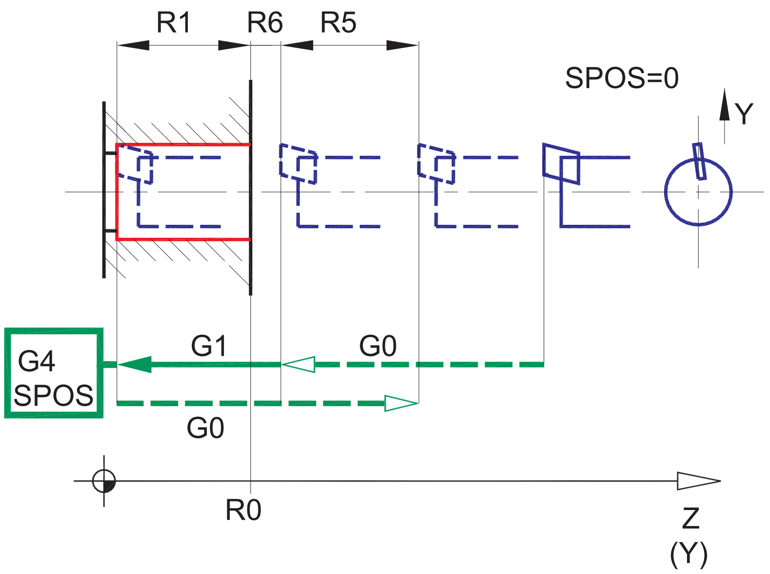
Input parameters

R0 Reference plane with respect to reference point
R1 Rotation depth
R5 Retraction
R6 Safety distance

Application example

```
N100 X100 Y100
N110 R0=0 R1=30 R5=50 R6=5
N120 L73
N130 X200
N140 R5=100
N150 L73
```

L74 Reverse boring 2, G4 and SPOS



8.2.4 - 2 Cycle diagram L74

Input parameters

R0 Reference plane with respect to reference point

R1 Rotation depth

R2 Dwell time in seconds [s]

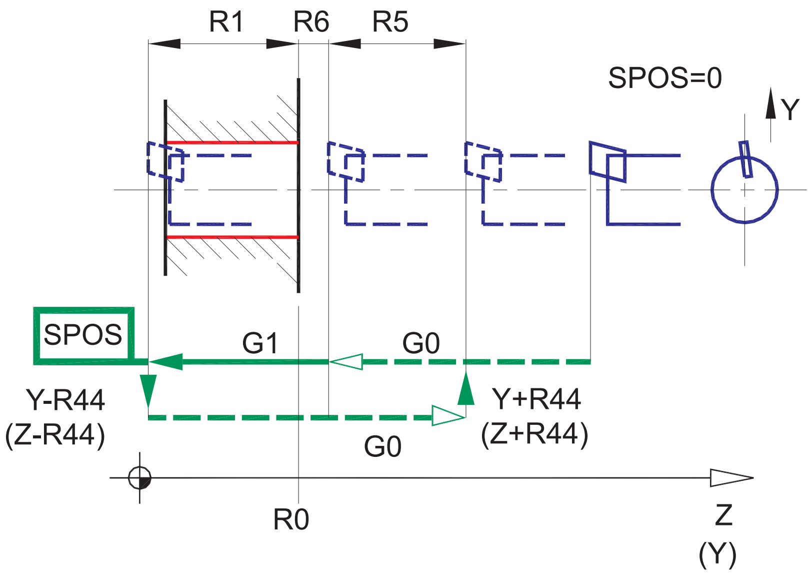
R5 Retraction

R6 Safety distance

Application example

```
N100 X100 Y100
N110 R0=0 R1=30 R2=1 R5=50 R6=5
N120 L74
N130 X200
N140 L74
```

L75 Reverse boring 3, eccentric extend



8.2.4 - 3 Cycle diagram L75

Input parameters

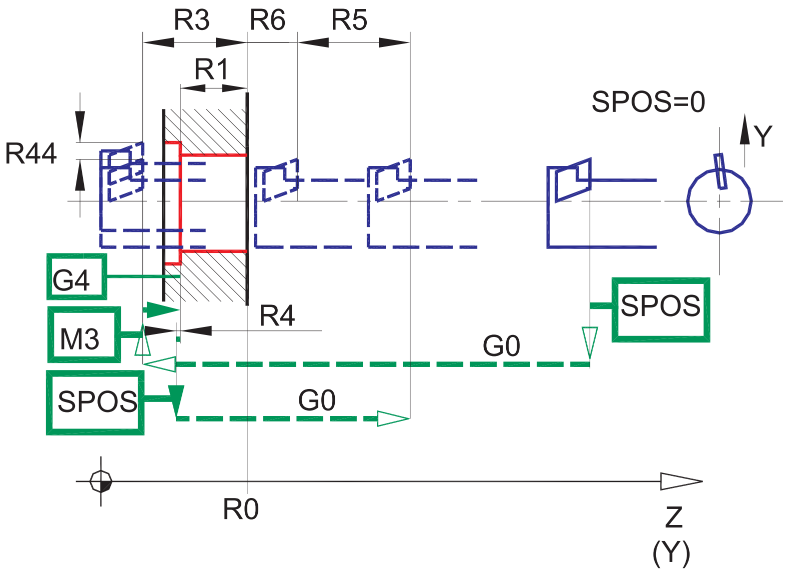
R0 Reference plane with respect to reference point
R1 Rotation depth
R5 Retraction
R6 Safety distance
R42 Spindle alignment position SPOS=...
R44 Lifting dimension

The axis to be lifted, the spindle alignment position, the cutting edge position in the direction of the spindle alignment position and the sign of the lifting dimension must all be carefully observed.
The cutting edge should usually point towards SPOS=0 in the XY plane with +Y. Lifting then takes place with a positive lifting dimension R44 and correct spindle alignment position R42=0 (Y down). Lifting off in the XY plane always takes place in Y. If the tip points in a different direction it can be corrected using R42.

Application example

```
N100 X100 Y100
N110 R0=0 R1=30 R5=50 R6=5 R42=0.2 R44=0
N120 L75
```

L76 Backwards reverse boring



8.2.4 - 4 Cycle diagram L76

Input parameters

R0 Surface area
R1 Rotation depth
R2 Dwell time in seconds [s]
R3 Entry depth
R4 Axial lifting dimension

R5 Retraction
R6 Safety distance
R42 Spindle alignment position SPOS=...
R44 Eccentric lifting dimension

The axis to be lifted, the spindle alignment position, the cutting edge position in the direction of the spindle alignment position and the sign of the lifting dimension must all be carefully observed. The cutting edge should usually point towards SPOS=0 in the XY plane with +Y. Lifting then takes place with a positive lifting dimension R44 and correct spindle alignment position R42=0 (Y down). Lifting in the rotated XY plane is always in Y. If the cutting edge is pointed in another direction, this can be corrected by R42.

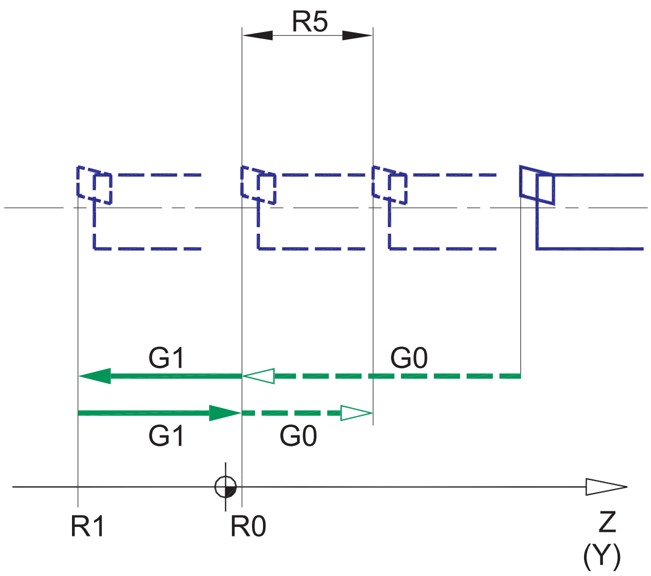
The length of the backwards cutting edge must be activated before calling up L76 (with a tool modification D2 if necessary).

Application example

```
N100 X100 Y100 D2 D2;tool cutting edge;
R110 R0=0 R1=30 R2=0.1 R3=45 R4=0.2 R5=50 R6=5 R42=5 R44=0
N120 L76
```

8.2.4.2 L008x Reverse boring cycles according to DIN 66025

L85 Reverse boring 1



8.2.4 - 5 Cycle diagram L85

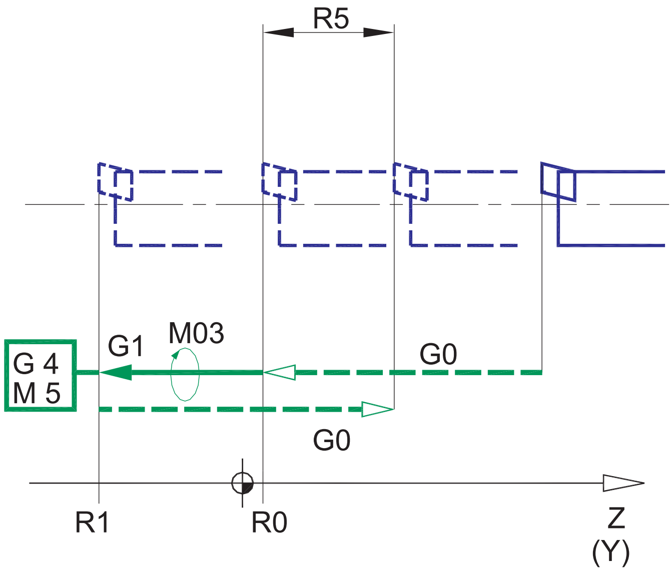
Input parameters

R0 Reference plane
R1 Drilling depth
R5 Retraction, incremental dimension

Application example

```
N100 X100 Y100
N110 R0=0 R1=-40 R5=50
N120 L85
N130 X200
N140 R5=0
N150 L85
```

L86 Reverse boring 2, (with M3 and M5)



8.2.4 - 6 Cycle diagram L86

Input parameters

R0 Reference plane with respect to reference point

R1 Drilling depth

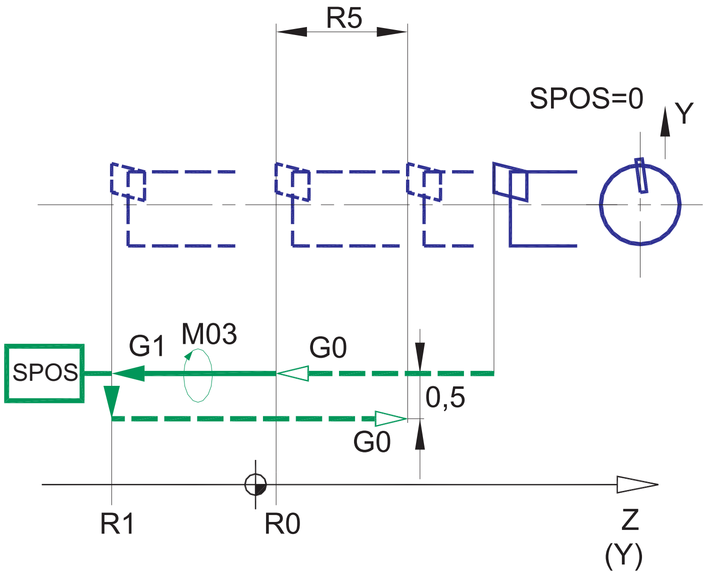
R2 Dwell time in seconds [s]

R5 Retraction, incremental dimension

Application example

```
N100 X100 Y100
N110 R0=0 R1=-40 R2=0.2 R5=0
N120 L86
N130 X200
N140 R5=50
N150 L86
```

L87 Reverse boring 3 with lift off



8.2.4 - 7 Cycle diagram L87

Input parameters

R0 Reference plane

R1 Drilling depth

R5 Retraction, incremental dimension

Lifting off takes place in the -Y direction, the cutting edge position must point to SPOS=0 with +Y.

With a different cutting edge position, L75 can be used with variable spindle alignment position.

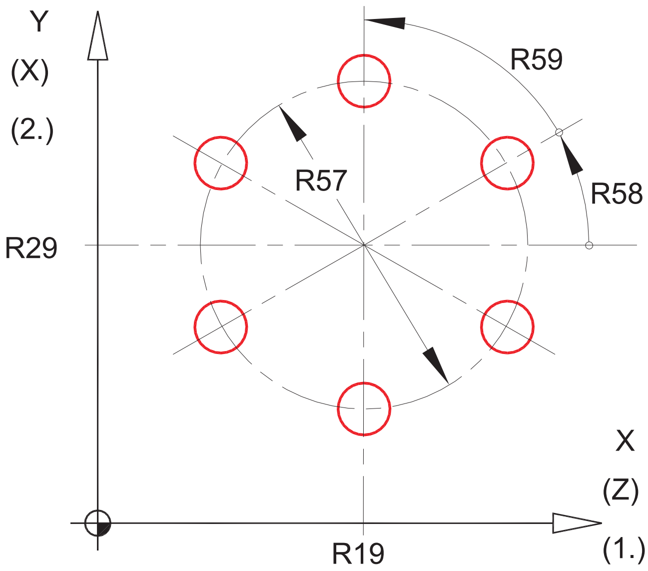
Application example

```
N090 R0=0 R1=-30 R5=0
N100 MCALL L87
N110 X100 Y100
N120 X200
N130 Y150
N140 MCALL
```

8.3 L2xxx Sub-programs for geometry

8.3.1 L20xx Geometry in the plane

8.3.1.1 L2001 Pitch circle



8.3.1 - 1 Cycle diagram L2001

Input parameters

R19 Centre point (G17 X, G18 Z)
R29 Centre point (G17 Y, G18X)
R57 Pitch circle diameter
R58 Initial angle
R59 Incremental angle
R68 Number of machining positions

Sequence description

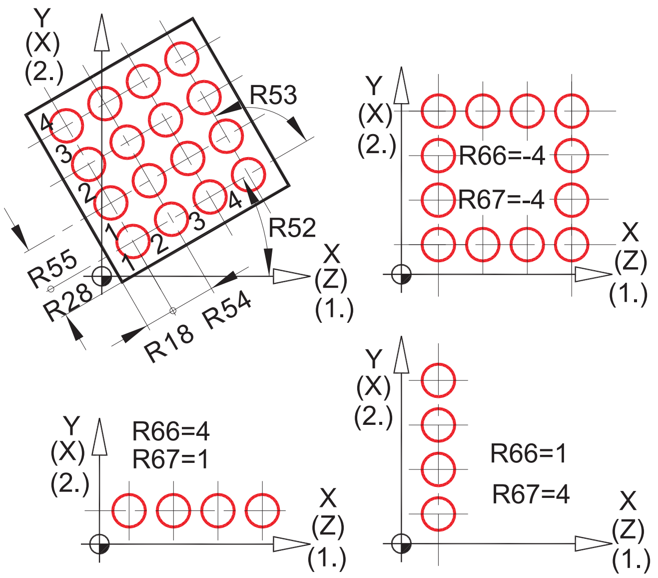
The first machining position is approached. The UP specified by "MCALL" is called. This is repeated for all following machining positions until the specified number of machining operations have been carried out.

The current machining position is located in R10, R20, R18, R28. This also allows a series of hole patterns or machining cycles with positioning details to be called, for example.

Application example

```
N100 R0=0 R1=20 R5=200 R6=1
N110 MCALL L61
N120 R19=100 R29=50 R57=100 R58=0 R59=60 R68=6
N130 L2001
N140 MCALL
```

8.3.1.2 L2002 Hole pattern (rectangle, parallelogram)



8.3.1 - 2 Cycle diagram L2002

Input parameters

R18 Starting position (G17 X, G18 Z)

R28 Starting position (G17 Y, G18 X)

R52 Inclination angle of lines

R53 Angle of columns to lines, normal =90
R54 Column distance (G17 X, G18 Z)
R55 Row distance (G17 Y, G18 X)
R66 Number of columns (G17 X, G18 Z)
R67 Number of lines (G17 Y, G18 X)

i

If R66 and R67 are specified as negative, only the outer drillings (frame) of the rectangle are carried out.

Sequence description

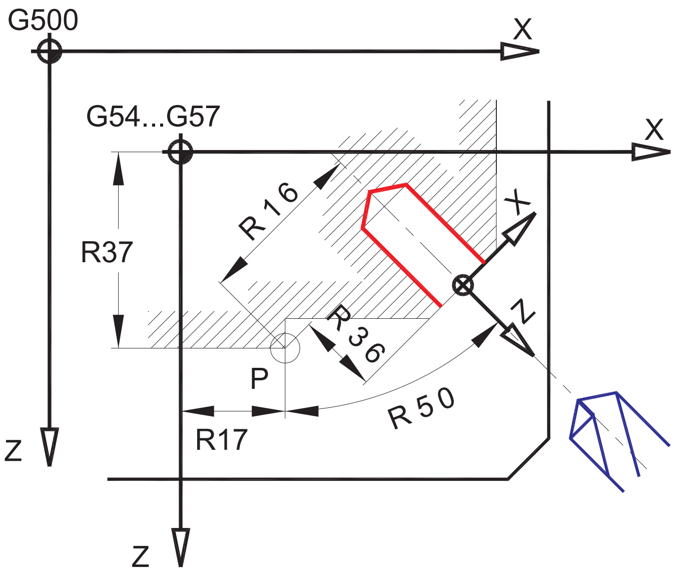
The first machining position is approached. The UP specified by "MCALL" is called. This is repeated for all following machining positions until the specified number of machining operations have been carried out. The sequence is meander shaped for a complete pattern and anticlockwise for the outer border only.

The current machining position is in R10, R20, R19,R29 in each case, the angle of inclination of the lines is also entered in R42. This also allows e.g. a series of pitch circles or machining cycles with positioning details to be called.

Application example

```
N100 R0=0 R1=-30 R5=5
N110 MCALL L81
N120 R18=100 R28=100 R54=30 R55=30 R66=2 R67=5
N130 R52=30 R53=90
N140 L2002
N150 MCALL
```

8.3.1.3 L2004Inclined drilling by rotating about the B-axis



8.3.1 - 3 Cycle diagram L2004

Input parameters

R16 Distance in X drilling centre-reference point (rotated)
R17 Reference point in X
R26 Distance in Y drilling centre-reference point
R27 Reference point in Y
R36 Distance in Z surface-reference point (rotated)
R37 Reference point in Z
R50 Angle of drilling axis to reference system

This sub-program applies exclusively for the ZX-level!
B axis rotates around the Y-axis.

Sequence description

L2004 calculates the XY position and the R0 surface for a bore, the axis of which is randomly rotated in the ZX plane. L2004 causes the calculated XY position to be approached.

A machining sub-program must then be called. The parameter R0 for the surface may no longer be specified as this was calculated by L2004. The B axis must be positioned before L2004 is called up. The position approached is also entered in R10, R20, R18, R28, R19, R29 for provision of any necessary parameters for the sub programs called up.

L2004 assumes that the machine zero point G500 corresponds to the point of rotation of the pallet. The reference point details can be based on any desired zero point offset G54...G57, or G505...G599.

The resulting position of the drilling and the surface R0 can be determined via the specified angle to the reference system on the basis of the coordinates of the reference point P with respect to the current zero point and the spacings of the drilling axis and the surface in the rotated position.

1. If the reference point P lies on the drilling axis and on the surface, then R16, R26, R36 = 0.
2. If the reference point P lies on the drilling axis but not on the surface, then R16, R26 =0, R36 specifies the distance from the surface to the reference point.
3. If the reference point P corresponds exactly to the current zero point (G54...G57, G505...G599), then R17, R27, R37 = 0 and R16, R26, R36 =0 are also specified.

Application example 1

According to the drawing, the reference point is at position B0 with zero point displacement G54. A drilling is to be made under B33 with L61.
The reference point lies on the drilling axis and on the surface.

```
N100 B=DC(33);rotate B pallet to B33;
N110 R17=100 R27=200 R37=300 R50=33 R16=0 R26=0
N120 R36=0
N130 L2004
N140 R1=30 R5=100 R6=2; R0 automatic off L2004;
N150 L61
```

Application example 2

According to the drawing, the reference point is at position B50 with zero point displacement G54. A drilling is to be made at 33 ° with reference to B50 with L61. The reference point lies next to the drilling axis and not on the surface.

```
N100 B=DC(83);rotate B pallet to B=50+33;
N110 R16=100 R17=200 R26=300 R27=33 R36=20 R37=30 R50=40
N120 L2004
N130 R1=30 R5=100 R6=2
N140 L61;R0 off L2004;
```

8.4 Measuring probe - sub-programs

8.4.1 General description

Types of measuring probes

The measuring programs described below work with radially and axially switching measuring probes from Blum, types TC50 and TC51 and radially and axially switching measuring probes from Renishaw, types OMP60 und RMP60.

Measurements with other measuring devices are not taken into account here and must be operated using special sub-programs in individual cases.

Measuring probe interface

A measuring probe interface allows the switching pulses to be transmitted to the controller. The calculation and output of the measurements takes place via the following measuring probe sub-programs.

Use of measuring probe with active FRAME

If the measuring probe sub-programs are used within one or more FRAME commands (TRANS, ATRANS, ROT, AROT, SCALE, ASCALE, MIRROR or AMIRROR), the zero point data may not be changed. Since FRAME commands are referred to the last settable zero point displacement that was called (G54 bis G599), the basis for the FRAME command is destroyed by correcting the zero point displacement. For this reason, measurements are carried out in the definition system and corrected in the machine coordinate system.

8.4.1.1 Measuring probe

Standard measuring probe

Handling

The measuring probe is held in a tool holder like a normal tool and inserted into the work spindle by the tool changing device. The tool name is determined by the user, e.g. "T9990" or "measuring probe".

In the tool data for D1, the radius of the measuring ball is now entered under address DP6 and the length of the measuring probe to the centre of the ball is entered under addressDP3.

Sideways displacement of measuring ball

The sideways displacement of the measuring ball should be as small as possible to prevent errors in particular measurements.

Fixing the measuring probe in the tool holding fixture

When fixing the measuring probe in the tool holding fixture, ensure that the measuring direction in the G17 plane (XY) corresponds to the positive X-axis with the spindle oriented to SPOS=0. In all other planes, the measuring direction is assigned to the positive 1st axis with SPOS=0 .

Switching pulse

The switching pulse from the measuring probe occurs at a deflection of about 0.5 ° in the case of radial measurements.

With axial measurement, switching pulse occurs after approx. 0.2 mm.

Preventing break

The measuring pin on new probes has no predetermined break point. In case of break, the ceramic measuring pin has to be replaced.

Number of cutting edges

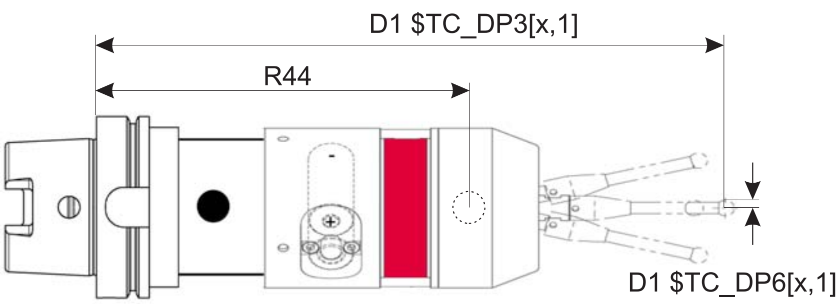
Tips 1 and 5 have to be created bedore the measuring probe into the workpiece magazine. For cutting edge 5, first the length and radius of the default value (0) have to be entered. This also indicates that the measuring probe has not yet been calibrated. During calibration, the calibration values are entered under cutting edge number 5 so that every measuring probe in a system can be assigned its own calibration data.

Blum measuring probe

The direction of measurement is shown by a small arrow on the probe. The probe must be adjusted in the tool holder so that the measuring direction is in the direction of the positive X-axis with spindle alignment position SPOS=0.

Type TC50

Universal measuring probes with multi-directional measuring mechanism.



8.4.1 - 1 Measuring probeBlum type TC50

D1 $TC_DP3[x,1]	Length of the measuring probe to the centre of the ball
D1 $TC_DP6[x,1]	Radius of the measuring ball
R44	Length to measuring pin pivot point

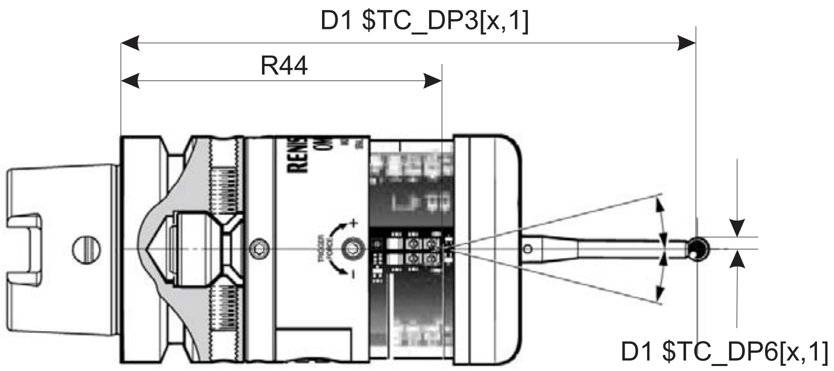
Detaillierte Informationen zum Messtaster entnehmen Sie der Technischen Dokumentation des Herstellers.

Renishaw measuring probes

The RENISHAW OMP60 and RMP60 measuring probes are capable of measuring in all directions, i.e. they are multidirectional.

Type OMP60

Measuring probe system with optical signal transmission.



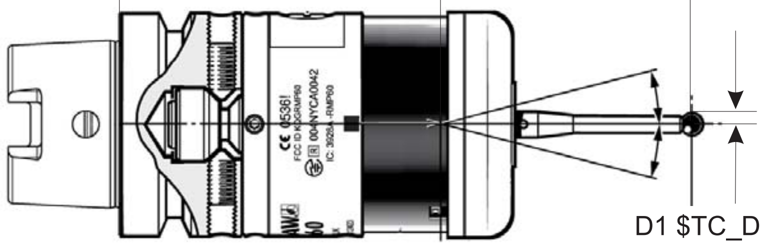
8.4.1 - 2 Measuring probeRenishaw type OMP60

D1 $TC_DP3[x,1]	Length of the measuring probe to the centre of the ball
D1 $TC_DP6[x,1]	Radius of the measuring ball
R44	Length to measuring pin pivot point

Detaillierte Informationen zum Messtaster entnehmen Sie der Technischen Dokumentation des Herstellers.

Type RMP60

Measuring probe with radio signal transmission.



8.4.1 - 3 Measuring probeRenishaw type RMP60

D1 $TC_DP3[x,1]	Length of the measuring probe to the centre of the ball
D1 $TC_DP6[x,1]	Radius of the measuring ball
R44	Length to measuring pin pivot point

Detaillierte Informationen zum Messtaster entnehmen Sie der Technischen Dokumentation des Herstellers.

Special measuring probe

Handling

This includes measuring probes having several combinations of measuring pin, each measuring ball having its own data (radius, length, calibration values) in a cutting edge. The geometric values of the first measuring ball for cutting edge D1 are entered under radius and length. Calibration values for this measuring ball are automatically entered under cutting edge D5, radius and length. The radius and length of the second measuring ball are stored in D2, the calibration values in D6. Each successive measuring ball is assigned a cutting edge number incremented by 1. These measuring pins enable special measurements to be carried out that are not possible with the standard measuring pin.

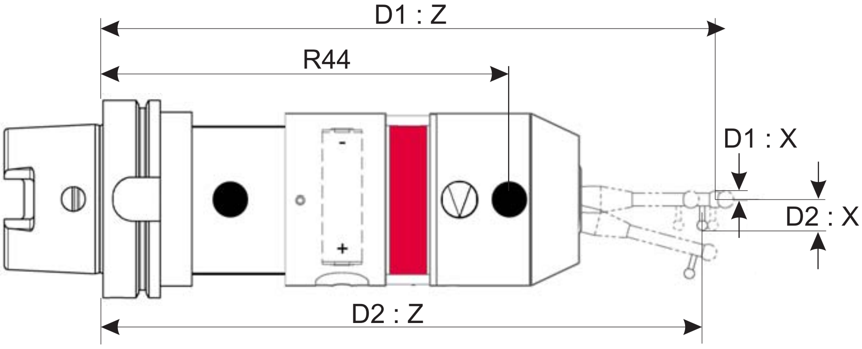
see section
"Standard measuring probe" **page 225**

Special measuring probe from BLUM

The direction of measurement is shown by a small arrow on the probe. The probe must be adjusted in the tool holder so that the measuring direction is in the direction of the positive X-axis with spindle alignment position SPOS=0.

Type TC51

Highly-dynamic measuring probes for rapid, pulling measurements with additional measuring direction in +Z.



8.4.1 - 4 Special measuring probe type TC51 from Blum

The tool data is determined as follows: ...$TC_DP...			
Measuring ball 1	D1	Radius $TC_DP6[x,1]	Length $TC_DP3[x,1]
Calibration values	D5	Radius $TC_DP6[x,5]	Length $TC_DP3[x,5]
Measuring ball 2	D2	Radius $TC_DP6[x,2]	Length $TC_DP3[x,2]
Calibration values	D6	Radius $TC_DP6[x,6]	Length $TC_DP3[x,6]

Detaillierte Informationen zum Messtaster entnehmen Sie der Technischen Dokumentation des Herstellers.

Switching points of measuring probe

Measuring point and measurement result

The measurement result is available under the following variables for the corresponding axes:

Machine coordinate system $AA_MM [axis]

Workpiece coordinate system $AA_MW [axis]

Following a measurement signal, the feed axis is stopped immediately, the remaining traverse is deleted and "Position reached" is displayed. The next block can now be executed. If the CNC receives no measuring signal while the measurement block is being processed, the block is executed normally and the value 0 is entered in the status variable $AC_MEA[n] (n=number of the measuring probe). When the measurement is complete, value 1 is entered in $AC_MEA[n].

The "Measurement carried out" or "Measurement not carried out" cases can thus be distinguished by a program jump. The programmed "Over-travel" must be arranged so that the maximum displacement of the probe is not exceeded to prevent damage to the measuring pin if the measurement is incorrect.

Calibrating the measuring probe

General

A measuring probe must be calibrated before use, i.e. the radial and axial displacements must be determined exactly. The calibration data for each probe or measuring pin are stored in the tool data.

The probe must be recalibrated after changing the measuring pin or making any change to the probe.

Calibrating with high-accuracy measurements

For accurate measurements, calibration and measurement processes should always be carried out with override=100 % and the same feed rate (e.g. F1000). For high-accuracy measurements, recalibration should be carried out every time the measuring probe is fitted in the spindle to eliminate any tool change error.
Suitable calibration cycles are provided to carry out calibration.

See sub-chapter
"L36xx Calibration" page 258

Measuring radially in the plane

Measuring principle

Measurement takes place by radial deflection of the measuring probe by traversing in the appropriate direction. With an uni-directional measuring probe, the work spindle must be aligned so that the traverse direction corresponds to the direction of displacement of the measuring probe (marking on measuring probe). Inclined measurements can also be carried out by linking the spindle orientation to the direction of traverse.

See Section
"Measuring probe" page 225

HELLER

As described above, in the case of very accurate measurements, the probe fitted should be calibrated before each first measurement, in other cases it is sufficient to calibrate after making changes to the measuring probe.

Measuring variants

The various measurement possibilities can be found in the following sub-program descriptions.

The prerequisite is the correct calibration of the probe in advance.

See chapter
"L36xx Calibration" **page 258**

Axial measurement perpendicular to the plane

Measuring principle

Measuring perpendicular to the plane is carried out using the axial switching point.

Measuring variants

The various measurement possibilities can be found in the following sub-program descriptions.
The prerequisite is the correct calibration of the probe in advance.

See sub-chapter
"L36xx Calibration" **page 258**

Charge status of accumulator or battery

Reduction of battery voltage

The reduction of the battery voltage is monitored by fault indication. If, however, a discharged battery or a defective measuring probe is used, no switching signal can be transmitted. Therefore, the functionality of the measuring probe must always be tested with a "positive measurement".

8.4.1.2 Parameters

Internal parameters

Processing internal parameters

The parameters are processed according to the rules stated in the Siemens "Production Engineering" PA, chapter 1.

See programming instructions from Siemens.

Input parameters

General

The input parameters of the measuring cycles are subject to the groupings specified.

See sub-chapter
"Classifying the sub-programs in groups" **page 114**

Details regarding measuring plane

R10 Position of measurement in the plane X

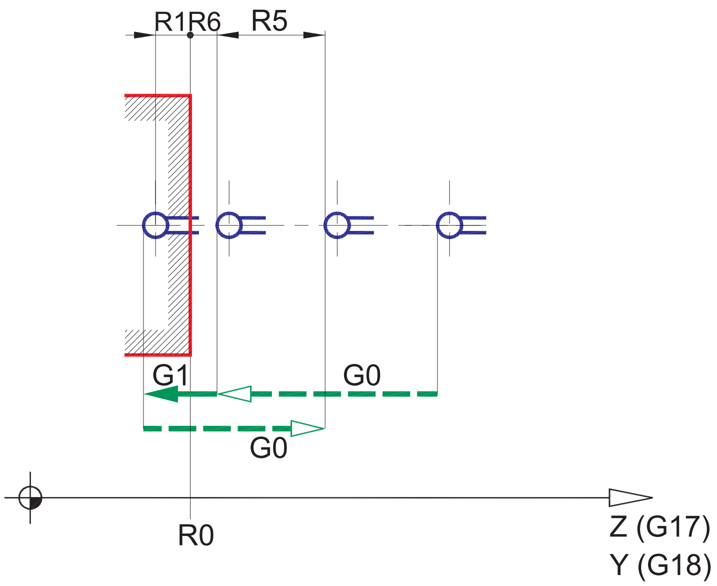
R20 Position of measurement in the plane Y

R40 General dimension details (e.g. width, diameter)

R... General dimension details (e.g. width, diameter)

R49 General dimension details (e.g. width, diameter)

Details regarding measurement depth



8.4.1 - 5 Input parameters

Input parameters

R0 Surface area

R1 Measuring depth (related to R0)
R5 Retraction path in incremental dimension
R6 Safety clearance (radial and axial)

General information

Input parameters	Explanation
R65	Repeat value
R65=1	Simple measurement
R65=2	Double measurement with averaged value
R66	Maximum permissible correction If this is exceeded, an error message is given. An entry in the zero point offset is only made for permissible corrections.
R67=1	Correction is transferred to the zero point offset.
R67=0	No correction. Measured value is only stored in parameters.
R68	Stop condition after occurrence of errors.
R68=1	Stop with M0 if no measurement available and other errors occur.
R68=2	Stop with M0 if measurement available
R68=0	Proceed with caution with R68=0. No stop after occurrence of errors. Error handling must take place using sequence marker R89.

i

"R68=0" is required for unmanned shifts, for example. If an error occurs a jump occurs via "R89" in the program called up. (e.g. replacing the measuring probe and programming measurement repeat).

If the measuring probe sub-programs are used within one or more FRAME commands TRANS, ATRANS, ROT, AROT, SCALE, ASCALE, MIRROR or AMIRROR, the zero point data may not be changed (R67=0). Since these FRAME commands refer to the last settable zero point displacement that was called (G54 to G599), the basis for the FRAME command is destroyed by correcting the zero point displacement. For this reason, measurements are carried out in the definition system and corrected in the machine coordinate system.

L3000 Standard settings

Description

If the standard measuring probe is used, certain input parameters can be allocated initial values by first calling up the L3000 sub-program. These parameters do not then have to be specified again when calling the calibration and measuring cycles. If different details are necessary (e.g. special measuring probe, unmanned operation etc.), the corresponding value is specified when calling up the measuring or calibration cycle. The default value set by L3000 is then overwritten.

Parameter assignment in L3000

Input parameters	Explanation
R1=4	Measuring depth
R6=3	Safety distance
R65=1	Repeat value=1, i.e. a complete measuring cycle
R66=0,5	Maximum permitted correction 0.5 mm
R68=1	Stop condition, stop as long as no measurement has occurred
R70=600	Calibrate and measure standard measuring feed F600 [mm/min]
R44=141	Length to pivot point of measuring pin (standard measuring probe)

L3001 Acceleration correction of axes during measurement

Input parameters

ld_r10 Acceleration correction X-axis
ld_r20 Acceleration correction Y-axis
ld_r30 Acceleration correction Z-axis
ld_r40 Acceleration correction rotary table axis
ld_r50 Acceleration correction, spindle

Sequence description

During the measuring sequence the measuring probe may "debounce", depending on machine type and measuring probe use. This prevents any further measurements. To prevent this situation, L3001 is called before the start of measurement in each cycle and the default values are set. Following the measurement, the acceleration values are reset to 100 % on al axes.

If corrections are required during a measuring sequence, the defaults values for axis acceleration can be altered using L3001.

Default values

X axis	ACC[CA_X]=100
Y-axis	ACC[CA_Y]=100
Z axis	ACC[CA_Z]=100
B-axis	ACC[B]=100
Spindle	ACC[S]=30

Examples of use

Acceleration reduction to standard values defined in the cycle

```
N160 L3001
```

Set acceleration values to 100 %

```
N300 L3001(100,100,100,100,100)
```

Set acceleration value of spindle to 30 %

```
N500 L3001(100,100,100,100,30)
```

Output parameters

Description

Output parameters	Explanation
R80...R88	The measured values are entered in the parameters R80...R88. The individual assignments can be found in the corresponding sub-program description.
R89	Marker for type of measuring sequence.
R89=0	Measuring cycle terminated correctly. Measurement result available.
R89=1	No measurement, no measuring result available. Cause: - Measuring probe defective - No workpiece present or - "Hole is present"
R89=2	Maximum permissible correction exceeded (R66)
R89=99	Measuring cycle is not finished

The measured values are shown in a message on the screen. The measured values are also available in the "PARAMETER" area under "R-PARAMETER" until they are overwritten by a new measurement.

8.4.1.3 Control functions

M54 Measuring probe OFF

M54 switches off the measuring probe.

M54 is the home position.

M54 does not normally have to be programmed, as the measuring probe is switched off at the next tool change.

M56 Measuring probe On

Function M56 is used to switch on the measuring probe and the measuring probe interface. Additionally, transmission between measuring probe and interface is visually checked or on a wireless basis. When switched on, the measuring probe must always be connected to the interface. If transmission is interrupted, an error message is displayed and the machine stops immediately. If, however, you are using a "defective" measuring probe or working with exclusively "negative measurements", which means no measuring probe switching point is expected, a function test must be carried out by a "positive measurement" and a query of the status signal $AC_MEA[n]==1.

MEAS=1 Measurement block

The command "MEAS=1" and an interpolation type are used to traverse to actual positions on the workpiece and transfer measured values. The remaining distance between desired and actual position is deleted.

"MEAS=1"is programmed in a block with movement instructions. The feed rate must be adapted to suit the particular measurement case.

See chapter 5.5. for more information: "Measuring with switching probes, MEAS, MEAW", PA Siemens "Production engineering"!

System variables

$AA_MM[Achse]

$AA_MW[Achse]

$AC_MEA[n]

The meaning of the variables that are used during the measuring process are described in the Siemens PA "Production engineering" SINUMERIK 840D.

8.4.1.4 Program advice

Block search

All measuring cycles are generally skipped during block search by interrogating the sequence memory $P_SEARCH, as a calculation can only take place if a measuring result is present. Access to measuring cycles that have already started is also prevented.

Call up measuring programs

Measuring cycles are called with the address L... within the structure of the NC program (e.g. L3101). The appropriate input parameters must be specified before the cycle is called up. The cycles are normally processed in an automatic sequence.

However, the cycles can also be called in MDA mode.

If the default settings have previously been made using the L3000 sub-program, these do not need to be specified again when calling up the measurement cycle.

See section
"Switching points of measuring probe" page 229

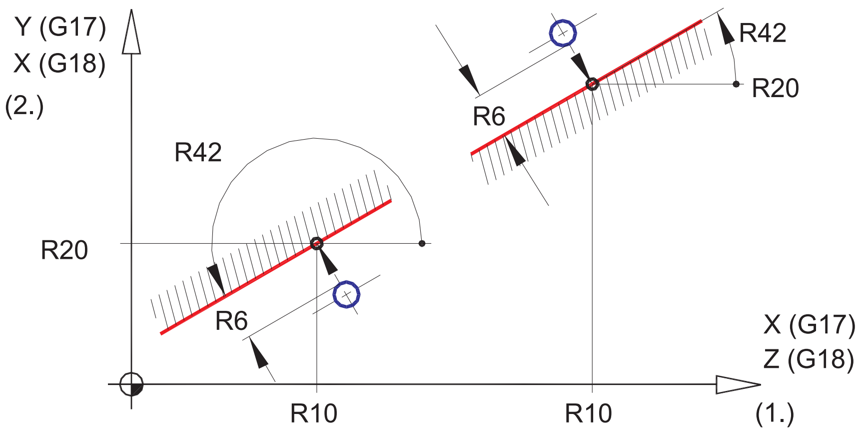
It must be ensured that calibration and measurement are carried out at the same feed rate.

See section
"Switching points of measuring probe" page 229

Beachten Sie die Technische Dokumentation des Messtaster-Herstellers.

8.4.2 L31xx Measuring in the plane

8.4.2.1 L3101 Scan, measure/check edge



8.4.2 - 1 Measuring plane cycle diagram L3101

Input parameters measuring plane	
Input parameters and explanation	
R0	Surface Z (G17) / Y (G18)
R1	Measuring depth
R5	Retraction path
R6	Safety distance
R10	1. Coordinate X (G17) / Z (G18)
R20	2. Coordinate Y (G17) / X (G18)
R42	Angle of inclination
R65	Repeat value 1, 2
R66	Maximum permissible correction
R67	Zero point offset correction no=0, yes=1
R68	Stop condition on error: 1, 2, 0
R68=1	Stop with M0 if no measurement available and other errors occur
R68=2	Stop with M0 if unexpected measurement available and other errors
R68=0	No stop for errors, caution with R68=0 Errors then handled using R89!

Input parameters and explanation	
R70	Measuring feed rate

Output parameters measuring plane

Output parameters and explanation	
R81	Measured edge in X (G17) / Z (G18)
R82	Measured edge in Y (G17) / X (G18)
R89	Marker for measurement sequence types 0, 1, 2, 99
R89=0	Measurement OK, result available
R89=1	No measurement available
R89=2	Maximum permissible correction exceeded (R66)
R89=99	Measuring cycle has not terminated.

If a horizontal edge is measured, only R82 (Y/X) is of interest, if a vertical edge, then only R81 (X/Z), if the measurement is sloping (for example less than 30 °), then both values are needed.

Sequence description

The machine traverses to the start position depending on the selected approach direction, the spindle is aligned and the measuring probe is switched on. Feed then takes place on the measuring plane (R0-R1) and the edge is approached at the measuring feed rate. With repeat value R65=2, the measurement is repeated and averaged. Movement then takes place to the withdrawal position.

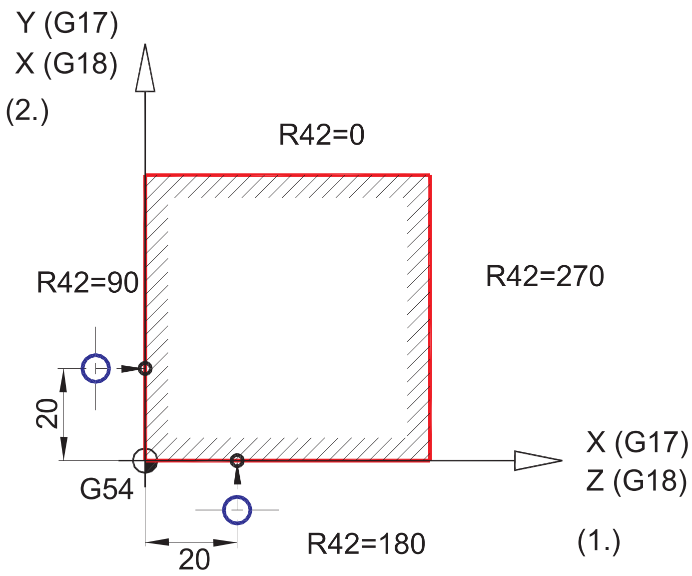
As soon as a measuring result is available, it is compared to the permissible correction R66, and for R67=1 the correction is also entered in the active zero point offset. The machine is normally brought to a stop in case of errors with R68.

If the presence of a surface is to be checked, R68=1 can be used to ensure that an error message and stop is issued if no switching impulse has been registered.

The values for the measured edges in the X and Y directions are quoted in R81 and R82.

Application example

Measuring task:
The zero point G54 is to be set exactly corresponding to a vertical and a horizontal edge with a measuring probe so that a hole pattern can be machined at the correct dimension from the edges.



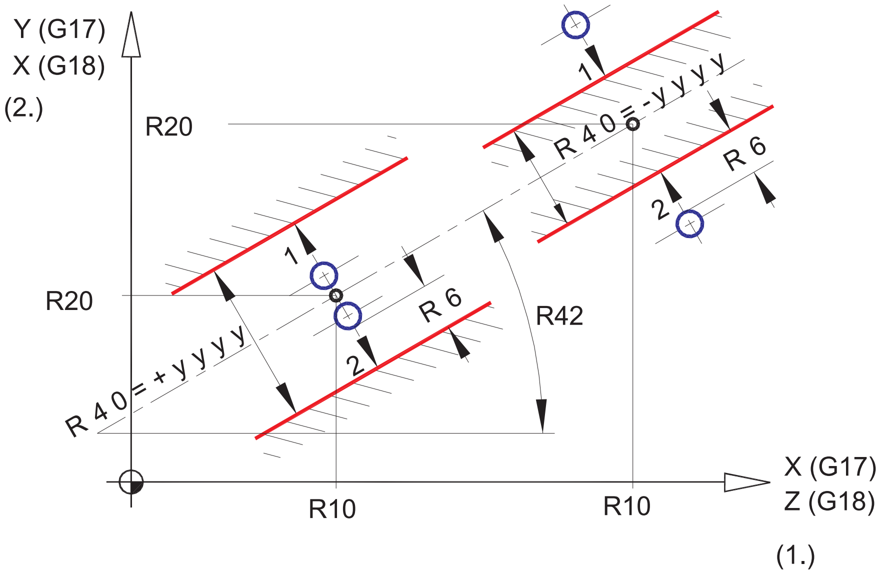
8.4.2 - 2 Measuring task L3101

without cycle support

```
N100 (Fit in measuring probe, cutting edge D1 active)
N100 CS_TOOL("9990",1)
N110 G54
N120 L3000 ;Set default values;
N130 R10=20 R20=0 R42=180 R0=0 R5=0 R67=1
N140 L3101 ;horizontal edge, from below;
N150 R10=0 R20=20 R42=90
N160 L3101 ;vertical edge, from left;
```

HELLER

8.4.2.2 L3102 Measuring centre and width of groove/web



8.4.2 - 3 Measuring plane cycle diagram L3102

Input parameters measuring plane

Input parameters and explanation	
R0	Surface Z (G17) / Y (G18)
R1	Measuring depth
R5	Retraction path
R6	Safety distance
R10	1. Coordinate X (G17) / Z (G18)
R20	2. Coordinate Y (G17) / X (G18)
R40	Width, +=groove, -=web
R42	Angle of inclination
R65	Repeat value 1, 2
R66	Maximum permissible correction
R67	Zero point offset correction no=0, yes=1
R68	Stop condition on error: 1, 2, 0
R68=1	Stop with M0 if no measurement available and other errors occur
R68=2	Stop with M0 if unexpected measurement available and other errors

Input parameters and explanation	
R68=0	No stop for errors, caution with R68=0 Errors then handled using R89!
R70	Measuring feed rate

Output parameters measuring plane

Output parameters and explanation	
R81	Measured edge in X (G17) / Z (G18)
R82	Measured edge in Y (G17) / X (G18)
R83	Measured width
R89	Marker for measurement sequence types 0, 1, 2, 99
R89=0	Measurement OK, result available
R89=1	No measurement available
R89=2	Maximum permissible correction exceeded (R66)
R89=99	Measuring cycle has not terminated

i

If a horizontal groove/web is measured, only R82 (Y) is of interest, if a vertical is measured, then only R81 (X). If the measurement is sloping, for example 30°, then both values are required.

Sequence description

The machine traverses to the start position in accordance with the selected approach direction, the spindle is aligned and the measuring probe is switched on. Feed then takes place on the measuring plane (R0-R1) and the edge is approached at the measuring feed rate. With repeat value R65=2, the measurement is repeated and averaged. After correct measurement of the 1st edge, the edge lying above is approached and measurement is made upwards. If R5=0 is specified, the probe pauses when traversing on the measuring plane, withR5>0 it is retracted to the safety clearance plus retraction path R5. Movement then takes place to the withdrawal position.

As soon as a measuring result is available, it is compared to the permissible correction R66 for both the X-Y position (R81, R82) and the width (R83).

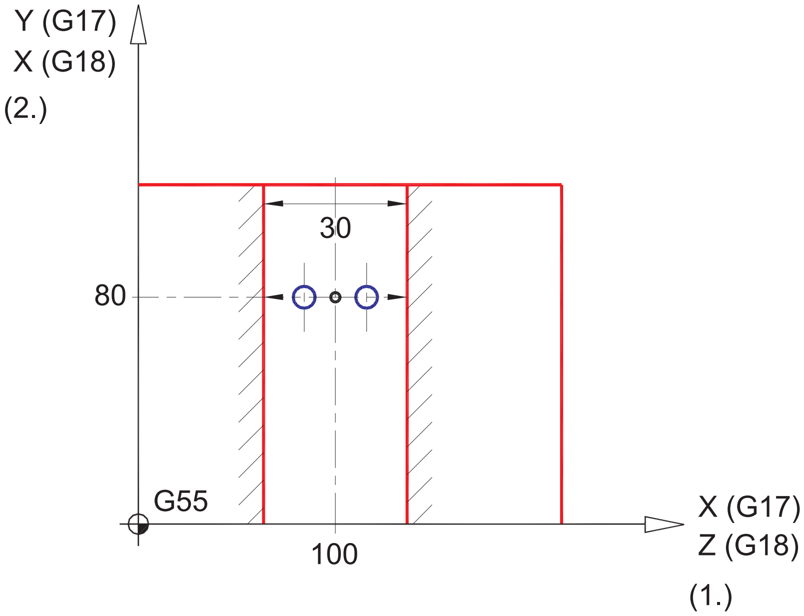
For R67=1, the correction is also entered into the active zero point offset.

With R68, the machine is usually stopped in the case of an error.

The values for the measured centre in the X and Y directions are in R81 and R82 and the measured width is in R83.

Application example

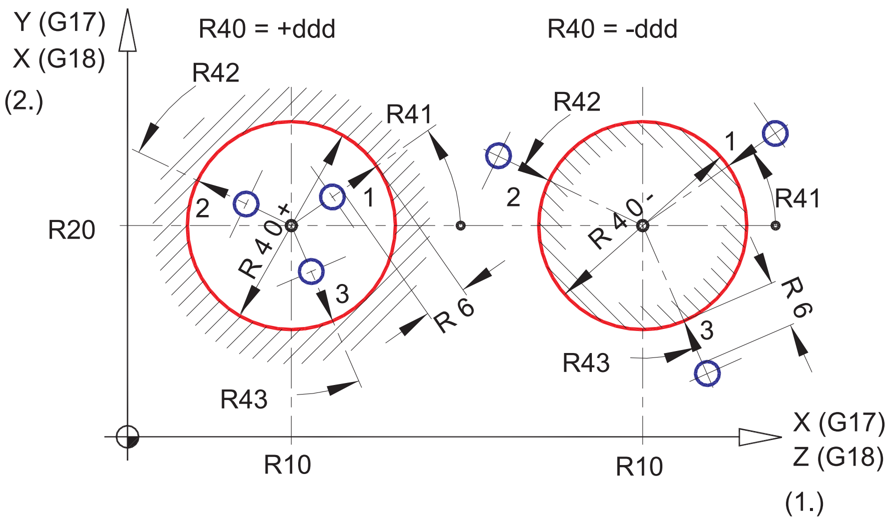
Measuring task:
The zero point G55 in the X direction is to be set corresponding to a vertical groove. The groove width and displacement may not be more than 0,2 mm greater than the nominal dimension.



8.4.2 - 4 Measuring task L3102

```
N100 ;Fit measuring probe, cutting edge 1 active;
N110 CS_TOOL("9990",1)
N120 G55
N130 ;Rated width; = 30.1 +- 0.1
N140 R10=100 R20=80 R40=30.1 R42=90
N150 R0=0 R1=4 R5=100 R6=2 R65=1 R66=0.1 R67=1
N160 R68=1 R70=1000
N170 L3102
```

8.4.2.3 L3103 3-point measurement of bores or journals



8.4.2 - 5 Measuring plane cycle diagram L3103

Input parameters measuring plane

Input parameters and explanation	
R0	Surface Z (G17) / Y (G18)
R1	Measuring depth
R5	Retraction path
R6	Safety distance
R10	1. Coordinate X (G17) / Z (G18)
R20	2. Coordinate Y (G17) / X (G18)
R40	Diameter, + = Bore, - = Pin
R41	Inclination angle for 1st measurement
R42	Inclination angle for 2nd measurement
R43	Inclination angle for 3rd measurement
R65	Repeat value 1, 2
R66	Maximum permissible correction
R67	Zero point offset correction no=0, yes=1
R68	Stop condition on error: 1, 2, 0
R68=1	Stop with M0 if no measurement available and other errors occur
R68=2	Stop with M0 if unexpected measurement available and other errors

Input parameters and explanation	
R68=0	No stop for errors, caution with R68=0 Errors then handled using R89!
R70	Measuring feed rate

Output parameters measuring plane

Output parameters and explanation	
R81	Measured centre in X (G17) / Z (G18)
R82	Measured centre in Y (G17) / X (G18)
R83	Measured diameter
R89	Marker for measurement sequence types 0, 1, 2, 99
R89=0	Measurement OK, result available
R89=1	No measurement available
R89=2	Maximum permissible correction exceeded (R66)
R89=99	Measuring cycle is not finished

Sequence description

The machine traverses to the start position in accordance with the selected approach direction, the spindle is aligned and the measuring probe is switched on. Feed then takes place on the measuring plane (R0-R1) and the first measuring point is approached at the measuring feed rate. If R5=0 is specified, the probe pauses when traversing to the next measuring point on the measuring plane, with R5>0 it is retracted to the safety clearance plus retraction path R5. With repeat value R65=2, the measurement (2*3 measurements) is repeated and averaged. Movement then takes place to the withdrawal position.

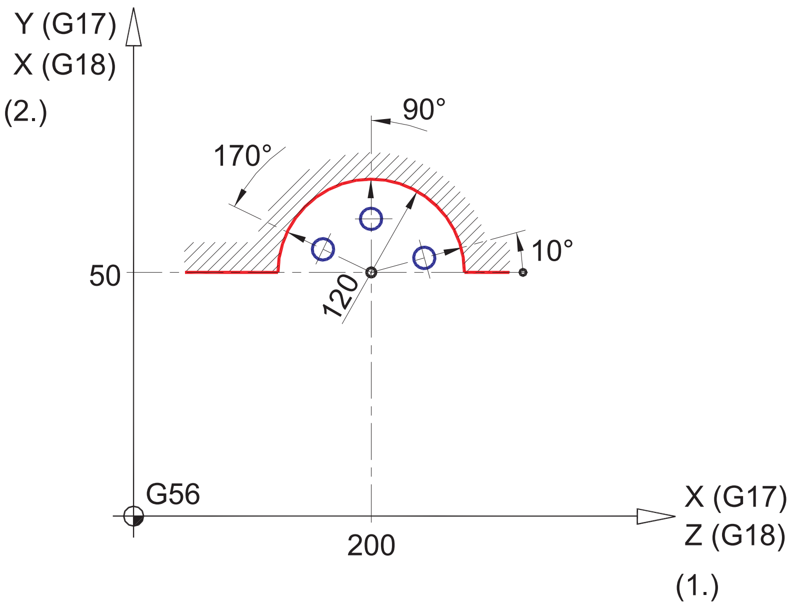
As soon as a measuring result is available, it is compared to the permissible correction R66 (centre and diameter), and for R67=1 the correction is also entered in the active zero point offset. The machine is normally brought to a stop in case of errors with R68.

The values for the measured centre in the X and Y directions are in R81 and R82 and the measured diameter is in R83.

Application example

Measuring task:

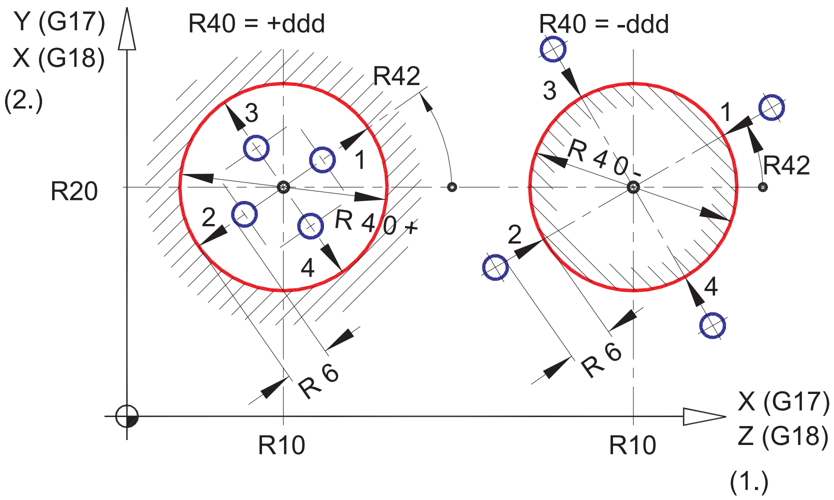
The zero point G56 is to be set corresponding to the centre point of a bored half-shell. The offset and diameter difference may only be less than +- 0,15 mm.



8.4.2 - 6 Measuring task L3103

```
N100 ;Fit measuring probe, cutting edge 1 active;
N110 CS_TOOL("9990",1)
N120 G56
N130 L3000;Set default values R1, R6, etc.;
N140 R10=200 R20=50 R40=120 R41=10 R42=90 R43=170
N150 R0=0 R5=0 R66=0.15 R67=1
N160 L3103;Measurements 10,90,170 degrees;
```

8.4.2.4 L3104 4-point measurement of bores or journals



8.4.2 - 7 Measuring plane cycle diagram L3104

Input parameters measuring plane

Input parameters and explanation	
R0	Surface Z (G17) / Y (G18)
R1	Measuring depth
R5	Retraction path
R6	Safety distance
R10	1. Coordinate X (G17) / Z (G18)
R20	2. Coordinate Y (G17) / X (G18)
R40	Diameter, + = Bore, - = Pin
R42	Angle of inclination
R65	Repeat value 1, 1.5, 2
R66	Maximum permissible correction
R67	Zero point offset correction no=0, yes=1
R68	Stop condition on error: 1, 2, 0
R68=1	Stop with M0 if no measurement available and other errors occur
R68=2	Stop with M0 if unexpected measurement available and other errors
R68=0	No stop for errors, caution with R68=0 Errors then handled using R89!

Input parameters and explanation	
R70	Measuring feed rate

Output parameters measuring plane

Output parameters and explanation	
R81	Measured centre in X
R82	Measured centre in Y direction
R83	Measured diameter in (from 1st and 2nd measurement)
R84	Measured diameter in (from 3rd and 4th measurement)
R89	Marker for type of measurement sequence 0, 1, 2, 99
R89=0	Measurement OK, result available
R89=1	No measurement available
R89=2	Maximum permissible correction exceeded (R66)
R89=99	Measuring cycle is not finished

Sequence description

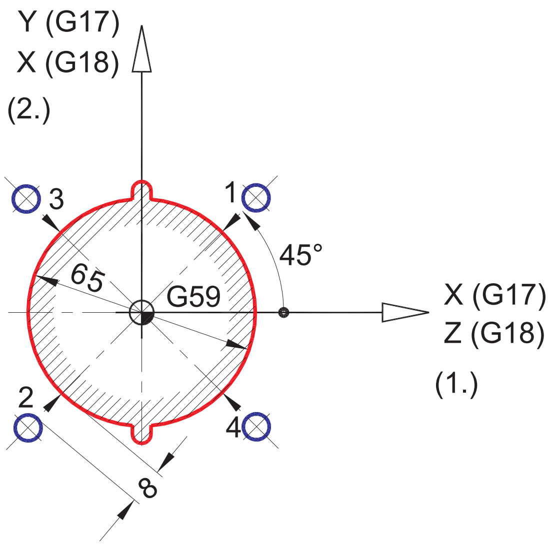
The machine traverses to the start position in accordance with the selected approach direction, the spindle is aligned and the measuring probe is switched on. Feed then takes place on the measuring plane (R0-R1) and the first measuring point is approached at the measuring feed rate. If R5=0 is specified, the probe pauses when traversing to the next measuring point on the measuring plane, with R5>0 it is retracted to the safety clearance plus retraction path R5. With repeat value R65=1, 4 measurements are carried out. If repeat value R65=1,5, the 1st and 2nd measurements are repeated, with R65=2 all 4 measurements are repeated. Movement then takes place to the withdrawal position.

As soon as a measuring result is available, it is compared to the permissible correction R66 (centre and diameter), and for R67=1 the correction is also entered in the active zero point offset. With R68, the machine is usually stopped in the case of an error.

R81 and R82 contain the value for the measured centre in X and Y, R83 contains the measured diameter in X (i.e. from 1st and 2nd measurement), and R84 contains the diameter in Y (i.e. from 3rd and 4th measurement).

Application example

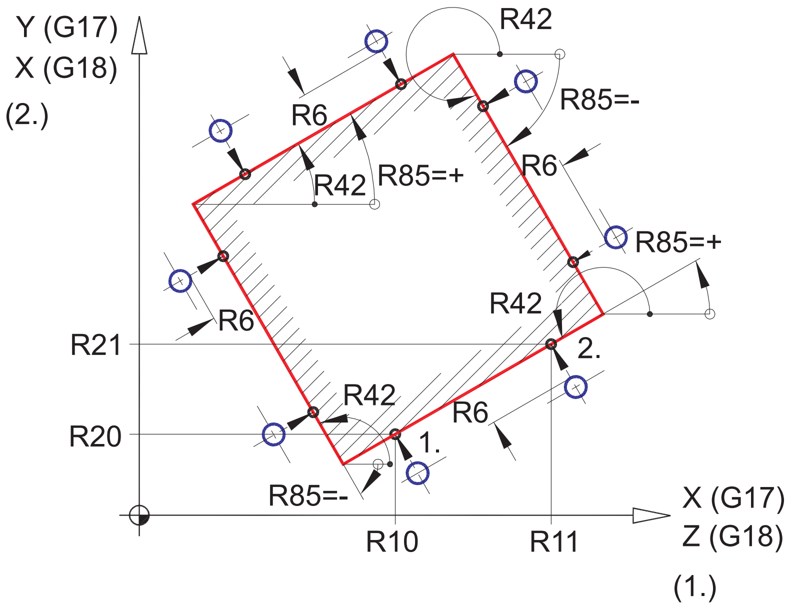
Measuring task:
The zero point G509 is to be set to d=65 corresponding to the centre of a raw shaft end. The measurement is to be carried out as 4*90 ° measurement. However, there is a forging burr top and bottom and measurement must not be made in those areas. Offset and diameter can vary up to 5 mm.



8.4.2 - 8 Measuring task L3104

```
N100;Fit measuring probe, cutting edge 1 active;
N110 CS_TOOL("1234",1)
N120 G509
N130 L3000 ;Set default values R1, etc.;
N140 R10=0 R20=0 R40=-65 R42=45 R0=350 R5=50 R6=8
N150 R66=5 R67=1
N160 L3104
```

8.4.2.5 L3105 Measure inclined position in XY/YZ plane



8.4.2 - 9 Measuring plane cycle diagram L3105

Input parameters measuring plane

Input parameters and explanation	
R0	Surface Z (G17) / Y (G18)
R1	Measuring depth
R5	Retraction path
R6	Safety distance
R10	1. Measuring point 1. axis X (G17) / Z (G18)
R11	2. Measuring point 1. axis X (G17) / Z (G18)
R20	1. Measuring point 2. axis Y (G17) / X (G18)
R21	2. Measuring point 2. axis Y (G17) / X (G18)
R42	Angle of inclination
R65	Repeat value 1, 2
R66	Maximum permissible correction
R67	Zero point offset correction no=0
R68	Stop condition on error: 1, 2, 0
R68=1	Stop with M0 if no measurement available and other errors occur

Input parameters and explanation	
R68=2	Stop with M0 if unexpected measurement available and other errors
R68=0	No stop for errors, caution with R68=0 Errors then handled using R89!
R70	Measuring feed rate

Output parameters measuring plane

Output parameters and explanation	
R81	1st measuring point in X (G17) / Z (G18)
R82	1st measuring point in Y (G17) / X (G18)
R83	2nd measuring point in X (G17) / Z (G18)
R84	2nd measuring point in Y (G17) / X (G18)
R85	Measured angle to positive 1st axis
R89	Marker for measurement sequence types
R89=0	Measurement OK, result available
R89=1	No measurement available
R89=2	Maximum permissible correction exceeded (R66)
R89=99	Measuring cycle is not finished

Sequence description

With this program, sub-program L3101 is called in each case at the 1st and 2nd measuring points.

R81 and R82 contain the values for the 1st measuring point in X and Y.

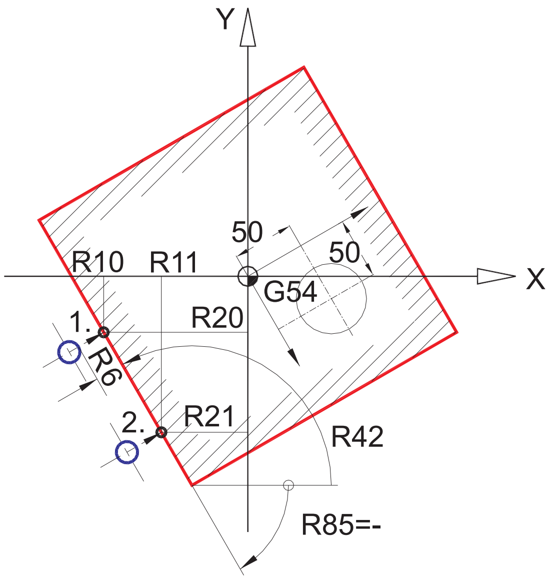
R83 and R84 contain the values for the 2nd measuring point in X and Y.

The angle between the two measuring points is calculated and output in R85. Regardless of the measurement direction, the angle R85 is output scaled to +- 90° to the positive 1st axis of the plane. This angle can be called, for example for coordinate rotation with ROT or AROT.

Application example

Measuring task:

The angle of the workpiece is to be measured with respect to the left bottom edge so that the workpiece can be rotated to the zero point using "ROT Z". The dimensions of the workpiece assume it is not rotated.

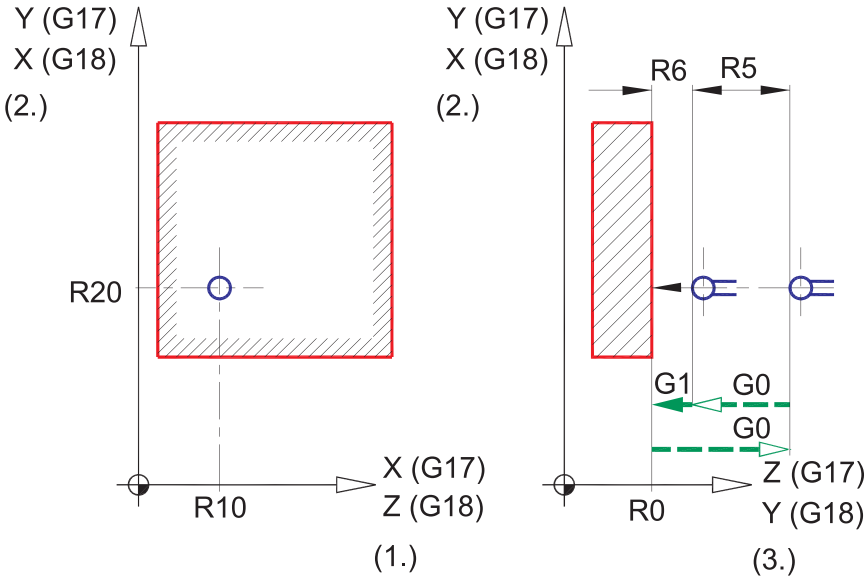


8.4.2 - 10 Measuring task L3105

```
N100 CS_TOOL("9990",1);Fit measuring probe;
N110 G54
N120 R10=-75 R20=-39 R11=-23.6 R21=-75 R42=180-60
N130 R0=0 R1=5 R5=10 R6=2 R65=1 R66=1 R67=0 R68=1
N140 R70=600
N150 L3105;measure inclined edge, bottom left;
N160 ROT RPL=R85;Coordinate rotation about zero point G54 wit
     ...
N170 X50 Y50;Machine drilled hole;
N180
```

8.4.3 L32xx Measure perpendicular to the plane

8.4.3.1 L3201 Scan surface axially



8.4.3 - 1 Cycle diagram L3201

Input parameters measuring plane

Input parameters and explanation	
R0	Surface Z (G17) / Y (G18)
R5	Retraction path
R6	Safety distance
R10	1. Coordinate X (G17) / Z (G18)
R20	2. Coordinate Y (G17) / X (G18)
R65	Repeat value 1, 2
R66	Maximum permissible correction
R67	Zero point offset correction no=0, yes=1
R68	Stop condition on error: 1, 2, 0
R68=1	Stop with M0 if no measurement available and other errors occur
R68=2	Stop with M0 if unexpected measurement available and other errors

Input parameters and explanation	
R68=0	No stop for errors, caution with R68=0. Errors then handled using R89!
R70	Measuring feed rate

Output parameters measuring plane

Output parameters and explanation	
R83	Measured surface Z (G17) / Y (G18)
R89	Marker for measurement sequence types 0, 1, 2, 99
R89=0	Measurement OK, result available
R89=1	No measurement available
R89=2	Maximum permissible correction exceeded (R66)
R89=99	Measuring cycle is not finished

Sequence description

The machine traverses to the start position, the spindle is aligned and the measuring probe is switched on. Feed then takes place to the surface and the measuring point is approached axially at measuring feed rate.

The measuring depth is calculated using the double value of R6. R6 must not be selected too large in order to prevent the overtravel of the measuring probe being exceeded.

Beachten Sie die Technische Dokumentation des Messtaster-Herstellers.

With a repeat value of R65=2, the measurement is repeated and the measured values averaged. Movement then takes place to the withdrawal position.

As soon as a measuring result is available, it is compared to the permissible correction R66, and for R67=1 the correction is also entered in the active zero point offset. With R68, the machine is usually stopped in the case of an error.

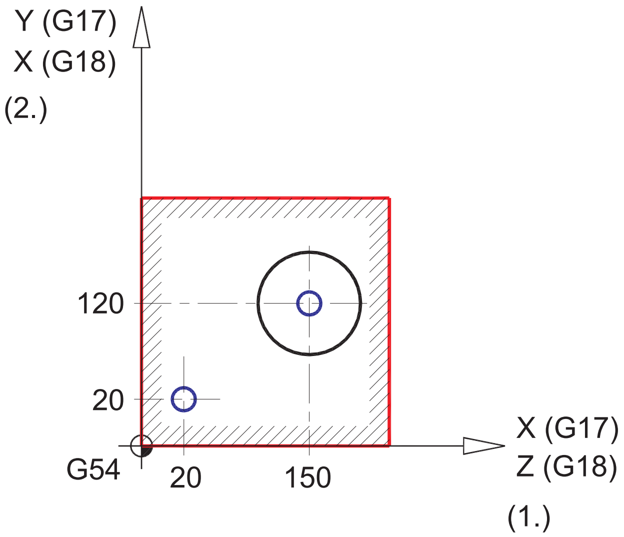
If the presence of a surface is to be checked, R68=1 can be used to ensure that an error message and stop is issued if no switching impulse has been registered.

If the presence of a bore is to be checked, "R68=2" can be used to ensure that an error message and stop is issued if a switching impulse has been registered.

The value for the measured surface is quoted in "R83".

Application example

Measuring task:
The zero point G54 Z.... is to be corrected with respect to an existing surface by a maximum of 0,4 mm. In addition, the presence of a particular hole is to be checked in position X=150 Y120. If the hole is not present, the machine must stop.

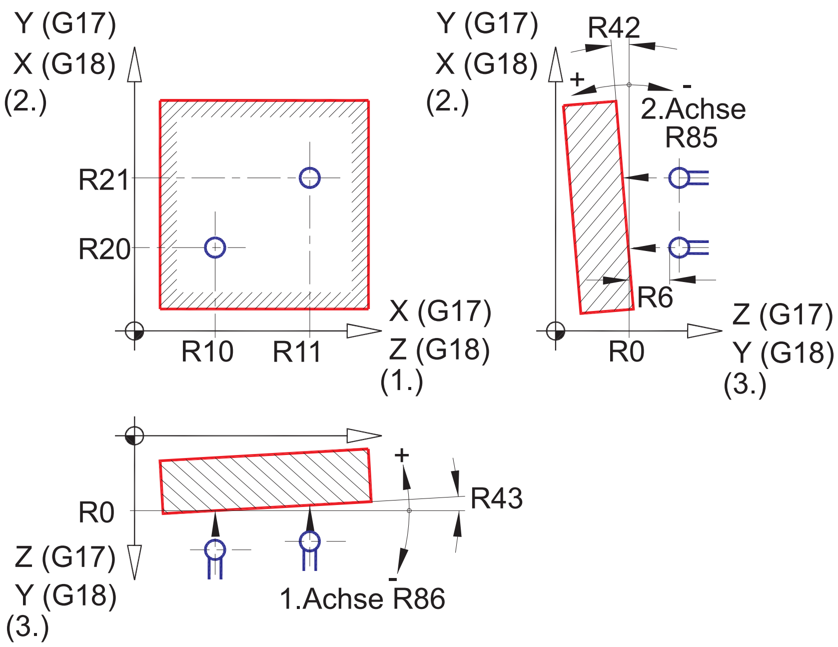


8.4.3 - 2 Measuring task L3201

```
N100 ;Fit measuring probe, cutting edge 1 active;
N110 CS_TOOL("2345",1)
N120 G54
N130 ;Measure Z twice, stop if no measurement occurs;
     L3201(20,20,100,0,2,2,0.4,1,1,600)
N140 ;Bore present?, stop if measurement occurs;
N150 L3201(150,120,100,0,2,1,1,0,2,600)

N160
```

8.4.3.2 L3202 Measuring inclined position, 2nd/ 3rd axis



8.4.3 - 3 Cycle diagram L3202

Input parameters measuring plane

Input parameters and explanation	
R0	Surface Z (G17) / Y (G18)
R5	Retraction path
R6	Safety distance
R10	1. Measuring point 1. axis X (G17) / Z (G18)
R11	2. Measuring point 1. axis X (G17) / Z (G18)
R20	1. Measuring point 2. axis Y (G17) / X (G18)
R21	2. Measuring point 2. axis Y (G17) / X (G18)
R42	Inclination angle for 2nd axis, with no inclination 0 must be specified
R43	Inclination angle for 3rd axis, with no inclination 0 must be specified
R65	Repeat value 1, 2
R66	Maximum permissible correction
R67	Zero point offset correction no=0
R68	Stop condition on error: 1, 2, 0

Input parameters and explanation	
R68=1	Stop with M0 if no measurement available and other errors occur
R68=2	Stop with M0 if unexpected measurement available and other errors
R68=0	No stop for errors, caution with R68=0 Errors then handled using R89!
R70	Measuring feed rate

Output parameters measuring plane

Output parameters and explanation	
R83	1st measuring point in Z/Y/X
R84	2nd measuring point in Z/Y/X
R85	measured angle to the positive 2nd axis Y/X
R86	Measured angle to positive 1st axis X/Z
R89	Marker for measurement sequence types 0, 1, 2, 99
R89=0	Measurement OK, result available
R89=1	No measurement available
R89=2	Maximum permissible correction exceeded (R66)
R89=99	Measuring cycle is not finished

Sequence description

With this program, sub-program L3201 is called in each case at the 1st and 2nd measuring points.

The angle between the two measuring points is calculated.

However, this program can only determine the angle parallel to either the 1st or 2nd axis.

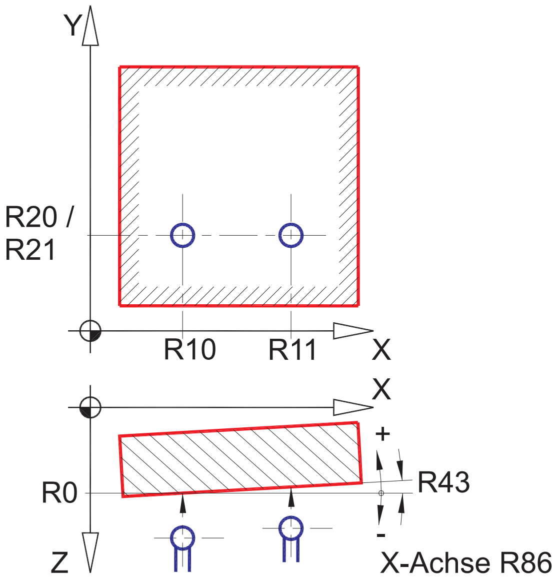
The angle to the 1st axis X/ Z can be used, for example, to rotate the B-axis.

The angle to the 2nd Y/ X can be used, for example, to detect faulty clamping.

Application example

Measuring task:

A workpiece is clamped at an incline in the ZX plane. The angle to the X axis is used for rotation of the B axis.



8.4.3 - 4 Measuring task L3202

```
N100 CS_TOOL("9990",1);Fit measuring probe;
N110 G54
N120 R10=30 R20=50 R11=100 R21=50 R42=0 R43=0
N130 R0=0 R5=100 R6=2 R65=1 R66=1 R67=0 R68=1
N140 R70=600
N150 L3202;Measure inclined workpiece;
N160 G54 B=DC(R86);Set B-axis to measured angle;
```

8.4.4 L36xx Calibration

General

The measuring probe must be calibrated before use or after a change to the measuring pin. This is carried out both radially and axially with the following calibration cycles.

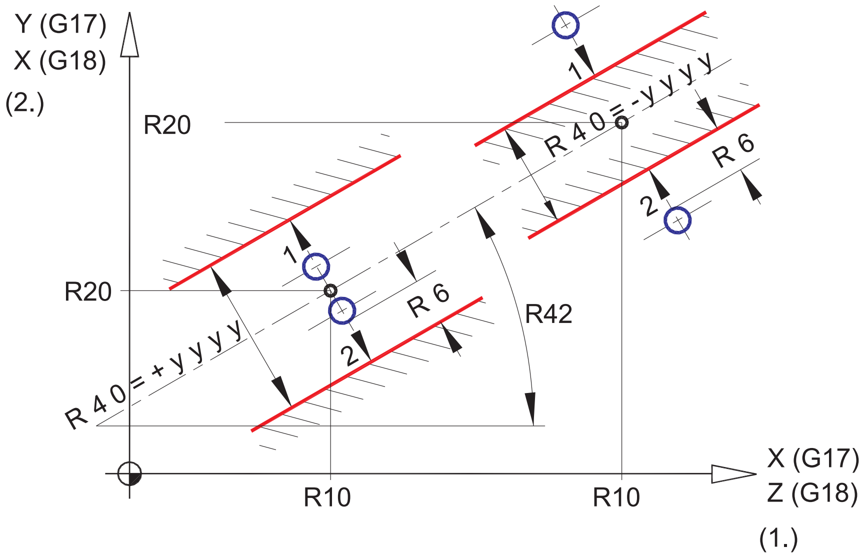
See section
"Calibrating the measuring probe" **page 230**

To calculate the displacement of any measuring pin, the length (R44) of the measuring probe up to the point of rotation of the measuring pin is required for the calibration cycles.

HELLER

See section
"Standard measuring probe" **page 225**

8.4.4.1 L3601 Calibration, radial to groove/web



8.4.4 - 1 Cycle diagram L3601

Input parameters

Input parameters and explanation	
R0	Surface Z (G17) / Y (G18)
R1	Measuring depth
R5	Retraction path
R6	Safety distance
R10	1. Coordinate X (G17) / Z (G18)
R20	2. Coordinate Y (G17) / X (G18)
R40	Width, +=groove, -=web
R42	Angle of inclination
R43	Inclination angle for 3rd measurement
R44	Length to measuring pin pivot point
R65	Repeat value 1, 2
R66	Maximum permissible repeat difference. Standard=0.005 mm

Input parameters and explanation	
R68	Stop condition on error: 1, 0
R68=1	Stop with M0 if no measurement available and other errors occur
R68=0	No stop for errors, caution with R68=0 Errors then handled using R89!
R70	Measuring feed rate

Output parameters

The calibration value is automatically entered in the cutting edge specified for the calibration value.

See section
"Standard measuring probe" page 225

See section
"Special measuring probe" page 228

Output parameters and explanation	
R89	Marker for measurement sequence types 0, 1, 2, 99
R89=0	Measurement OK, result available
R89=1	No measurement available
R89=2	Maximum permissible correction exceeded (R66)
R89=99	Measuring cycle is not finished

Sequence description

The machine traverses to the start position in accordance with the selected approach direction, the spindle is aligned and the measuring probe is switched on. Feed then takes place on the measuring plane (R0-R1) and the edge is approached twice at the measuring feed rate. The repeat difference is compared with the permissible repeat difference R66.

Groove measurement:

If R5=0 is specified, the probe pauses when traversing in the groove on the measuring plane, with R5>0 it is retracted to the safety clearance plus retraction pathR5.

Web measurement:

Lift off to the safety clearance usually takes place.

Movement then takes place to the withdrawal position.

The calibration value determined is automatically entered into the corresponding cutting edge under address radius 1.

With R68, the machine is usually stopped in the case of an error.

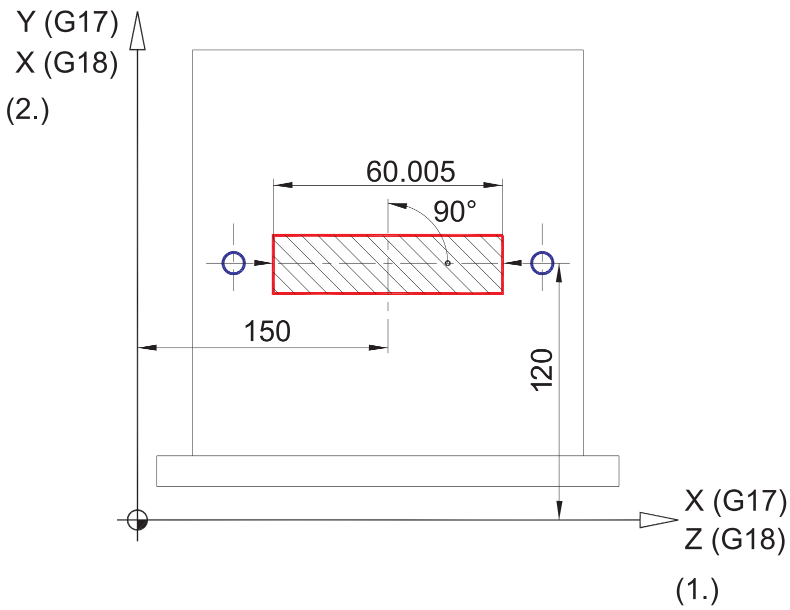
See section
"Standard measuring probe" page 225

See section
"Special measuring probe" page 228

Application example

Calibrate standard measuring probe with L3601:

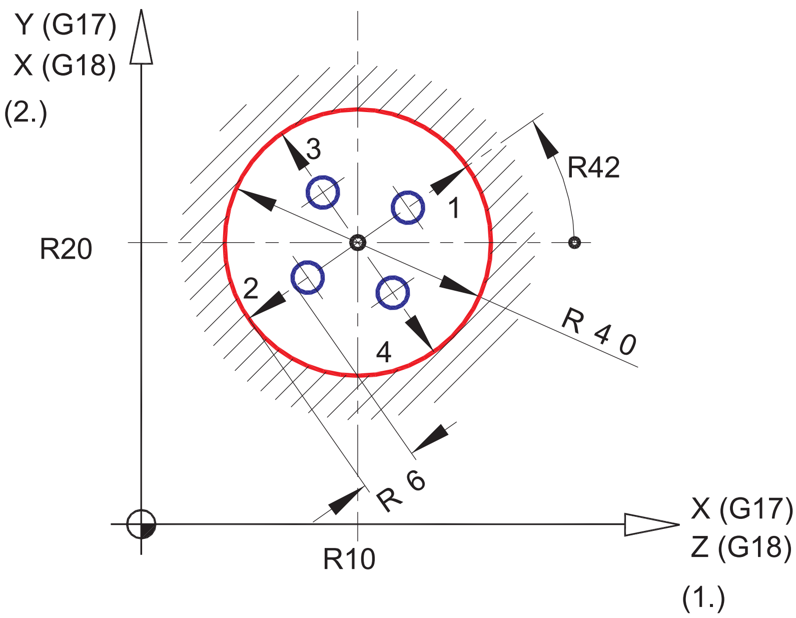
The measuring probe is to be calibrated radially using a gauge block fitted horizontal on the fixture, parallel to the axis and of known width (e.g. exactly 60,005 mm). The position of the centre and the surface should also be known within about 2-3 mm.



8.4.4 - 2 Application example L3601, calibrate

```
N100 ;Fit measuring probe, cutting edge 1 active;
N110 CS_TOOL1("1234",1)
N120 L3000;Standard setting;
N130 R10=150 R20=120 R40=-60.005 R42=90 R0=100
N140 R5=200 R6=4 R65=2 R66=0.005
N150 L3601;Double measurement, compared to 0.005 mm
     ;The calibration value is entered into cutting edge D5 i
N160
```

8.4.4.2 L3602 Calibration, radial with measuring ring



8.4.4 - 3 Cycle diagram L3602

Input parameters

Input parameters and explanation	
R0	Surface Z (G17) / Y (G18)
R1	Measuring depth
R5	Retraction path
R6	Safety distance
R10	1. Coordinate X (G17) / Z (G18)
R20	2. Coordinate Y (G17) / X (G18)

HELLER

Input parameters and explanation	
R40	Exact positive diameter (negative permits calibration on journals)
R42	Angle of inclination
R44	Length to measuring pin pivot point
R65	Repeat value 1, 2
R66	Maximum permissible repeat difference. Standard=0,5 mm
R68	Stop condition on error: 1, 0
R68=1	Stop with M0 if no measurement available and other errors occur
R68=0	No stop for errors, caution with R68=0 Errors then handled using R89!
R70	Measuring feed rate

Output parameters

The calibration value is automatically entered in the cutting edge specified for the calibration value.

See section
"Standard measuring probe" page 225

Output parameters and explanation	
R89	Marker for measurement sequence types 0, 1, 2, 99
R89=0	Measurement OK, result available
R89=1	No measurement available
R89=2	Maximum permissible correction exceeded (R66)
R89=99	Measuring cycle is not finished

Sequence description

The machine traverses to the start position in accordance with the selected approach direction, the spindle is aligned and the measuring probe is switched on. Feed then takes place on the measuring plane (R0-R1), and the measuring position is approached at the measuring feed rate. The repeat difference is compared with the permissible repeat difference R66.

Bore measurement:

If R5=0 is specified, the probe pauses when traversing in the groove on the measuring plane, with R5>0 it is retracted to the safety clearance plus retraction pathR5.

Journal measurement:

Lift off to the safety clearance usually takes place.

Movement then takes place to the withdrawal position.

The calibration value determined is automatically entered into the corresponding cutting edge under address radius 1.

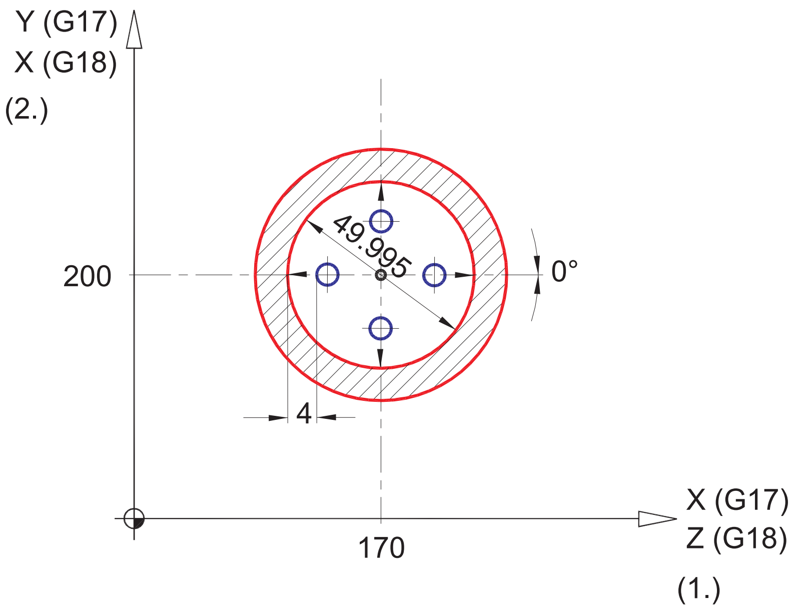
With R68, the machine is usually stopped in the case of an error.

See section
"Standard measuring probe" **page 225**

See section
"Special measuring probe" **page 228**

Application example

The measuring probe is to be calibrated radially using a measuring ring attached to the fixture and parallel to the axis and of known diameter (e.g. exactly 49.995 mm). The position of the centre and the surface should also be known within about 2-3 mm.



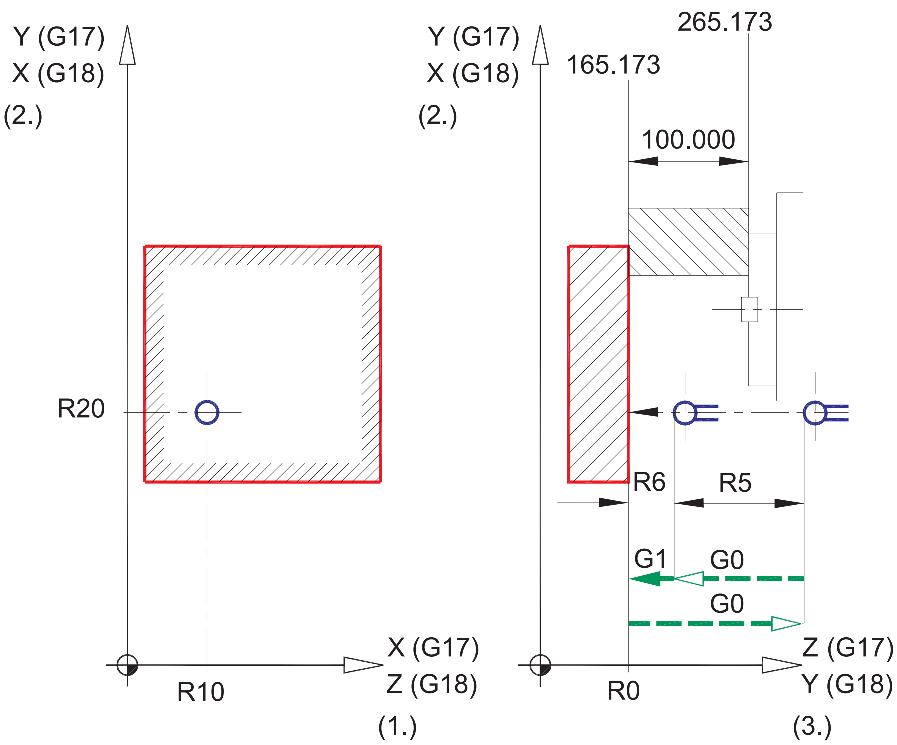
8.4.4 - 4 Application example L3602, calibrate without cycle support

```
N100 ;Fit measuring probe, cutting edge 1 active;
N110 CS_TOOL1("1234",1)
N120 L3000;Standard setting;
N130 R10=170 R20=200 R40=49.995 R42=0 R0=100
N140 R5=200 R6=4 R65=2 R66=0.005
N150 L3602;Double measurement, compared to 0.005 mm
     ;The calibration value is entered into cutting edge D5 i
N160
```

8.4.4.3 L3603 Axial calibration

Note

The surface R0 must be exactly known in advance in order to accurately calibrate the measuring probe axially. If all possible deviations of the surface position (R0) are to be eliminated, a gauge block or other suitable measuring instrument must be used to measure on the front edge of the spindle (diagram below shown with gauge block 100.000)



8.4.4 - 5 Cycle diagram L3603

Input parameters

Input parameters and explanation	
R0	Surface, exact Z (G17) / Y (G18)
R1	Measuring depth (normally R1=10 mm, max. axial displacement is 20 mm)
R5	Retraction path
R6	Safety distance
R10	1. Coordinate X (G17) / Z (G18)
R20	2. Coordinate Y (G17) / X (G18)
R40	Width, +=groove, -=web
R65	Repeat value 1, 2
R66	Maximum permissible repeat difference
R68	Stop condition on error: 1, 0
R68=1	Stop with M0 if no measurement available and other errors occur
R68=0	No stop for errors, caution with R68=0 Errors then handled using R89!
R70	Measuring feed rate

Output parameters

The calibration value is automatically entered in the cutting edge specified for the calibration value.

See section
"Standard measuring probe" **page 225**

See section
"Special measuring probe" **page 228**

Output parameters and explanation	
R89	Marker for measurement sequence types 0, 1, 2, 99
R89=0	Measurement OK, result available
R89=1	No measurement available
R89=2	Maximum permissible correction exceeded (R66)
R89=99	Measuring cycle is not finished

Sequence description

The Z-coordinate must be known exactly to obtain an exact calibration dimension. To do this, a gauge block is normally inserted between the surface and the front edge of the spindle so that it can still be moved slightly. The Z dimension determined in this way is then entered in R0 or in the zero point displacement.

The measuring probe must now be fitted without delay and sub-program L3603 started.

The machine traverses to the start position, the spindle is aligned and the measuring probe is switched on. Feed then takes place to the surface and the measuring point is approached twice axially at measuring feed rate. Movement then takes place to the withdrawal position.

The calibration value determined is automatically entered into the corresponding cutting edge under address length 1.

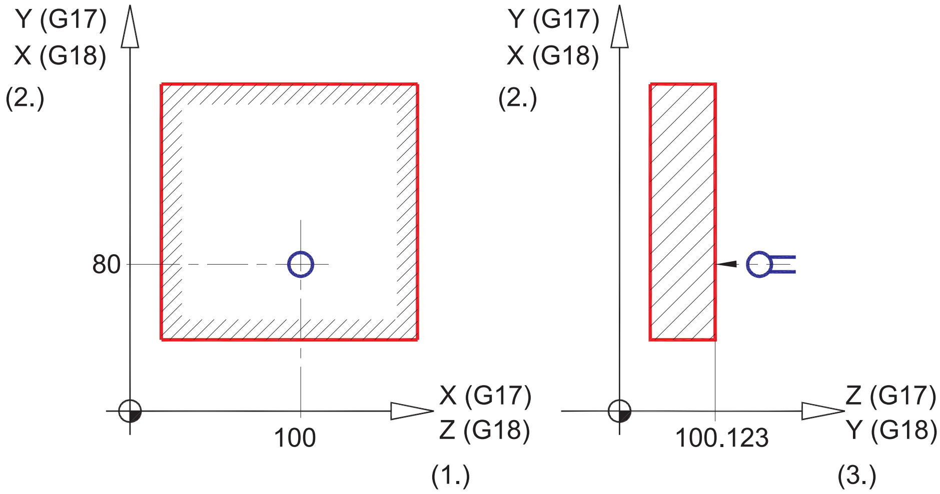
With R68, the machine is usually stopped in the case of an error.

See section
"Standard measuring probe" page 225

See section
"Special measuring probe" page 228

Application example

The measuring probe is to be calibrated axially at a surface of the fixture parallel to the axis, the Z-coordinate of which is known exactly. The position of the surface R0 should be measured exactly, e.g. using a gauge block on the front edge of the spindle.



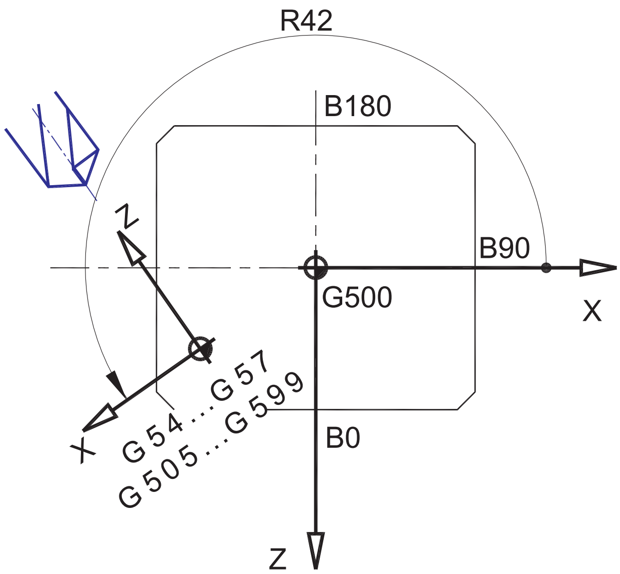
8.4.4 - 6 Application example L3603, calibrate

```
N100 ;Fit measuring probe, cutting edge 1 active;
N110 CS_TOOL1("1234",1)
N120 L3000;Standard setting;
N130 R10=100 R20=80 R0=100.123 R1=10 R5=200
N140 R65=2 R66=0.005
N150 L3603;Double measurement, compared to 0.005 mm
     ;The calibration value is entered into cutting edge D5 i
N160
```

8.5 Sub-programs for calculation

8.5.1 Zero point calculation

8.5.1.1 L5101 Recalculate zero point to B position



8.5.1 - 1 Cycle diagram L5101

Input parameters

Input parameters and explanation	
R42	Rotation angle B
R62	G54...G599 initial zero point
R64	G54...G599 target zero point

Sequence description

The specified initial zero point in the X-, Y-, Z-axis is read. The ZX values are recalculated according to the angle of rotation R42, the Y value is simply accepted. These values are then entered automatically in the target zero point.

The source zero point can be referred to any B position. The angle of rotation is referred to this B position.

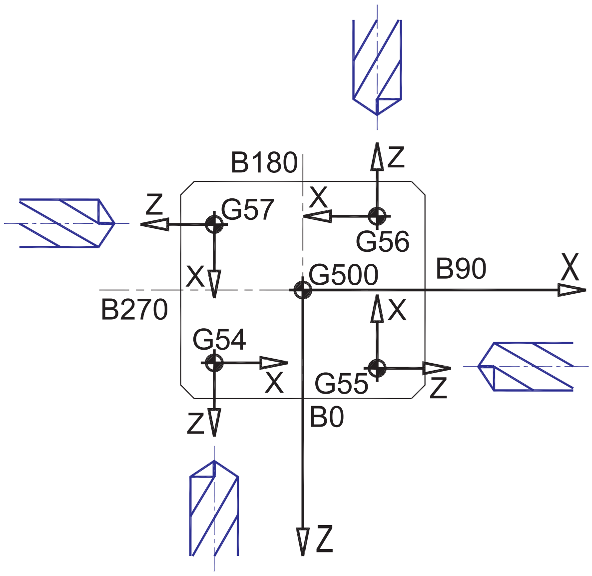
Application example

The initial zero point G56 applies to B position B90. The zero point for B120 is to be calculated and stored in G54.

Programming example

```
N100 R42=30 R62=56 R64=54
N110 L5101;zero point offset from B90-->B120;
```

8.5.1.2 L5102 Zero point correction to B0-B90-B180-B270



8.5.1 - 2 Cycle diagram L5102

Input parameters

Input parameters and explanation	
R12	Correction of zero point in X to B0, no correction=0
R22	Correction of zero point in Y to B0, no correction=0
R32	Correction of zero point in Z to B0, no correction=0
R61	G54...G599 Zero point offset to B0, for example 54, no offset = 0

HELLER

Input parameters and explanation	
R62	G54...G599 Zero point offset to B90, for example 55, no offset = 0
R63	G54...G599 Zero point offset to B180, for example 56, no offset = 0
R64	G54...G599 Zero point offset to B270, for example 57, no offset = 0

Sequence description

A necessary displacement of the zero point in XYZ is first determined in advance. This can be done either manually or with a measuring probe. This displacement refers to B0 and is specified by the values R12, R22, R32.

The specified existing zero points for B0, 90,180, 270 are read, the desired displacement is added with reference to the axes and re-entered. The displacement determined at B0 is thus effective in the other B positions. A Y displacement is transmitted to all zero points.

i

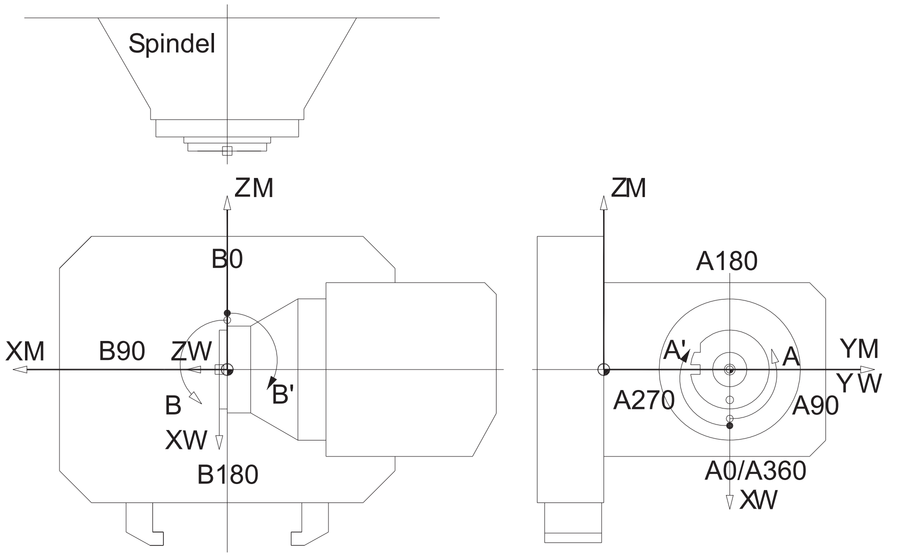
For example, if there is no zero point for B90, its number is specified as 0.

8.5.1.3 L5110 Zero point calculation in 5 axes

General information

Machines with 6 degrees of freedom are necessary for the machining of surfaces, drillings and other form elements in any spatial orientation. Various kinematic systems are in use for this purpose. One of these systems consists of the orthogonally related XYZ axes, an axis B (pallet) rotating about the Y axis and an A axis located on the rotary axis B. The rotary axes B and A in turn have an orthogonal relationship to each other.

Arrangement of axes



8.5.1 - 3 L5110 Arrangement of axes

The system consists of a standard XYZB machining centre and a bolt-on rotary attachment RLE (A axis). The A axis rotates about the X axis in the zero position of the B axis. The holding and clamping of the workpiece takes place using a face plate or steep taper holder or similar mechanical components.

Sub-program L5110 deals with alignment in any desired direction. The calculated values are stored, if required, directly in the active zero point displacement G54...G57, G505...G599..

Zero point calculation with L5110

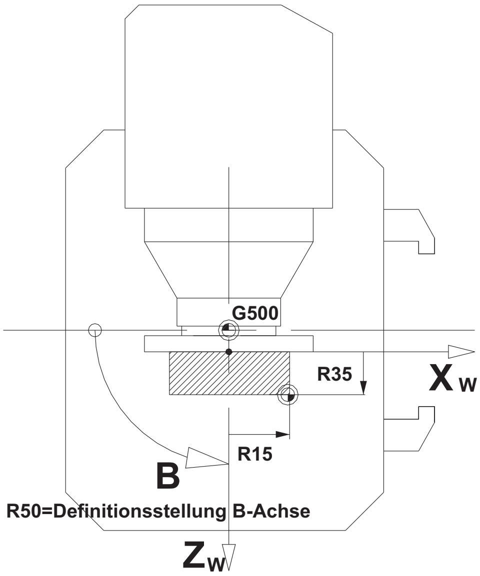
The machining zero point is specified in XYZ components in the workpiece zero position, in other words in the as yet un-rotated coordinate system. The XY origin is the centre of the face plate, the Z origin is the surface of the face plate. After calculating the rotary movements necessary for alignment, this zero point is transformed into the machining position and entered with components X, Y, Z, A, B in any desired active zero point displacement G54...G599.

To be able to deal flexibly with different configuration situations, the definition positions of the A and B axes can be specified.

This is defined as follows: An A position and a B position are specified at which the workpiece coordinate system XwYwZw is aligned identically to the machine coordinate system XmYmZm. This is very helpful if the clamping elements between the face plate and the workpiece produce a rotary displacement from the outset. This allows the XwYwZw system to be rotated in the "zero position".

Definition position of the B-axis

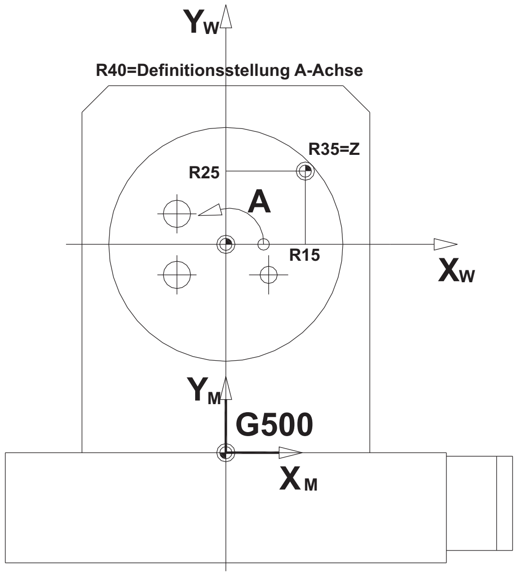
The definition position of the B axis permits a rotated clamping in the A axis to be eliminated in the B axis. The above system generally requires the definition position to be R50=90 ° degrees (in other words at B90, Zw is normal to the face plate and parallel to the Zm axis in the zero position).



8.5.1 - 4 R50 Definition position of the B axis

Definition position of the A axis

Rotated clamping on the face plate can be eliminated with the definition position of the A axis. Ideally, the definition position is specified with R40=0 (in other words at A0, the Wx axis is parallel to the Xm axis).



8.5.1 - 5 R40 Definition position of the A axis

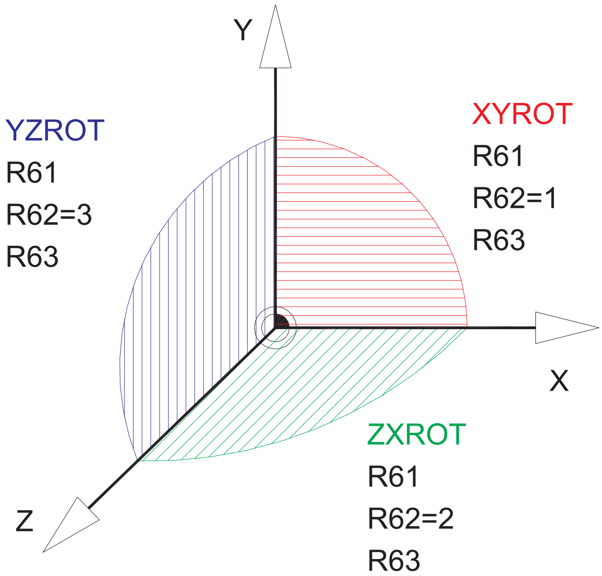
Input parameters and explanation	
R15	Zero point in X
R25	Zero point in Y
R35	Zero point in Z direction

Aligning the coordinate system with L5110

The alignment of the spatial Cartesian coordinate system can take place with up to three successive rotations each with a choice of rotation planes XYROT, ZXROT, YZROT. If only one/two rotations are desired, the remaining rotations are specified with "0" angle of rotation. The control dialogue also contains the following graphics for aligning the system.

1. Rotation is specified in R61.
2. Rotation is specified in R62.
3. Rotation is specified in R63.

Choice of rotation plane:



8.5.1 - 6 Selection of the rotation plane L5110

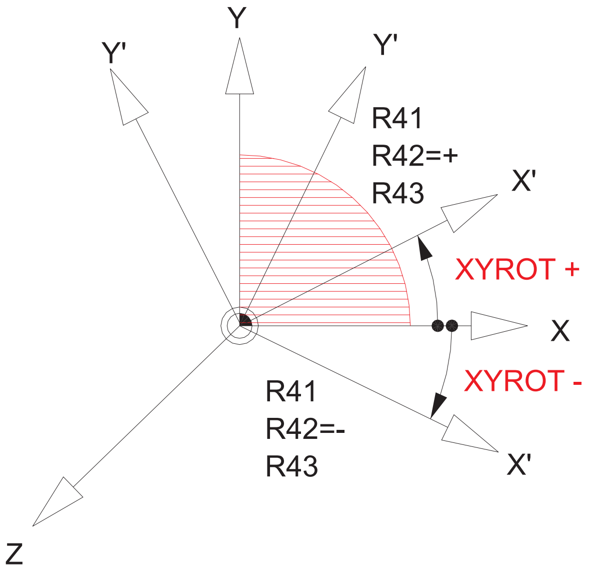
Input parameters for angle of rotation

R41 Angle of rotation for the 1st rotation, optionally as XY, ZX, YZROT

R42 Angle of rotation for the 2nd rotation, optionally as XY,ZX,YZROT

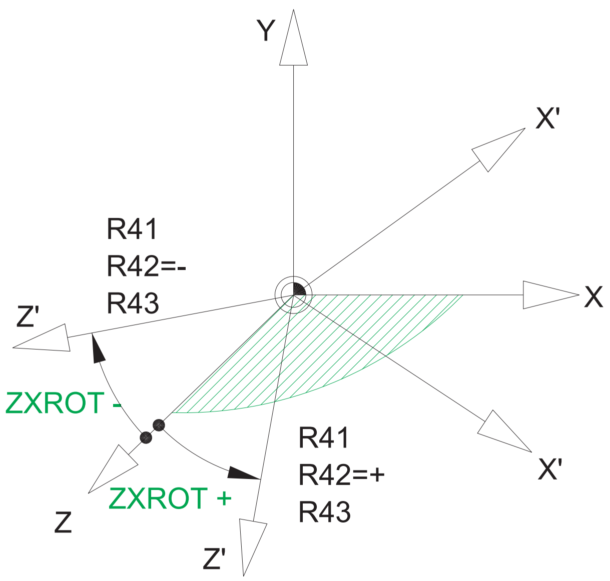
R43 Angle of rotation for the 3rd rotation, optionally as

Entry of angle of rotation as XYROT



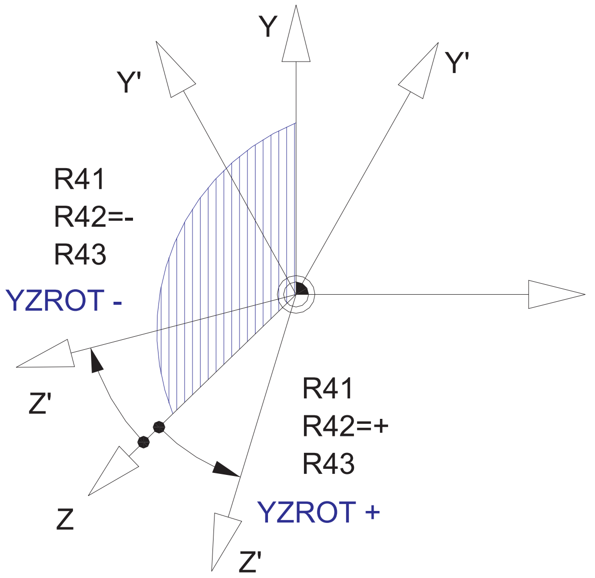
8.5.1 - 7 L5110 Rotation angle XYROT

Entry of angle of rotation as ZXROT



8.5.1 - 8 L5110 Rotation angle ZXROT

Entry of angle of rotation as YZROT



8.5.1 - 9 L5110 Rotation angle YZROT

Preferred position R44

Since alignment can generally take place with 2 possible solutions, R44 is used to choose a preferred setting. If R44 is selected with +1, this chooses the positive A rotation; if R44 is specified with -1, this chooses the negative A rotation. In principle, both solutions are equally valid. In the case of extremely large dimensional and angular ratios, the choice can be determined by, for instance, the stroke ratios (X stroke) of the machine.

i

To prevent an accidental entry in the active zero point displacement, a confirmation prompt is issued with R60 asking whether the calculated zero point should really be entered. It must be ensured that the desired zero point displacement G54...G599 to be entered is active before starting the calculation program. Otherwise, an existing zero point displacement would be overwritten.

The XY angle of rotation required for the machining that follows is stored in parameter R80. The programmed machining must then take place rotated in the XY plane by the angle R80 XYROT. This angle arose from the fact that the desired (rotated) workpiece Zw axis was forced to be parallel to the Zm axis (drilling axis, spindle).

At the end of the program, the zero point in the X, Y, Z, A, B directions is written to the zero point data. However, alignment of the axes only takes place when the A and B axes are positioned (for example G0 G54 A0 B0).

This means, shown diagrammatically, that the tool tip would touch the defined zero point in the aligned position with a positioning G54 X0 Y0 Z0 A0 B0.

All subsequent machining positions must be rotated in R80 with respect to the zero position by the angle XYROT, that is also calculated. This can be done using a ROT, AROT programmed rotation.

Several different alignments in sequence

Application

If a machining process requires several different alignments, either the alignment can be recalculated using L5110 or a single calculation takes place at the start of the program to calculate the alignments in sequence for entry into various zero points G54...G599. The applicable XY rotation angle R80 must, however, then be stored in a suitable manner.

In the following example, it suffices to call the appropriate zero point displacement at a suitable point. It should be noted that the XY angle of rotation R80 must also be stored in a suitable manner.

Example

DEF REAL Alignment 1, alignment 2, alignment 3

...

Alignment 1: R80 ==> Alignment 1

Alignment 2: R80 ==> Alignment 2

Alignment 3: R80 ==> Alignment 3

...

Movement of the AB-axes only occurs when commanded with for example G0 G54 A0 B0!

Commissioning L5110

Before using L5110, the sub-program must know the real data of the axis system. Correction constants are defined for this purpose from block N1970. These are filled with pseudo values on shipment and must be adapted to the real conditions.

Commissioning parameters

Parameters	Explanation
N1980 LV_DAX	Specifies the X position of rotary axis A in G500 machine coordinates. For this, the face plate must be facing the machine spindle, this means: A-axis and Z-axis on the machine are parallel. The face plate should be checked using a dial gauge for planar accuracy and corrected if necessary. After achieving the necessary planar accuracy, the centre of the face plate is circled and the G500 X position is entered in LV_DAX. With fault-free assembly of the A-axis, LV_DAX=0 must be valid.
N1990 LV_DAY	The circled position is entered for LV_DAX in Y in a similar manner to LV_DAY in X.
N2000 LV_DAZ	Specifies the Z surface of the face plate in G500 Z. To determine this, it makes sense to fit a gauge block between the face plate and the front edge of the (empty) spindle and read off the Z position on the CNC (minus gauge block). If the face plate is in front of the centre of the pallet, then LV_DAZ is positive.
N2010 LV_WFA	An angular error of the A axis to the zero position of the face plate can be eliminated by this. With correct adjustment of the A-axis, LV_WFA=0.
N2010 LV_WFB	This permits the elimination of an angular error in the B axis of the rotary attachment RLE clamped on pallet B. If, for example, a deviation of the B axis is discovered after checking the face plate with a dial gauge (see above), this value must be entered here or corrected mechanically (determine indexing play or similar). If the adjustment is correct, LV_WFB=0.
Spatial correction	After the above values have been entered correctly, mechanical deviations are calculated correctly for any desired spatial alignment, in other words they are taken into account for every transformation.

Chapter 9

Error messages

9 Error messages

9.1 NC error messages

9.1.1 66xxx alarm messages - with movement stop

9.1.1.1 66001 - programmed tool not equal to spindle tool

Explanation

Block search run is used in an attempt to enter a new NC block. The tool located in the spindle at the start of the block search is different than the one required for the spindle according to the entry point destination of the NC program.

Response

Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.
The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy

Delete alarm using RESET key.
Insert the correct tool for the program position via MDA.

9.1.1.2 66002 - Call up parameter for the HELLER-system cycle %4 it incorrect.

Explanation

The system cycle ... was called up with a call-up parameter containing an undefined value.

Response

Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.

The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy

Delete alarm using RESET key.
NC program correction Restart part program.

9.1.1.3 66003 - Referencing sequence of the axis: %4 double allocation

Explanation

The sequence of the axes during referencing/home position drive is defined via the axis specific machine date $MA_REFP_CYCLE_NR.
The axis %4 was allocated the sequence number twice.

Response

Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.
The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy

Correct the machine data $MA_REFP_CYCLE_NR.
Run NCK reset.

9.1.1.4 66004 - Internal HELLER system error. Error code: %4

Explanation

The error was caused by HELLER system cycles without user influence.
Error code:
10 ASUP-call up with invalid code.

Response

Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.
The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy

Contact HELLER Service

9.1.1.5 66005 - Incorrect value in parameter %4 with HELLER cycle call-up

Explanation

A HELLER-system cycle CS_xxxx was called up with an incorrect parameter.
%4 specifies the number of the parameter.

Response

Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.
The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy

Correct the cycle call up.

9.1.1.6 66045 - Call up parameter for the HELLER-system cycle %4 it incorrect.

Explanation

The system cycle ... was called up with a call-up parameter containing an undefined value.

Response

Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.
The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy

Correct the cycle call up.

9.1.1.7 66070 - Incorrect tool length. Tool: %4

Explanation

An impermissible deviation from the setpoint value was detected when the tool lengths (TC_DP3+TC_DP12+$TC_DP21) of the tool %4 were checked with the HELLER system cycle CS_TCHCK.

Response

Display.

Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.
The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy Check tool data.

9.1.1.8 66071 - Incorrect tool length2. Tool: %4

Explanation An impermissible deviation from the setpoint value was detected when the tool lengths2 (TC_DP4+TC_DP13+$TC_DP22) of the tool %4 were checked with the HELLER system cycle CS_TCHCK.

Response Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.
The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy Check tool data.

9.1.1.9 66072 - Incorrect tool radius. Tool: %4

Explanation An impermissible deviation from the setpoint value was detected when the tool radius (TC_DP6+TC_DP15) of the tool %4 were checked with the HELLER system cycle CS_TCHCK.

Response Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.
The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy Check tool data.

9.1.1.10 66075 - Tool type cannot be checked, tool: %4

Explanation

The tool %4 cannot be checked with the HELLER system cycle CS_TCHCK.
The tool type is incorrect.

Response

Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.
The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy

Check tool data.

9.1.1.11 66100 - SBBK: no reference value found for tool %4

Explanation

Response

Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.
The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy

Delete alarm using RESET key.
NC program correction Restart part program.

9.1.1.12 66101 - SBBK is requested in the NC-Program. However, the user - tool date has not been entered.

Explanation

The SBBK is requested via tool user data. $TC_TPC7 must be 1 to ensure that the tool is scanned with SBBK. An error occurs if a measurement with GV_SBBK is requested in the NC program but the tool was not defined for SBBK according to the tool date. GV_SBBK is only available based on its compatibility with MCi/MCH and has no function of its own.

Response

Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.
The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy

Delete alarm using RESET key.
NC program correction Restart part program.

9.1.1.13 66102 - Oversize tool should be positioned outside of the traversing range: %4

Explanation

Prior to the tool change a position was assigned with GV_X and/or GV_Y that cannot be approached with the new tool based on its swing diameter.

Response

Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.
The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy

Delete alarm using RESET key.
NC program correction Restart part program.

9.1.1.14 66103 - NC-run-up was not run through successfully.

Explanation

The NC must execute diverse functions and settings during run-up after an NCK reset. The run-up process was either terminated prior to completion or not initiated.

Response

Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.

The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy
Check the machine date $MC_PROG_EVENT_MASK.

9.1.1.15 66104 - Status %4 of the tool changer not defined

Explanation

Response
Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.
The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy
Delete alarm using RESET key.
NC program correction Restart part program.

9.1.1.16 66105 - Magazine configuration is not loaded. Run-up not executed.

Explanation
During run-up it was detected that the magazine configuration is not loaded. The controller is unable to manage tools if the magazine in not configured.

Response
Display.
Entry in the log file.
Interface signals are set.
NC start disable in this channel.
Run-up aborted.

Remedy
Select "load magazine configuration" in the HELLER system functions.

9.1.1.17 66106 - Incorrect definition of the spindle envelope contour

Explanation

The NC must recognize the envelope contour of the spindle tool for diverse inspections. In this case it was not defined correctly.
The envelope contour is stored in the user tool data base:
$TC_TPC4 diameter $TC_TPC5 length - if no user tool parameter was defined then the system will attempt to use cutting edge 1 as envelope data.

Response

Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.
The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy

Delete alarm using RESET key.
Correct tool data.

9.1.1.18 66107 - Traversing range limitations cannot be used for modulo axes. Axis: %4

Explanation

A modulo axis cannot be restricted by using the HELLER traversing range restrictions. Software limit switches have no effect on modulo axes.

Response

Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.
The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy

Delete alarm using RESET key.
NC program correction Restart part program.

9.1.1.19 66108 - tool management inconsistent.

Explanation

Not all of the NC ordered corrections were acknowledged after the tool change was terminated. The tool management could contain wrong data.

Response

Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.
The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy

Delete alarm using RESET key.
Check the tool management and correct if necessary.
Issue approval for the current status of the tool management in the HELLER system functions. (the channel specific variables SYG_IM[12] and SYG_IM[13] must contain -1 if the tool management is correct).

9.1.1.20 66109 - pallet change position is not approached by the user cycle.

Explanation

The user specific cycle US_PAW_ANF.SPF was used during the pallet change for approaching the pallet change position. However, the pallet change position was not reached. The following conditions must be met:

- Z-axis on GD_PAW_POS[2].
- B-axis on GD_PAW_POS[4].
- Tool length shorter than the max. permissible tool length for pallet change (GD_PAW_MAXWZL).
- X-axis outside the collision area.

Response

Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.
The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy

Configure the user specific cycle US_PAW_ANF.SPF to ensure that the pallet change position can be approached properly.

9.1.1.21 66110 - Run time monitoring tool change door open

Explanation

Response

Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.
The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy

Delete alarm using RESET key.
NC program correction Restart part program.

9.1.1.22 66111 - %4 contains incorrect value

Explanation

During run-up it was determined that the specified machine date contains an incorrect value. The control unit cannot continue to work with this value.

Response

Display.
Entry in the log file.
Interface signals are set.
NC start disable in this channel.
Control run-up is aborted.

Remedy

Provide the specified machine date with a correct value.

9.1.1.23 66112 - Tool should be positioned outside of the traversing range: %4

Explanation

Prior to the tool change a position was assigned with GV_X and/or GV_Y that cannot be approached.

Response

Display.

Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.
The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy

Delete alarm using RESET key.
NC program correction Restart part program.

9.1.1.24 66113 - Incorrect definition of the preparation tool envelope contour

Explanation

The NC must recognize the envelope contour of the spindle tool for diverse inspections. In this case the envelope contour of the preparation tool was incorrectly defined.
The envelope contour is stored in the user tool data base:
$TC_TPC4 diameter
$TC_TPC5 length
If no user tool parameter was defined then the system will attempt to use cutting edge 1 as envelope data.

Response

Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.
The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy

Delete alarm using RESET key.
Correct tool data.

9.1.1.25 66114 - All tools must be fixed location coded

Explanation

Response

Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.

The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy

Delete alarm using RESET key.
NC program correction Restart part program.

9.1.1.26 66115 - Error during preparation: No tool %4 ready for operation

Explanation

Preparation for tool %4 could not be executed. No tool is available with the identifier %4 that is enabled and not disabled. Machining can continue until a change command is executed for this tool.

Response

Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.
The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy

Delete alarm using RESET key.
NC program correction Restart part program.

9.1.1.27 66116 - Preparation (T) was not programmed prior to M6

Explanation

Preparation must be programmed in advance of the tool change with M6.

Response

Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.
The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy

Delete alarm using RESET key.
NC program correction Restart part program.

9.1.1.28 66117 - Active frame contains reflecting, scaling, or rotating

Explanation

The frame that is active during tool change may not contain any reflection, scaling, or rotating portions.

Response

Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
Movement stop if the message is set off in the main process.
The machine continues to run until the interpreter is empty if the message comes from a synchronous action.

Remedy

Delete alarm using RESET key.
NC program correction Restart part program.

9.1.1.29 66118 - Label for alternative strategy was not programmed

Explanation

Using the alternative strategy, the jump destination where processing is to be continued must be specified by CS_ESCLAB.

9.1.1.30 66119 - Tolerance for SBBK outside value range

Explanation

A tolerance for SBBK (fast tool break monitoring) outside the permissible value range has been programmed (0.1<=GV_SBBK_TOL<=10) or assigned in the machine data (1<= $MN_USER_DATA_INT[160]<=100).

9.1.1.31 66121 - Error on accessing file %4

Explanation

Access to the specified file could not be executed (reading or writing).
Possible causes:
The file is currently in use or opened.
No storage space available.

9.1.1.32 66122 - Invalid or inadmissible call-up parameter

Explanation

Parameters, specified when a cycle was called up, are inadmissible or invalid.

9.1.1.33 66123 - Magazine side to be measured not calibrated

Explanation

The magazine side to be measured has not been calibrated yet.

Remedy

Run MS_WMR_ABGLEICH at reference location.

9.1.1.34 66124 - HMI/Camera not ready for measuring

9.1.1.35 66125 - PLC not ready for measuring

9.1.1.36 66126 - Measurement invalid

9.1.1.37 66127 - Calibrated location does not correspond to programmed location

Explanation

The position of the specified magazine place (reference) does not correspond to the loader position.

9.1.1.38 66128 - Camera axes deviate too far from loader axes

Remedy

Align camera housing/retaining angle with the X-axis of the loader.

9.1.1.39 66129 - Spindle not empty

Remedy

Perform empty change.

9.1.1.40 66130 - Rotary station not empty

Remedy

Empty rotary station.

9.1.1.41 66131 - Selected tool not suitable for pocket check

Remedy

Select tool with location type 1 and tool size 1 at top and tool size 1 at bottom.

9.1.1.42 66132 - Traversing range restriction cannot be set. Reason: %4

Explanation

The axis to be restricted is located at a position outside the range to which it is to be restricted.

9.1.1.43 66133 - Active axis zero point shift: %4 contains inadmissible values

Explanation

The entered axis contains inadmissible zero point shifts.

Remedy

Tool changer axes and rack magazine axes must not contain zero point offsets.

9.1.1.44 66134 - Cycle cannot be started in channel %4

Remedy

NC program correction

9.1.1.45 66135 - A/C axis positioning violates collision paths

Explanation

An attempt is made to swivel the C-head to a position at which the C-head collides with the machine.

Remedy

A position on X, Y and Z must first be approached, at which the C-head can be swivelled without collision. See AZ drawing.

9.1.1.46 66136 - Tool requested for manual pick-up is already in magazine

Remedy

The tool requested for manual pick-up via CS_THPU is already in the magazine. Tools for manual pick-up must not already be loaded into the magazine.

9.1.1.47 66137 - Tool to be replaced is not a pick-up tool

Remedy

The CS_THPU cycle has been called up to empty the spindle. CS_THPU can be used to remove from the spindle only those tools that were loaded as pick-up tools.

9.1.1.48 66138 - Spindle tool loaded via manual pick-up

Explanation

Spindle tool loaded via manual pick-up. The tool must be unloaded using CS_THPU.

Remedy

Tools loaded via manual pick-up must not be unloaded using M6. These tools must be unloaded using CS_THPU.

9.1.1.49 66250 - Machine data for facing slide tool %4 not entered!

Explanation

Machine data for facing slide tool are not entered!
See list of facing slide types!

9.1.1.50 66300 - Error on tool movement MVTOOL Code: %4

Explanation

An unwanted situation occurred in the rack-type magazine when tool data was resaved.

9.1.1.51 66301 - WMR: rack-type magazine not available. Measuring mode is active.

Explanation

The rack-type magazine cannot execute any jobs at present. It is in measuring mode.

9.1.1.52 66600 - %5: Enter point angle 1-179 degrees

Remedy

NC program correction

9.1.1.53 66601 - %5: Enter tool radius > 0

Remedy

NC program correction

9.1.1.54 66602 - %5: Enter point angle 1-180 degrees

Remedy

NC program correction

9.1.1.55 66603 - %5: Maximum permissible centre correction exceeded!

Remedy NC program correction

9.1.1.56 66604 - %5: No measurement executed!

9.1.1.57 66605 - %5: STOP, unexpected measurement!

9.1.1.58 66606 - %5: No msmt. possible with active inversion and rotation!

9.1.1.59 66607 - %5: Cycle cannot be executed with active inversion and rotation!

9.1.1.60 66608 - %5: Errors: Tool > bore hole!

Remedy NC program correction

9.1.1.61 66609 - %5: Errors: Unmach. diameter > mach. diameter!

Remedy NC program correction

9.1.1.62 66610 - %5: Maximum permissible diameter correction exceeded!

Remedy

NC program correction

9.1.1.63 66611 - %5: Maximum permissible repeat difference exceeded!

Remedy

NC program correction

9.1.1.64 66612 - %5: Msmt. probe not clbrd axially!

9.1.1.65 66613 - %5: Maximum permissible correction exceeded!

Remedy

NC program correction

9.1.1.66 66614 - %5: Maximum permissible width correction exceeded!

Remedy

NC program correction

9.1.1.67 66615 - %5: Maximum permissible centre offset R6 exceeded!

Remedy

NC program correction

9.1.1.68 66616 - %5: Maximum permissible diameter correction R83 exceeded!

Remedy

NC program correction

9.1.1.69 66617 - %5: Maximum permissible diameter correction R83 exceeded!

Remedy

NC program correction

9.1.1.70 66618 - %5: Tool radius < = 0!

Remedy

NC program correction

9.1.1.71 66619 - %5: R47 not defined!

Remedy

NC program correction

9.1.1.72 66620 - %5: Unfinished diameter < pin diameter!

Remedy

NC program correction

9.1.1.73 66622 - %5: Nominal diameter < core diameter!

Remedy

NC program correction

9.1.1.74 66623 - %5: Tool diameter < unmachined diameter+R6!

Remedy
NC program correction

9.1.1.75 66624 - %5: Incorrect starting point R61 1....4

Remedy
NC program correction

9.1.1.76 66625 - %5: Wrong milling mode R60!

Remedy
NC program correction

9.1.1.77 66626 - %5: Apply sub-program L63!

Remedy
NC program correction

9.1.1.78 66627 - %5: R1 must be > R3!

Remedy
NC program correction

9.1.1.79 66628 - %5: R1 must be > R8!

Remedy
NC program correction

9.1.1.80 66629 - %5: Bore depth too small!

Remedy
NC program correction

9.1.1.81 66630 - %5: Wrong angle R42, should be < +-90 degrees!

Remedy

NC program correction

9.1.1.82 66631 - %5: Wrong angle R42, should be <> 0 degrees!

Remedy

NC program correction

9.1.1.83 66632 - %5: Enter maximum permissible speed!

Remedy

NC program correction

9.1.2 67xxx alarm messages - only display

9.1.2.1 67003 - Traversing range restriction violated. Clear the axes with JOG.

Explanation

The working area is limited if an oversize tool is in the spindle. Each spindle movement is obstructed if the machine is located outside of the limited working area.

Response

Display.
Entry in the log file.
Spindle movement is obstructed.

Remedy

Clear the working area with the HELLER system functions.
Position the axes in the permissible working area.

9.1.2.2 67004 - Online temperature compensation: Compensation value too high. Compensation limited to maximum.

Explanation

The online temperature compensation function calculates a compensation value that is greater than a set threshold. The compensation value has been frozen at this threshold and does not increase. As soon as the calculated compensation value drops below the set threshold, the compensation is executed again online.

9.1.2.3 67005 - Magazine place status or tool status invalid, location %4

Explanation

Magazine place status or tool status have invalid values.

Remedy

Correct data: blocked, reserved, free, for loading, for unloading and in change.

9.1.2.4 67006 - Traversing range restrictions C-head/tilting head switched off

Explanation

9.1.2.5 67021 - Diagnostics area: Instruction from the NC program

Explanation

The messages area displays additional information.

9.1.2.6 67022 - Diagnostics area: Errors from the NC program

Explanation

The messages area displays additional information.

9.1.2.7 67101 - Spindle tool was disabled because of IPM overload

Explanation

IPM overload was triggered during the preceding machining. The spindle tool was disabled during the block search run.

Response

Display.
Entry in the log file.
NC continues to run.

Remedy

9.1.2.8 67102 - Position of the tool changer axis CT1 in the current state: %4 of the tool changer is not correct

Explanation

The position of the TC axis is different from the one that is currently allowed.
Possible causes:

- Axes TC/TZ were not positioned using setup functions or tool change command.
- Axes TC/TZ have violated the specified path (collision).

Response

Display.
Entry in the log file.
NC continues to run.

Remedy

Bring the changer in the home position with setup functions.
Automatic home position movement.

9.1.2.9 67103 - Position of the tool changer axis ZT1 in the current state: %4 of the tool changer is not correct

Explanation

The position of the TC axis is different from the one that is currently allowed.
Possible causes:

- Axes TC/TZ were not positioned using setup functions or tool change command.
- Axes TC/TZ have violated the specified path (collision).

Response

Display.
Entry in the log file.

NC continues to run.

Remedy

Bring the changer in the home position with setup functions.
Automatic home position movement.

9.1.2.10 67104 - Tool change position X/Y not approached

Explanation

The tool changer is not in the home position and the X and/or Y axis is not on the tool change position.

Response

Display.
Entry in the log file.
NC continues to run.

Remedy

Bring the changer in the home position with setup functions.
Automatic home position movement.

9.1.2.11 67105 – Spindle not aligned

Explanation

The tool changer is not in the home position and the spindle is not on the tool change position.

Response

Display.
Entry in the log file.
NC continues to run.

Remedy

Bring the changer in the home position with setup functions.
Automatic home position movement.

9.1.2.12 67106 - Tool change door not open

Explanation

The tool changer is not in the home position and the tool change door is not opened.

Response

Display.
Entry in the log file.
NC continues to run.

Remedy

Setup functions tool change door open.
Automatic home position movement.

9.1.2.13 67107 - Tool in spindle not released

Explanation

The tool changer is not in the home position and the collet chuck is not released. However, it must be released in the current status of the tool changer.

Response

Display.
Entry in the log file.
NC continues to run.

Remedy

Release the setup functions collet chuck.
Automatic home position movement.

9.1.2.14 67108 - Tool in spindle not clamped

Explanation

The tool changer is not in the home position and the collet chuck is not clamped. However, it must be clamped in the current status of the tool changer.

Response

Display.
Entry in the log file.
NC continues to run.

Remedy

Use the setup functions to clamp the collet chuck.
Automatic home position movement.

9.1.2.15 67109 - Status %4 of the tool changer not defined

Explanation

Status of the tool changer is unknown.

Response

Display.
Entry in the log file.
NC continues to run.

Remedy

Contact HELLER Service

9.1.2.16 67110 - Error during preparation: No tool %4 ready for operation

Explanation

Preparation for tool %4 could not be executed. No tool is available with the identifier %4 that is enabled and not disabled. Machining can continue until a change command is executed for this tool.

Response

Display.
Entry in the log file.
NC continues to run.

Remedy

9.1.2.17 67111 - Incorrect definition of the spindle tool envelope contour

Explanation

The NC must recognize the envelope contour of the spindle tool for diverse inspections. In this case it was not defined correctly.
The envelope contour is stored in the user tool data base:
$TC_TPC4 diameter
$TC_TPC5 length
If no user tool parameter was defined then the system will attempt to use cutting edge 1 as envelope data.

Response

Display.
Entry in the log file.
NC continues to run.

Remedy

Delete alarm using RESET key.
Correct tool data.

9.1.2.18 67112 - tool management inconsistent. Unknown T-number

Explanation

The NC contains information for tool management recovery that cannot be correct.

Response

Display.
Entry in the log file.
NC continues to run.

Remedy

Delete alarm using RESET key.
Check the tool management and correct if necessary.

Issue approval for the current status of the tool management in the HELLER system functions.
(The channel specific variables SYG_IM[12] and SYG_IM[13] must contain -1 if the tool management is correct.)

9.1.2.19 67113 - Tool data was not rewritten

Explanation

The tool change was terminated. During the reset that follows, the tool data must be rewritten by the system to ensure that the tool management buffer and the actual configuration of the tool coincide.

Response

Display.
Entry in the log file.
NC continues to run.

Remedy

9.1.2.20 67114 - last magazine place was disabled

Explanation

The last magazine place must be disabled for chain magazines since the cartridge for the spindle tool occupies this location.

Response

Display.
Entry in the log file.
NC continues to run.

Remedy

9.1.2.21 67115 - Tool specific speed > maximum speed. Speed limited to machine specific limit speed.

Explanation

The tool specific limit speed ($TC_TPC2) of the spindle tool is greater than the maximum speed of the spindle. Therefore the speed is limited to the machine specific limit speed.

Response

Display.
Entry in the log file.
NC continues to run.

Remedy

Correct the $TC_TPC2 of the spindle tool.

9.1.2.22 67116 - Tool broken: %4

9.1.2.23 67117 - NC-Stop through monitoring event: Replacement tool missing

9.1.2.24 67118 - NC-Stop through monitoring event: Tool monitoring

9.1.2.25 67119 - NC-Stop through monitoring event: SBBK tool broken: %4

9.1.2.26 67120 - NC stop caused by user-defined monitoring event

9.1.2.27 67121 - No monitoring event detected

9.1.2.28 67122 - Attention - Program continued

9.1.2.29 67123 - Caution! Alternative strategy has responded Branch destination: %4

Explanation

The alternative strategy has responded. The program run has been cancelled and will be continued at the jump destination.

9.1.2.30 67124 - The magazine place user data do not contain 2 places without cartridge

Explanation

More than 2 or less than 2 places without cartridge have been defined in the magazine place user data (column K) in the magazine list.

Remedy

The machine must contain exactly 2 places without cartridges. These places must be in magazine1 (chain) or at the provisioning place.
0=no cartridge at magazine place
1=cartridge at magazine place
3=cartridge at magazine place is return tool cartridge

9.1.2.31 67125 - The magazine place user data $TC_MPPC2 contain more than 2 places with return cartridges

Explanation

More than 2 places for return cartridge have been defined in the magazine place user data (column K) in the magazine list.

Remedy

The machine can contain exactly 2 places without cartridges. These places must be in magazine1 (chain) or at the provisioning place.
0=no cartridge at magazine place
1=cartridge at magazine place
3=cartridge at magazine place is return tool cartridge

9.1.2.32 67126 - Runtime reset handling

Explanation

The PLC did not react within 5 seconds during reset handling.

9.1.2.33 67127 - Envelope contour of tool: %4 incorrectly defined

Explanation

The NC must recognize the envelope contour of the tool %4 for various inspections.

Remedy

In this case it was not defined correctly. The envelope contour is stored in the OEM tool data base:
$TC_TPC4 Diameter
$TC_TPC5 length
If no OEM tool parameter has been defined, the system will attempt to use cutting edge 1 as envelope data.

9.1.2.34 67128 - WMR: Provisioning place is occupied

9.1.2.35 67129 - WMR: Magazine B-axis not in home position

9.1.2.36 67130 - WMR: Provisioning place is not occupied

9.1.2.37 67131 - WMR: Magazine place %4 is not occupied

9.1.2.38 67132 - WMR: Magazine place %4 is occupied

9.1.2.39 67133 - WMR: Invalid magazine number %4

9.1.2.40 67134 - WMR: Invalid place number %4

9.1.2.41 67135 - WMR: No return place for tool at provisioning place

9.1.2.42 67136 - WMR: Tool transport failed: %4

9.1.2.43 67137 - WMR: Maximum tool length exceeded (MagNo, LocNo, WZL): %4

9.1.2.44 67138 - Tool data absent or tool not inserted (place: %4). magazine movement disabled

Explanation

The following was detected on inspecting the magazine:

- A tool is entered at place %4, but the bit: "Tool at magazine place" not set.
- No tool is entered at place %4, but the bit: "Tool at magazine place" is set.

9.1.2.45 67139 - WMR: Tool management and presence switch in loader are inconsistent

Explanation

Two possible scenarios:
- The tool presence switch of the loader signals "Tool present in loader". The magazine place data for magazine place 9998.4 (loader), however, contains no tool.
- The loader's tool presence switch does not signal "Tool present in loader". The magazine place data for magazine place 9998.4 (loader), however, contains a tool.

9.1.2.46 67140 - WMR: Status variables invalid

Explanation

Status SYG_IM[19] contains undefined value,
or magazine SYG_IM[20] and/or place SYG_IM[21] do not match the status.

9.1.2.47 67141 - WMR: Tool depositing was cancelled. Loader not on specified path

Explanation

Transportation of a tool by the the loader cancelled. The loader's position no longer matches the path to be travelled. Automatic clear traversing is therefore impossible.

9.1.2.48 67142 - WMR: Loader in provisioning area

Explanation

The loader is located in the provisioning place area. Only restricted movements are permissible.

9.1.2.49 67143 - WMR: Loader in tool area

Explanation

The loader and/or the tool in the loader is located in the tool area. Only restricted movements are permissible.

9.1.2.50 67144 - WMR: Loader in provisioning place entry area. Position X, Y, Z or B impermissible.

Explanation

The tool loader is located in the provisioning area. Only positions brought about by the +/- 1mm approach movement are permissible. The approach movement occurs via a pre-position GD_WMR_BPVORPOS at GD_WMR_BPPOS.

9.1.2.51 67145 - WMR: Rack-type magazine not in home position

Explanation

To be able to work in automatic mode, the rack-type magazine must be in the home position. One of the following conditions must be met:

- Loader is at the provisioning position.
- Loader is in the collision-free magazine area and the B-axis is aligned.

9.1.2.52 67146 - WMR: Tool present in loader during tool pick-up.

Remedy

The loader must be empty before it picks up a tool.

9.1.2.53 67147 - WMR: Tool has invalid return place. Deposit in : (magazine, place, tool) %4

Explanation

The tool in the loader does not have a valid return location. Either no location or an occupied or reserved location. No empty location search is carried out for this tool. The tool is deposited at this location.

9.1.2.54 67148 - WMR: Tool %4 has an invalid return location. Empty place search unsuccessful

Explanation

The tool in the loader does not have a valid return location. Either no location or an occupied or reserved location.

Remedy

The empty place search initiated for this tool was unsuccessful. Tool must be unloaded.

9.1.2.55 67149 - WMR: Runtime monitoring place reservation

Explanation

On unloading the rotary station, tools without ownership position were discovered. The PLC does not respond when attempting to serve a place for this tool.

9.1.2.56 67150 - WMR: Tool without loading information found on place %4 of rotary station. Place has been disabled

Explanation

On swivelling in the rotary station, a tool was found at the defined location. However, no tool was entered at this location.

Remedy

Swivel out rotary station again.
Unlock location.
Enter tool information (load).

9.1.2.57 67151 - WMR: Loading information for place %4, but no tool present. Place data have been deleted.

9.1.2.58 67152 - WMR: Tool %4: Tool size, number of half-places incorrectly defined

Explanation

An inadmissible value was entered for the tool size in number of half places ($TC_TP3 and $TC_TP4). 1 to 5 are permitted.

Remedy

Correct tool size.
Repeat loading operation.

9.1.2.59 67153 - WMR: Runtime monitoring tool management acknowledgement

Explanation

The NC has requested a tool management acknowledgement from the PLC. This acknowledgement was not executed. The interaction between NC and PLC is disrupted.

9.1.2.60 67154 - WMR: No clear traversing possible. B-axis outside the tolerance range. Status: %4

Explanation

Clear traversing cannot be executed because the B-axis of the rack-type magazine is not within the required tolerance.

Remedy

Traverse B-axis in JOG to permissible value and clear traverse again.

9.1.2.61 67155 - WMR: Read code carrier: No or incorrect tool in loader

Explanation

If the code carrier of a tool is to be read, the CT_DUMMY tool must be in the loader. The tool is created by the HELLER system cycles.

9.1.2.62 67156 - WMR: Dummy tool for read code carrier cannot be created. Place %4 in rotary station disabled

Explanation

If the code carrier of a tool is to be read, the tool CT_DUMMY tool must be in the loader.
An error occurred on creating this tool.

Remedy

Check number of tools created. A maximum of 600 tools are possible.

9.1.2.63 67157 - WMR: Read code carrier unsuccessful. Place %4 in rotary station disabled.

9.1.2.64 67158 - WMR: Job cannot be executed. Measuring mode active

Explanation

The rack-type magazine is in measuring mode. Camera, laser distance measurement and retaining bracket are mounted. Regular operation not possible due to new collision contours.

9.1.2.65 67159 - WMR: Frame components <> 0

Remedy

No zero point shifts may be active in the second channel.

9.1.2.66 67160 - Please request check ghost tools

9.1.2.67 67161 - WMR: Write code chip failed. Tool %4 deposited at original location.

Explanation

During the unloading operation, an attempt is made to write to the code chip. The code chip could not be successfully written to. The unloading job will be terminated. The tool is returned to its magazine place, not to the rotary station.

9.1.2.68 67162 - WMR: Read code chip unsuccessful. Tool %4 deposited at loading location.

Explanation

During the loading operation, an attempt is made to read the code chip. The code chip could not be successfully read. The loading job will be terminated. The tool is returned to the rotary station, not to its magazine place.

9.1.2.69 67163 - WMR: Read/write code chip was cancelled. Loader not on specified path

Explanation

While the loader was on its way from or to the reading station, the transport operation was cancelled. The current positions of the X, Y, Z or B axes do not correspond to the intended path.

9.1.2.70 67166 - WMR: No. of half-places and envelope datum HD of the tool: %4 do not match

Explanation

9.1.2.71 67167 - WMR: Runtime monitoring tool shank cleaning: Communication between NC and OEM

Explanation

When the tool shank cleaning option is installed, data are exchanged between the NC and the OEM application. The OEM application does not respond within 2 seconds.

9.1.2.72 67168 - WMR: Incorrect envelope data for code chip tool. Place %4 in rotary station disabled.

Explanation

The envelope data of a tool loaded with code chip are invalid. The tool cannot be deposited in the magazine. The tool is returned to its place in the rotary station. The tool's place in the rotary station is disabled.

9.1.2.73 67169 - magazine place test: Place %4 unable to be tested!

Explanation

There is insufficient space in the magazine for manoeuvring large tools. Stated place not approached by test tool

9.1.2.74 67170 - Execute operating sequence for remove manual pick-up tool

Explanation

9.1.2.75 67171 - Execute operating sequence for insert manual pick-up tool

Explanation

9.1.2.76 67172 - Tool not clamped

Explanation

Tool not clamped during manual pick-up operating sequence.

9.1.2.77 67200 -IPM Signal adjustment not carried out

9.1.2.78 67210 - The machine is horizontally oriented

Explanation

The machine's coordinate system is horizontally aligned.
Tool lengths offset in Z direction.

9.1.2.79 67300 - Unable to clear traverse loader automatically

Explanation
The axis position of the rack magazine prevents the loader from automatically clear traversing.

Remedy
Traverse loader to collision-free area in JOG mode.

9.1.2.80 67301 - Incorrect magazine assignment (magazine1,place1,magazine2,place2): %4

Explanation
Inconsistencies occurred on attempting to restore data in tool magazine.

Remedy
Check entered magazine number and place number.

9.1.2.81 67600 - %5: Tool broken

9.1.2.82 67601 - %5: Maximum speed reached

Remedy
NC program correction

9.1.2.83 67602 - %5: Errors: Width R46<D-milling, if more R46 = milling diameter

Remedy
NC program correction

9.1.2.84 67603 - %5: Errors: Tool length wrong or effect of coolant

Remedy
NC program correction

9.1.2.85 67604 - %5: Diameter (R83)=%6

Remedy

NC program correction

9.1.2.86 67605 - %5: Tool OK

9.1.2.87 67606 - %5: Caution! Overrun limit in X/Y exceeded, check R6!

Remedy

NC program correction

9.1.2.88 67607 - %5: Caution! Overrun limit in Z exceeded, check R6!

Remedy

NC program correction

9.1.2.89 67608 - %5: Centre of 1st axis (R81)=%6, centre of 2nd axis (R82)=%7

9.1.2.90 67609 - %5: Calibration value, axial (R80)=%6

9.1.2.91 67610 - %5: Sequence mark(R89)=%6

9.1.2.92 67611 - %5: Surface 3rd axis (R83)=%6

9.1.2.93 67612 - %5: Calibration value (R81)=%6

9.1.2.94 67613 - %5: Edge 1st axis (R81)=%6, edge 2nd axis (R82)= %7

9.1.2.95 67614 - %5: Centre 1st axis (R81)=%6, centre 2nd axis (R82)= %7

9.1.2.96 67615 - %5: Centre X,R81=%6, centre Y,R82=%7

9.1.2.97 67616 - %5: Diameter X,R83=%6, diameter Y,R84=%7

9.1.2.98 67617 - %5: Width(R83)=%6

9.1.2.99 67618 - %5: 1st measuring point(R81,R82)=%6, %7

9.1.2.100 67619 - %5: 2nd measuring point(R83,R84)=%6, %7

9.1.2.101 67620 - %5: Angle(R85)=%6

9.1.2.102 67621 - %5: 1st measuring point(R83)=%6, 2nd measuring point(R84)=%7

9.1.2.103 67622 - %5: Angle 2nd axis(R85)=%6, angle 1st axis(R86)= %7

9.1.2.104 67700 - Magazine station number %4 inadmissible

Remedy

Permissible values are: 1-9, 11-19, 21-29, 31-39

9.1.2.105 67701 - Job acknowledge %4 unknown

9.1.3 68xxx alarm messages - with NC stop

9.1.3.1 68100 - Tool changer not in home position

Explanation

The tool changer is in the home position. Either the changer axes positions or the statuses of the collet chuck, tool changer door, or similar are not correct.

Response

Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
NC stop

Remedy

Place the tool changer in the home position.
(setup functions or Automatic home position drive)

9.1.3.2 68101 - SBBK: The teach-in process of the tool in preparation has failed

Explanation

The tool for this tool change is designated for checking with SBBK according to the tool user date special code Bit3. The teach-in process of the tool in preparation has failed therefore it is no longer possible to check the tool.

Response

Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
NC stop.

Remedy

9.1.3.3 68102 - No successful provisioning before tool change

Explanation

Provisioning must have been carried out successfully before the tool change. If no provisioning was programmed or the provisioning was negatively acknowledged, the tool management system can no longer be supplied with valid data.

9.1.4 69xxx alarm messages - with NC stop at the end of the block

9.1.4.1 69123 - SBBK: Tool breakage

Explanation

Response

Display.
Entry in the log file.
Interpreterstop.
NC start disable in this channel.
NC stop at the end of the block.

Remedy

9.1.5 75xxx - alarm messages

9.1.5.1 75120 - IPM interpreter configuration alarm error no.: %2

9.1.5.2 75121 - Block %2 IPM interpreter programming error, error no.: %3

9.1.5.3 75122 - IPM break alarm, method no.: %2, methods ID: %3%4

9.1.5.4 75123 - IPM overload alarm, method no.: %2, methods ID: %3%4

9.1.5.5 75124 - IPM interpolator alarm error no.: %2

9.1.5.6 75125 - Axis %1 IPM servo-alarm error no.: %2 %3%4

9.1.5.7 75126 - IPM break alarm (cut), method no.: %2, methods ID: %3%4

9.1.5.8 75127 - IPM overload alarm (blunt), method no.: %2, methods ID: %3%4

9.1.5.9 75128 - IPM break alarm (missing), method no.: %2, methods ID: %3%4

9.1.5.10 75129 - IPM general error no.: %2%3%4

9.1.5.11 75130 - Axis %1 IPM collision alarm %2%3%4

9.1.5.12 75131 - Axis %1 IPM belt break alarm %2%3%4

9.1.5.13 75133 - Block %2 IPM interpreter: GUD method data not found

9.1.5.14 75134 - Axis %1 IPM: Check rigidity of drive train%2%3%4

Chapter 10

Compatibility

10 Compatibility

10.1 Compatibility between H/MC machines and MCi/MCH machines

Incompatibilities between the H/MC machines and MCi/MCH machines are reduced to a minimum. If possible macros are used to produce the necessary compatibility. However, macros may soon no longer be capable of ensuring compatibility during continued stages of development.

10.1.1 Variable names

The following variable names were changed.

H/MC series	MCi/MCH	Compatibility macro
GV_WZW_ZPOS	GV_TC_ZPOS	yes
GD_WZW_POS[0]	GD_TC_POS[0]	yes
GD_WZW_POS[1]	GD_TC_POS[1]	yes
GD_WZW_POS[2]	GD_TC_POS[2]	yes

10.1.2 Axis name

Modulo axes

With H/MC machines, modulo axes position by default according to the "direct path" DC method. As a result the swivelling cycle `CYCLE800` is executable.
By default modulo axes position according to the AC method for MCi/MCH machines.

Axis name

The axis identifiers for the H/MC machines conform to DIN and differ from the axis identifiers for MCi/MCH machines as follows:

axis	H/MC machines		MCi/MCH		Compatibility macro
	machine axis name	Channel axis name / geo axis name	machine axis name	channel axis name	
X axis	M_X1	X1	MA_X	CA_X	yes
Y-axis	M_Y1	Y1	MA_Y	CA_Y	no
Z axis	M_Z1	Z1	MA_Z	CA_Z	no
A-axis	M_A1	A	MA_A	A	yes
B-axis	M_B1	B	MA_B	B	yes
C-axis:	M_C	C	MA_C	C	yes
Spindle	M_C1	C1	MA_C1	C1	yes
Chain	M_CM	CM	M_MC	MC	no
Rack magazine X	M_XM	X	M_MX	MX	no
Rack magazine Y	M_YM	Y	M_MY	MY	no
Rack magazine Z	M_ZM	Z	M_MZ	MZ	no
Rack magazine A	M_AM	A	M_MA	MA	no
Changer axis rotatory	M_CT	CT1	M_TC	TC	no
Changer axis linear	M_ZT	ZT1	M_TZ	TZ	no
NC setting station	M_BR	BR	M_RB	RB	no
Facing slide axis	M_U1	U	MA_U	U	no

Tool identifiers replaced with macros cannot be called up in macros that the user has defined. Macros cannot be nested.

10.1.3 Tool data

Tool data

Tool datum	H/MC machines	MCi/MCH
$TC_TPC1	Tool change speed Bit0: normal	Change speed
$TC_TPC2	Max. speed	Max. tool speed
$TC_TPC3	Special identifier Bit3: SBBK Bit4: Tool hat CC Bit5: Facing slide attachment Bit14: Tool not disposed of automatically (WMR) Bit16: Tool moved out of spindle	Tool diameter
$TC_TPC4	Envelope datum: Diameter HD	Tool height
$TC_TPC5	Envelope datum: Tool length HL (at spindle reference point)	Shank diameter
$TC_TPC6	Envelope datum: Tool height HH	Tool length
$TC_TPC7	Reference value SBBK	Shank length
$TC_TPC8	not present	Special identifier
$TC_TPC9	not present	Usual tool
$TC_TPC10	not present	Facing head type / angular tool type

Cutting edge data

Tool datum	H/MC machines	MCi/MCH
$TC_DPC1	Tip-specific facing slide type	Drill radius
$TC_DPC1	not present	Point angle (drill)

Monitoring data

Tool datum	H/MC machines	MCi/MCH
$TC_MOPC1	ID of assigned breakage monitoring method	ID of assigned breakage monitoring method
$TC_MOPC2	ID of assigned overload monitoring method	ID of assigned overload monitoring method

10.1.4 M functions

M function H/MC machines	M function MCi/MCH	New function	Compatibility
205	10	Automatic clamping on, clamp axis 5/B	yes, with macro
245	11	Automatic clamping off, clamp axis 5/B	yes, with macro
204	110	Automatic clamping on, clamp axis 4/A	yes, with macro
244	111	Automatic clamping off, clamp axis 4/A	yes, with macro
202	212	Automatic clamping on, clamp axis 2/Y	If rack axis Y has no clamping
242	213	Automatic clamping off, clamp axis 2/Y	If rack axis Y has no clamping

M functions M20 - suction before WZW

Up until now M20 could be used to blow the remaining coolant out of the spindle and tool with air before changing the tool.
Since the tools are now sucked dry this extension must take place with M23.

Blowing out blind holes

M20 can still be used for blowing out blind holes.

10.1.5 Tool change

Tool provisioning

The cycle CS_TP("identifier") can still be used but shouldn't be used if possible.

CS_TP will continue to be supported. In future machine generations, however, the cylce may no longer be available.

The tool preparation takes place as illustrated:

T="identifier"	T="SPIBO5"	
T followed by a number	T123	T0

Spindle tool change position

H/MC series	MCi/MCH	
not used	GD_TC_POS[3]	B
GD_TC_POS[3]	GD_TC_POS[4]	A
not used	GD_TC_POS[5]	AA
GD_TC_POS[5] GD_WZW_POS[5]	GD_TC_POS[7]	Spindle
GD_TC_POS[6]	GD_TC_POS[8]	Facing slide

No compatibility macro

Tool change

The tool change is triggered with M6 or M46.
The cycle CS_TOOL can still be used but shouldn't be used if possible.

CS_TOOL will continue to be supported. In future machine generations, however, the cylce may no longer be available.

The tool change cannot be called up with active zero point offsets (frames) which contain rotation, reflection or scaling. If this is attempted, the error message "66117 Active frame contains reflecting, scaling or rotation" is outputted. This was not possible with MCi/MCH unconditionally either.

Fast tool break monitoring SBBK

The measurement is no longer programmed in the NC program. The tool is informed that it should be measured based on the tool user datum $TC_TPC3. The tool is measure during first use and the reading is stored. The tool is then scanned during each tool change.

The variable GV_PBTD_TOOLL can no longer be used.

If variables GV_PBTD and GV_SBBK are programmed, although the tool has not been marked for SBBK, the following alarm is generated "66101 SBBK is in the NC-program. But OEM tool datum not entered".

The variable GV_PBTD_TOL can no longer be used. GV_PBTD_TOL describes the tolerance of the tool length in mm. The tolerance for SBBK can be programmed with GV_SBBK_TOL in number of increments of the measuring system of SBBK.

10.1.6 Pallet management

H/MC series	MCi/MCH	Compatibility macro
gv_palnv	gv_pal_ufr	yes
gv_palnv	gv_pwt_cnc	yes
gv_beanv	gv_wp_ufr	yes
gm_palncpara	gm_pal_par gm_pal_par=gv_pwt_par	yes
gm_beancpara	gm_wp_par	yes
gm_palzu	gm_pal_state	yes
gm_palzi	gm_pal_state_info	yes
gm_beazu	gm_wp_state	yes
gm_beazi	gm_wp_state_info	yes
gm_beabl	gm_set_wp_state	yes
gm_palnr	gm_pal_no	yes
gm_beaid	gm_wp_typ_id	yes
gm_beaind	gv_pwt_par gv_pwt_par=gm_pal_par	yes
gm_palpos	gm_pal_chg_ori	yes