HEIDENHAIN

Controller manual V420-V500

MillPlus

Teil 1: Extension V420-V500

Teil 2: Controller manual V410

HEIDENHAIN NUMERIC B.V.
Eindhoven (NL)
Tel: 31.40.250 13 00
Fax: 31.40.250 13 01

04/2001

344 938-20

Extension
V420-V500

HEIDENHAIN NUMERIC B.V.
Eindhoven (NL)
Tel: 31.40.250 13 00
Fax: 31.40.250 13 01

21/03/2001

First edition Software Version V420-V500 20010321

344 938-10 DER6-32.8-2574_000

1. Introduction

Dear Customer,

These instructions are intended to assist you in operating and programming the controller.

Please read the information in this manual carefully before you start your new machine. It contains important information on machine operation and safety to enable you to use your machine safely and effectively.

The following advice is important for your safety:

This manual is essential for safe use of the machine.
Please ensure that it is in the vicinity of the machine.

The machine should not be operated, even for a short period, by anyone who has not received suitable training, either in the company, at an Institute of Further Education or in one of the Training Centres.

Please read the general safety regulations issued by your professional association.
If they are not on display in the company, contact your appointed safety representative.

Observe the instructions for proper use of the machine.

The controller and the machine are coordinated using machine constants. Some of these constants are accessible to the user. Caution!
The meaning and function of the constants must be fully understood before any changes are made to these constants. If in doubt, please consult our service department.

The controller is fitted with a backup battery that safeguards the memory content for up to three years after the machine is switched off (but only if the battery is serviceable).

The user should always save the programs and specific data (e.g. technology data, machine constants, etc.) to a PC or to diskette. This will avoid the data becoming irrecoverably lost if the system or backup battery becomes defective.

We reserve the right to make changes to the design, equipment and accessories in the interest of further development. No liability will be accepted for any errors in the data, illustrations or descriptions.

1.1 MillPlus software and functions

This manual describes functions available in MillPlus (VME and LE4xx hardware) for the following or higher software versions:
- V410 (VME, LE4xx) Software number 341 482-xx
- V420 (VME, LE4xx) Software number 344 198-xx

The machine builder adapts the versatile capability of MillPlus to the machine in question by means of machine parameters. That is why some functions described in this manual are not available with every version of MillPlus.
MillPlus functions that are not available on every machine include, for example:
- Digitising option
- Tool measurement with laser system or with TT120/TT130
- DNC Plus (DNET) interface
- Ethernet interface (TCP/IP)
- Autostart (warm machine startup program)
- Milling/turning

Please contact the machine builder for individual support for the particular machine being controlled.

1.2 Software version V420

Note

The V420 software only functions in 16 Mbyte DRAM systems.

Operation:

Expanded calculation operations

- The number of possible addresses in Cycle Design is expanded and OPTIONAL and ACTIVE have been added.
- Calculations for E parameters expanded with the floor(E2), ceil(E2), round(E2,n), mod(E2,E3), sign(E2), atan(E2,E3), acos(E2,E3) and asin(E2,E3) functions.

New G functions:

Dynamic TCPM (Tool Centre Point Management) (G141)
Tool withdrawal movement (G174)
M19 with programmable direction (G303)
Query current G data (G320)
Query programmable kinematic elements (G329)
Write programmable kinematic elements (G339)
Writing to a window (G350)
Writing to a file (G351)

Milling/turning

G17	expanded
G36 and G37	turning
G94 and G95	expanded
G96 and G97	constant cutting speed
G302	overwrite tool orientation radius correction
G691	unbalance detection
G692	unbalance checking
G822, G823, G826 and G827	machining
G832, G833, G836 and G837	hollow boring
G842, G843, G846 and G847	plunge cutting

Expanded G functions:

Expansions: tool measurement cycles for laser measuring system

G600 laser system: calibration
G601 laser system: measuring length
G602 laser system: measuring length and radius
G603 laser system: individual cutting tip monitoring
G604 laser system: tool breakage monitoring

G8 selectable tool radius correction
Start of contour description (G198) expanded with I1 address.
Start of graphic contour description (G199) expanded with B=4.
Query active technology **(G139)** expanded with S1 address.
Query tool table **(G321)** expanded with O address.
Query current modal G function **(G324)** expanded.
Schreiben in die Werkzeugtabelle **(G331)** erweitert mit O Adresse.

2. Expanded G functions and calculation operations

2.1 Expanded G functions

2.1.1 G8 Selectable tool radius correction

L3=0 with radius correction (standard value)
L3=1 no radius correction

2.1.2 G197/G198 Start of internal/external contour description

Defines the start point of a geometry element:
N... G198 X... Y... {Z...} {I1=..}.
Possible colours (I1=):

1	red	11	light red
2	green	12	light green
3	yellow	13	light yellow
4	blue	14	light blue
5	grey	15	light magenta
6	cyan	16	light cyan
7	white	17	bright white
8	black	18	black
9	foreground	19	foreground
10	background	20	background

2.1.3 G199 Start of graphic contour description

Draw one or more geometry elements (line or circle) during the wire model graphic simulation.
N... G199 [co-ordinates of position] B4 {C1} {C2}

2.1.4 G321 Query tool table

I1=29 O tool orientation (with turning option only)

2.1.5 G324 Query current modal G function

11 G96, G97 (turning only)
12 G36, G37 (turning only)

2.1.6 G331 Writing to the tool table

I1=29 O tool orientation (turning only)

2.2 Expanded calculation operations

2.2.1 E parameters

Format:

Arc sine E1=asin(E2,E3)
Arc cosine E1=acos(E2,E3)
Arc tangent E1=atan(E2,E3)
Whole number conversion with large value E1=ceil(E2)
Whole number conversion with small value E1=floor(E2)
Rounding E1=round(E2,n) (n is no. of decimal places)
Remainder of division E1=mod(E2,E3)
Sign E1=sign(E2)

Remark: The integer function is changed with the floor function in V420 and higher.

2.2.2 Whole numbers

When using the integer function, the numerical value is rounded, i.e. all figures after the decimal point are ignored.
E1=int(E2)

Example: E2=8.9 results in 8, E2=-8.9 results in –8

2.2.3 Whole numbers with largest value

When using the integer function with the largest value, the numerical value is rounded

according to the largest argument.
E1=ceil(E2)

Example: E2=8.9 results in 9, E2=-8.9 results in –8

2.2.4 Whole numbers with smallest value

When using the integer function with the smallest value, the numerical value is rounded according to the smallest argument.
E1=floor(E2)

Example: E2=8.9 results in 8, E2=-8.9 results in –9

2.2.5 Rounding

When the rounding function is used, the numerical value is rounded according to the number of decimal places.
E1=round(E2,n) (n is number of decimal places)
Remark: If the number of decimal places is not entered, zero is assumed.

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

2.2.6 Remainder of division

When the remainder function is used, the remainder is returned by the argument.
E1 =mod(E2,E3)
Remarks:
-E1=E2-int(E2:E3)*E3
- If E3 is 0, E2 is returned.
- If E3 is not entered, 1 is assumed.
- The sign is the same as the sign of E1.

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

2.2.7 Sign

When the sign function is used, the sign is returned.
E1 =sign(E2)

Example: E2=8.9 results in 1, E2=0 results in 0, E2=-8.9 results in -1
Also possible (V429 and higher):
E1=asin(E3,E4) E1=acos(E3,E4) E1=atan(E3,E4) where E2=E3:E4
Remark: - abs(E2) must be less than or equal to 1 for acos and asin.
- the angle created lies between 0° and +360°

2.2.8 Variable parameter no.:

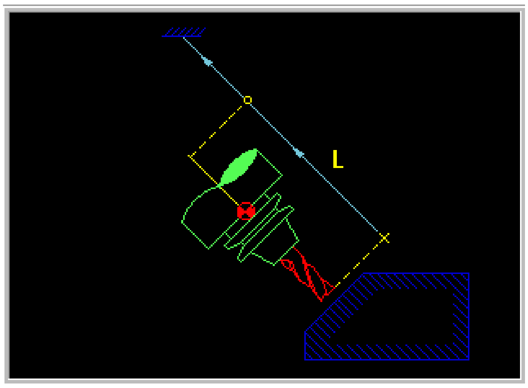
E(value or expression)=<value or expression>

Examples:
E(1)=
E(1.2e1)
E(E1)=
E(E1+E2)=
E(sin(45)*100)=

3. G174 Tool withdrawal movement

Movement to move the tool axis clear during 5-axis milling.

With this function, you are always able to move away in the direction of the tool axis (only if Z axis is programmed). The tool is withdrawn until the ‘first’ SW limit switch is reached. The direction of movement is determined by the position of the milling head. The calculation is carried out with the help of the kinematic model.



Format

N.. G107 {L....}

Parameter

G	Tool retract movement
L	Retract distance

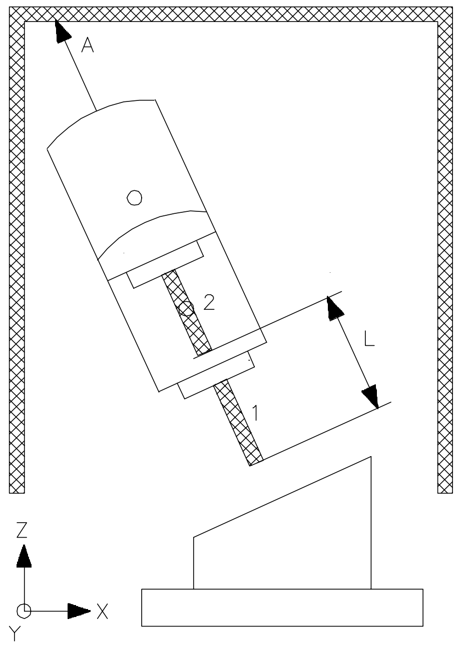
Modal words

F6=

Type of function

Non-modal

Notes and application



1	**Starting position**
L	**Withdrawal distance**
2	**End position**
A	**Limitation by software limit switch**

WITHDRAWAL DISTANCE (L)

The withdrawal distance defines the distance travelled in the direction of the tool.
L is always positive. The maximum travel distance is limited by the software limit switch. The basic setting is with the software limit switch.

DIRECTION OF MOVEMENT

The direction of movement is determined by turning the milling head. This is calculated with the help of the kinematic model.
The function is always active.

EXECUTION (G0 or G1)

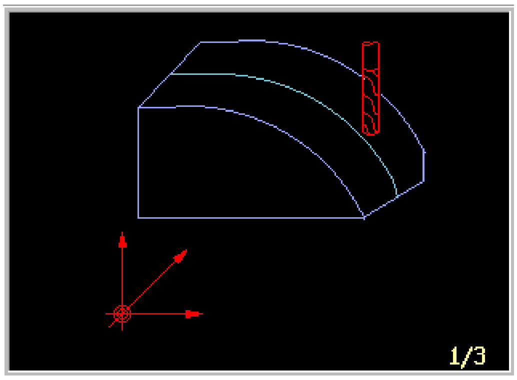
G174 is executed in rapid motion or if F6= <Feed per block> is programmed with F6=.
Following G107, G0 or G1 from the previous block is modally active again.

Example Tool withdrawal movement.

N10 G174 L100 Werkzeug zieht sich 100 mm zurück oder fährt bis Software Endschalter.

4. G141 3D tool correction with dynamic TCPM

Permits the correction of tool dimensions for a 3D tool path that is programmed in these points by its end point co-ordinates and normalised vectors perpendicular to the surface.



Format

To activate 3D tool correction:

N... G141 {R..} {R1=..} {L2=}

To program straight-line movements:

G141

N... G0/G1 [end point co-ordinates] [I.. J.. K..]

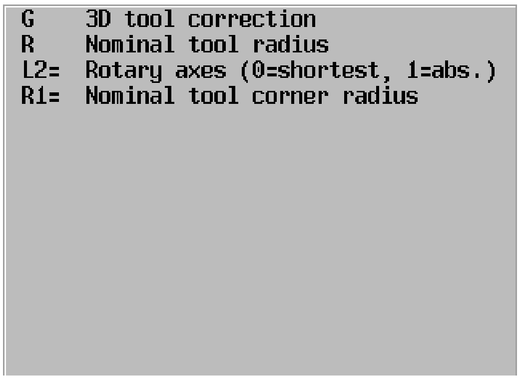
TCPM with active kinematic model

N... G0/G1 [end point co-ordinates] {I.. J.. K..} {I1=.. J1=.. K1=..} {A, B, C} {F..}

To delete 3D tool correction:

N... G40

Parameters



For G141

R	nominal tool radius
R1=	nominal tool corner radius
L2=	circular axes (0=shortest, 1=absolute)

For G0/G1

X, Y, Z	linear end point co-ordinates
I, J, K	axis components of surface normal vector
I1=, J1=, K1= **(TCPM)**	axis components of tool vector

A, B, C **(TCPM)**	circular axis components of tool vector
F	feed along the path

Modal words

M functions determined by F, S

Type of function

modal

Associated functions

G40 and G412 to G44 for radius correction in a plane

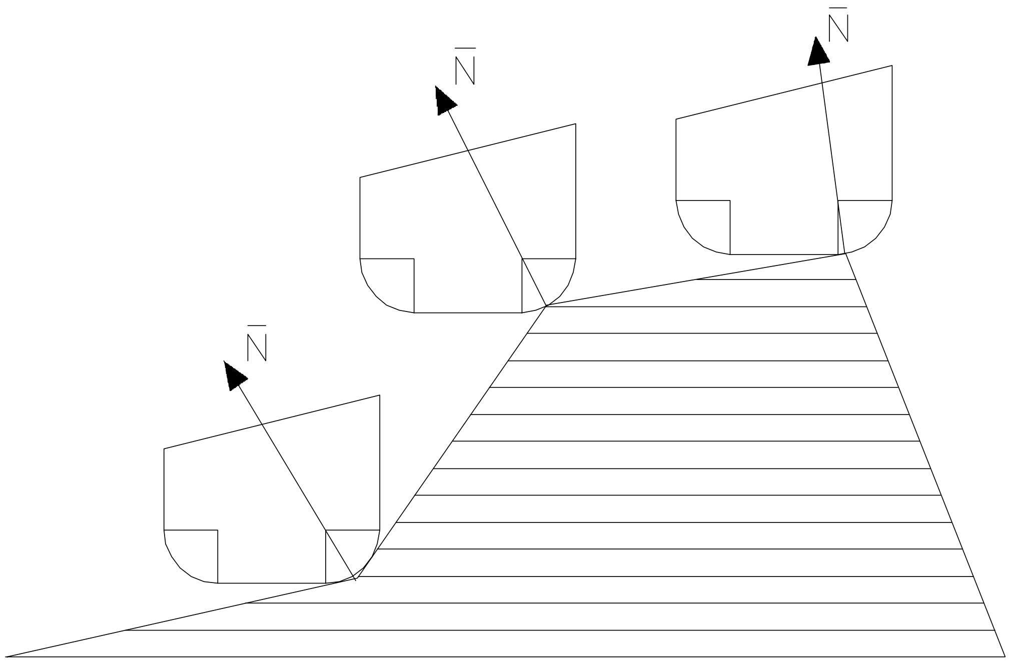
For TCPM G8

General principles of G141

When milling a 3D surface, a given tool is moved along the surface in straight-line movements with a particular tolerance.
The calculation of the tool path on a 3D surface requires many calculations that are usually carried out by an NC programming system or a CAD system.
The calculated tool path depends on the shape of the tool, the dimensions of the tool and the tolerance to the surface.

When executing the appropriate program **without G141**, the milling tool used must have the same dimensions as in the calculations, i.e. a standard milling tool must be used.
If a new tool is required while machining a 3D surface, this tool must also have the same dimensions as the **standard tool**.
If dimensional deviations are detected on the workpiece, a new calculation must be made using the programming system.

The 3D tool correction **(G141)** allows the use of tools whose dimensions differ from the dimensions of the standard milling tool. The corrections are carried out with the help of the direction vectors that are created by the programming system together with the end point co-ordinates.
In addition, the workpiece dimensions can be calculated by the programming system and the tool path by the CNC from the normalised vectors and the tool dimensions.



$\bar{N}$ = surface normal vector (I, J, K)

Notes and application

The values of R.. and R1=.. should correspond to the nominal tool dimensions as used by the **programming system** for calculating the tool path. If these values are not programmed, they automatically become zero.

RADIUS (R, R1=)

R defines the tool radius with which the end points of the G0/G1 blocks are calculated in the CAD system.

R1= defines the tool corner radius with which the end points of the G0/G1 blocks are calculated in the CAD system.

General principles of TCPM

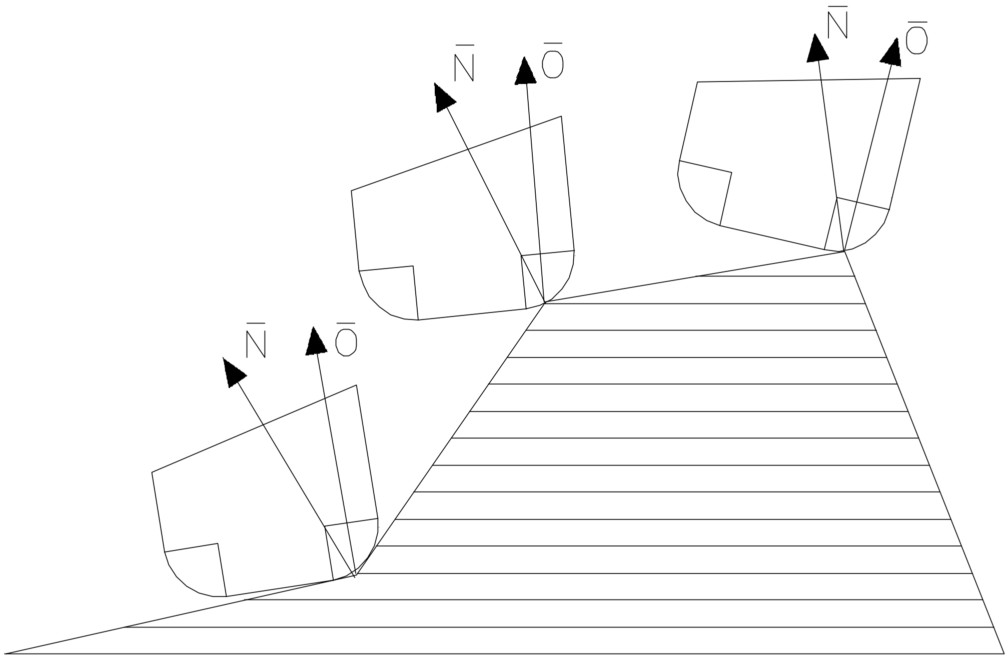
Maintaining position of tool tip when positioning swinging axes (TCPM)
(TCPM stands for "Tool Centre Point Management").

With **G141 '3D tool correction without TCPM'**, a curved (CAD) surface can be travelled taking the current tool dimensions into account. In this case, the path is described by end point co-ordinates and vectors perpendicular to the surface. The G141 function only guides the three linear axes but not the circular axes. In this way, the tool is always used in the same direction and is not guided over the workpiece surface at the optimum angle.

With **G8 'Tool orientation'** (static TCPM), the tool can be placed on the surface of the workpiece at an optimum angle. The G8 function is a feed movement and cannot be used continuously on a curved surface during a path movement.

In the case of **G141 with dynamic TCPM**, the tool is guided on a curved workpiece surface at an optimum angle. The current workpiece dimensions are taken into account. Dynamic TCPM is used for

5-axis milling. Dynamic TCPM also controls the circular axes. The tool is guided on the curved workpiece surface either vertically or at a programmed orientation.



$\overline{N}$ = surface normal vector (I, J, K)

$\overline{O}$ = tool vector (I1=, J1=, K1=) or (A, B, C)

The programming format of the linear blocks within G141 is expanded to include the option of programming a tool vector. Possible combinations are surface normal vectors and/or tool vectors. If only the tool vector is used, the tool correction must be calculated in the CAD system.

G7 may be active. In this case, the surface normal vectors and the tool vectors are defined in the G7 level.

Notes and application

ADDRESSES (R, R1=, L2=) **(TCPM)**

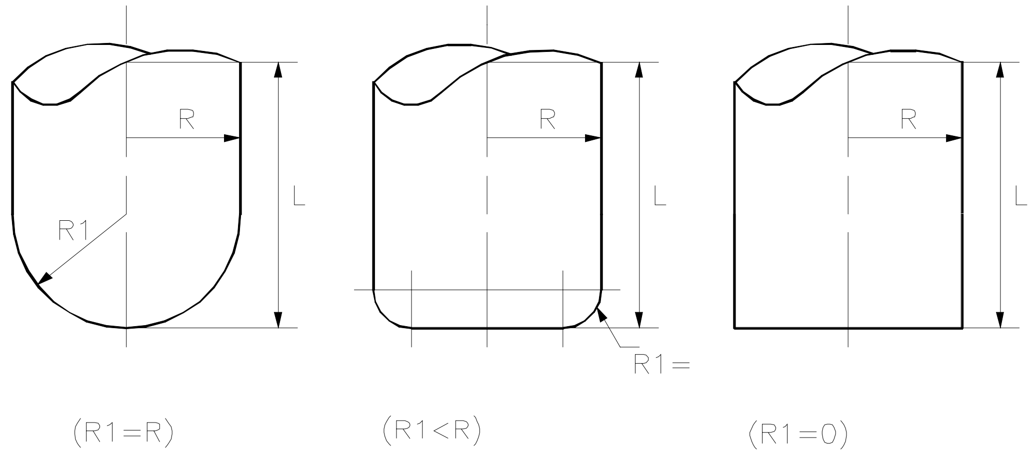
R defines the tool radius with which the end points of the G0/G1 blocks are calculated in the CAD system.

R1= defines the tool corner radius with which the end points of the G0/G1 blocks are calculated in the CAD system.

L2= 0 Circular axes travel the shortest distance (basic setting)

1 Circular axes travel to their absolute position (with circular axis programming).

POSSIBLE TOOLS



Tools used for the G141 function

TOOL MEMORY

The following dimensional details must be loaded into the tool memory to enable different types of tools to be used:

Radius milling tool	:	R (tool radius), L (tool length), C (=tool radius)
Radius end milling tool	:	R (tool radius), L (tool length), C (=rounding radius)
End milling tool	:	R (tool radius), L (tool length), C0

If no value of C is entered, C automatically becomes 0.
The standard milling tool is thus an end milling tool.

Note: The rounding radius in the G141 block is programmed with the word R1=. The rounding radius is stored in the tool memory with the C word.

CREATED TOOL PATH

When the programming system creates the tool path (surface normal vector is programmed), the dimensions of the nominal tool (R.. and R1=) are programmed in the G141 block. The tool dimensions stored in the tool memory are used by the CNC to correct the tool path.

WORKPIECE DIMENSIONS

When the programming system creates the workpiece dimensions (surface normal vector and tool vector are programmed), the R.. and R1= words are not programmed in the G141 block. The tool dimensions stored in the tool memory are used by the CNC to calculate the tool path.

ACTIVATING G141

In the first block after G141, the milling tool travels from the current tool position to the corrected position in this block.

END POINT CO-ORDINATES

Only absolute or incremental (X, X90, X91) Cartesian dimensional data can be used.
Up to V420, the co-ordinates in the first G141 block must be absolute and are measured from the programming zero point W.

MIRRORING

If the mirroring function (G73 and axis co-ordinates) is active before G141 is activated, the mirrored co-ordinates are used during the 3D tool correction.
Mirroring is possible as before once G141 is activated. Mirroring is cancelled by the G73 function.

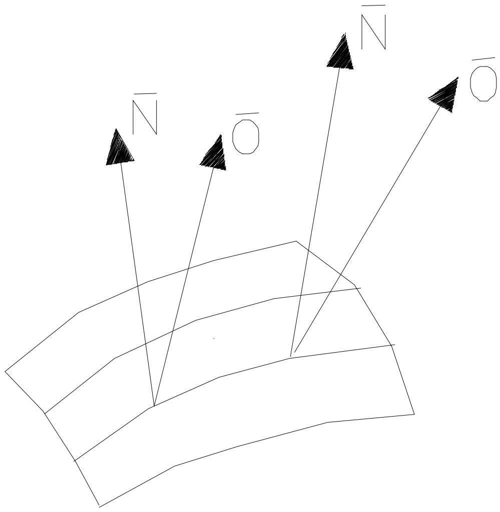
RADIUS CORRECTION G41...G44

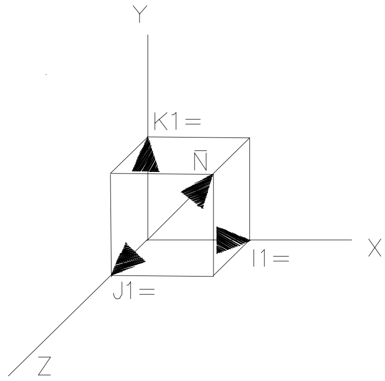
After activating a G141 block, the effective radius correction programmed with G41...G44 is deleted.

SURFACE NORMAL VECTOR (I, J, K) **(TCPM)**

Defines the surface normal vector perpendicular to the surface.

The surface normal vector is perpendicular to the workpiece surface. The tool is positioned so that this vector always passes through the centre point of the tool corner rounding. This vector controls the positioning of the linear axes within G141.





VECTOR COMPONENTS

The vector components of the axes are independent of the level selected.

If no vector components are programmed in a block, the values last programmed are used.

If no components are programmed in the first block, the components not programmed are set at zero.

DIMENSION FACTOR

The input format of the vectors (I, J, K, I1=, J1=, K1= words) is limited to three decimal places. The surface normal and tool vectors do not, however, have to have the length 1. To increase the dimensional accuracy, the values in question can be multiplied by a dimension factor between 1 and 1000. With the factor 1000, for example, the input accuracy of the vector components is increased to six significant figures.

BACK CUTTING

Back cutting or collisions between tool and material at points not to be machined are not detected by the CNC.

KINEMATIC MODEL **(TCPM)**

The kinematic model is used for calculations within G141.
If no kinematic model is active (MC312 'Free machining level' = 0), G141 remains compatible with the G141 functions in older CNC versions.

TOOL VECTOR **(TCPM)**

I1=, J1=, K1=	axis components of tool vector
or	
A, B, C	circular axis components of tool vector

The tool vector or the circular axis co-ordinates indicate the direction of the tool axis. The tool is turned so that it is parallel to this vector. This vector controls the positioning of the circular axes (and the associated compensation movement with linear axes) within G141.

DELETING

Function G141 is deleted by G40, M30, the program interrupt softkey or the CNC reset softkey. The milling tool stops at the last corrected position. The circular axes are not turned back automatically.

FUNCTIONS TO BE DELETED

When working with G141, functions G64, scale change (G73 A4=..), axis rotation (G92/G93 B4=..) and G182 must be deleted.

The following G functions are permitted if G141 **(TCPM)** is switched on:

Basic motions	0, 1, 7
Levels	17, 18
Program control	14, 22, 23, 29
Positioning feed	4, 25, 26, 27, 28, 94, 95, 96, 97
Radius correction	39, 40, 141
Zero points	51, 52, 53, 54, 92, 93
Geometry	72, 73
Co-ordinate measurement modes	70, 71, 90, 91
Graphics	195, 196, 197, 198, 199

If a G function that is not permissible is programmed, error message P77 'G function and Gxxx not permitted' is issued.

The following G functions are permitted if G141 **(TCPM)** is active:

Basic motions	0, 1 Parameters of G0 and G1 are limited G0 without positioning logic
Program control	14, 22, 23, 29
Positioning feed	4, 25, 26, 27, 28, 94, 95, 96, 97
Radius correction	40, 141 G40 switches G141 off
Zero points	51, 52, 53, 54, 92, 93
Geometry	72, 73
Co-ordinate measurement modes	90, 91

If a G function that is not permissible is programmed, error message P77 'G function and G141 not permitted' is issued.

PROGRAMMING LIMITATIONS

G functions that are not listed above may not be used.
Point definitions (P) and E parameters may not be used.
No tool change may be made after activating G141.

Notes and application for TCPM

RISK OF COLLISION

When G141 is switched on, compensation movements similar to those in G8 may occur.

In the case of the switch-on movement, the tool tip must not be resting on the surface of the workpiece and should be programmed with a distance from the material at least equal to the tool diameter.

Remark: If G141 is switched off via G40, M30 or program cancel, there is no compensation movement and the circular axes remain in their last positions.

When approaching the contour, it may happen that the table with the workpiece is turned through 180 degrees to achieve the programmed tool direction. **ATTENTION! RISK OF COLLISION!**

UNDERCUTTING

If the tool direction changes within a G1 block, this tool direction change is carried out interpolating with the movement to the end point. In doing this, the path between the start and end points is corrected for undercutting.

Undercutting is not detected during block transitions. This undercutting should be corrected by inserting a block without an end point and with only one change of the tool vector by the CAD system. In this case, the tool turns about the tool contact point until the new tool direction is reached.

DISPLAY

When G141 is active, a yellow icon is displayed behind the tool number and the programmed G141 tool vectors (I1, J1, K1) can be seen in the machining status (on the G7/G8 positions).

Remark: If G7 and G141 are active at the same time, the G7 angle or vector can be seen.

A small ‘p’ at the bottom right, near the ‘axis letters’, shows whether the position of the tool contact point or the position is in machine co-ordinates. The display changes with the same softkey as with G7.

FEED RATE

The programmed feed rate applies to the contact point between the surface and the tool. The tool head may make other movements.

ERROR MESSAGES

P341 Tool vector incorrect

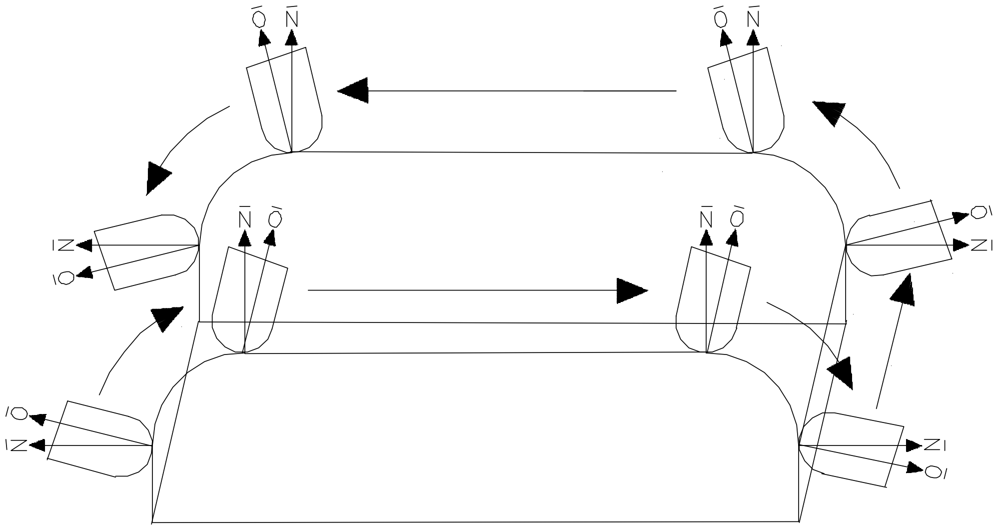
The tool vector (I1=, J1=, K1=) is incorrect. This error message is generated if all the components of the vector are zero.

P342 Surface normal vector incorrect

The surface normal vector (I, J, K) is incorrect. This error message is generated if all the components of the vector are zero.

Example: G141 and TCPM

Tool vector with (I1=, J1=, K1=)
This programming is independent of the machine.



N113 (square material with top rounding (R4) and swung tool (5 degrees)
N1 G17
N2 T6 M67 (10 round spherical milling tool: T6 R5 C5 in tool table)
N3 G54 I10
N4 G0 X0 Y0 Z0 B0 C0 S6000 M3
N5 F50 E1=0
N6 G141 R0 R1=0 L2=0 (all basic settings, do not need to be programmed)
N7 (R is 0 mm in CAD system)
N8 (R is 0 mm in CAD system)
N9 (L2=0 circular axes move shortest distance)
N10
N11 G0 X-1 Y=E1 Z0 I1=-1 K1=0
N12 (generated in CAD system)
N13 (front left arc)
N14 G1 X=0 Y=E1 Z=-4 I1=-0.996194698 K1=0.087155743
N15 G1 X=0.000609219 Z=-3.930190374 I1=-0.994521895 K1=0.104528463
N16 G1 X=0.002436692 Z=-3.860402013 I1=-0.992546152 K1=0.121869343
N17 G1 X=0.005481861 Z=-3.790656175 I1=-0.990268069 K1=0.139173101

N... (each degree one point)

N100 G1 X=3.790656175 Z=-0.005481861 I1=0.034899497 K1=0.999390827
N101 G1 X=3.860402013 Z=-0.002436692 I1=0.052335956 K1=0.998629535
N102 G1 X=3.930190374 Z=-0.000609219 I1=0.069756474 K1=0.99756405
N103 G1 X=4 Z=0 I1=0.087155743 K1=0.996194698
N104 (front right arc)
N105 G1 X=36 Z=0 I1=0.087155743 K1=0.996194698
N106 G1 X=36.06980963 Z=-0.000609219 I1=0.104528463 K1=0.994521895
N107 G1 X=36.13959799 Z=-0.002436692 I1=0.121869343 K1=0.992546152

N…

N194 G1 X=39.99756331 Z=-3.860402013 I1=0.998629535 K1=-0.052335956
N194 G1 X=39.99756331 Z=-3.930190374 I1=0.998629535 K1=-0.069756474
N196 G1 X=40 Z=-4 I1=0.996194698 K1=-0.087155743

```
N197 G40
N1971 (back right arc)
N1972 (move up to next cut)
N1973 G174 L100 (tool withdrawal movement)
N1974 G0 B0 C0 (rotate circular tables to original co-ordinate system)
N198 E1=E1+0.25
N1981 G1 Y=E1 (movement in normal X, Y, Z co-ordinate system)
N1982 G141
```

Or without deactivation of G141

```
N197 (back right arc)
N198 E1=E1+0.25 (move up to next cut)


N199 G1 X=40 Y=E1 Z=-4 I1=0.996194698 K1=0.087155743
N200 G1 X=39.99939078 Z=-3.930190374 I1=0.994521895 K1=0.104528463
N201 G1 X=39.99756331 Z=-3.860402013 I1=0.992546152 K1=0.121869343

N…

N287 G1 X=36.13959799 Z=-0.002436692 I1=-0.052335956 K1=0.998629535
N288 G1 X=36.06980963 Z=-0.000609219 I1=-0.069756474 K1=0.99756405
N289 G1 X=36 Z=0 I1=-0.087155743 K1=0.996194698
N290 (back left arc)
N291 G1 X=4 Z=0 I1=-0.087155743 K1=0.996194698
N292 G1 X=3.930190374 Z=-0.000609219 I1=-0.104528463 K1=0.994521895
N293 G1 X=3.860402013 Z=-0.002436692 I1=-0.121869343 K1=0.992546152

N…

N379 G1 X=0.002436692 Z=-3.860402013 I1=-0.998629535 K1=-0.052335956
N380 G1 X=0.000609219 Z=-3.930190374 I1=-0.99756405 K1=-0.069756474
N381 G1 X=0 Z=-4 I1=-0.996194698 K1=-0.087155743
N382 E1=E1+0.25

N383 G14 N1=10 N2=389 J40

N384 G40
N385 G174 L100 (tool withdrawal movement)
N386 G0 B0 C0 (rotate circular tables to original co-ordinate system)
N387 M30
```

5. Tool measurement cycles for laser measurements

5.1 General notes

MC 859 =1: signal type of second stylus (V410 only)

New machine constants

MC 360 -- MC 369 are intended for a second laser measurement device in another work area or an adapter spindle. The area used is determined by the IPLC.

MC 373 free space behind the laser beam in µm

5.2 G600 laser system: Calibration

{I1=..}

```
G    Laser: Calibration
X    Measure point   (optional)
Y    Measure point   (optional)
Z    Measure point   (optional)
S    Speed (rev/min)
I1=  Swing determination 0=no 1=yes
```

DETERMINING CONCENTRICITY ERROR (I1=)

Use address I1 to specify whether the concentricity error is to be measured and saved in the tool table against the calibration tool. It is recommended that the concentricity error should be determined once using a clean calibration stylus.

I1= 0 Do not determine concentricity error (basic setting)
1 Determine the concentricity error

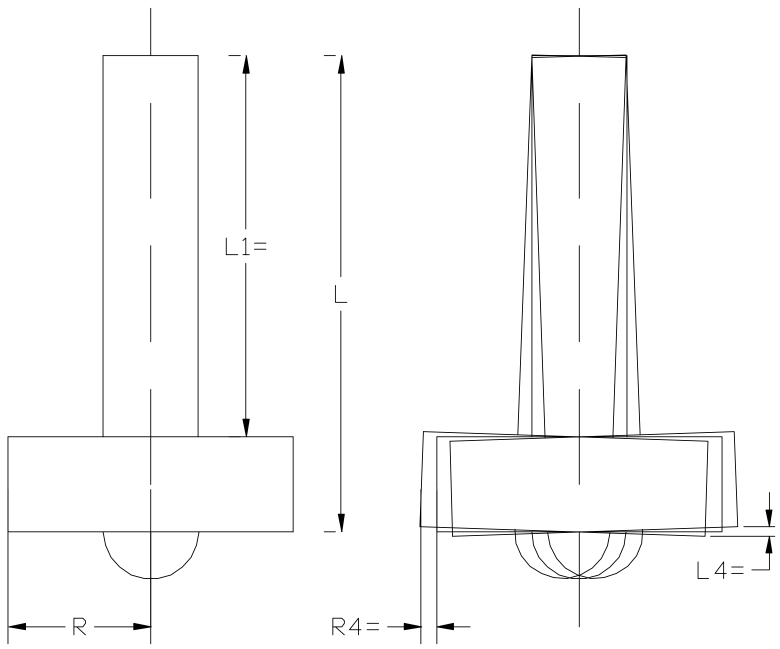
The radial concentricity error is written to the tool memory under R4=.

The axial concentricity error is written to the tool memory under L4= and the length L is reduced by the L4 value. The sum L+L4 remains constant.

CALIBRATION STYLUS, TOOL MEMORY ADDRESSES

The concentricity errors R4 and L4 of the calibration stylus are written to the tool memory by the calibration cycle.

R4= Radial concentricity error of calibration stylus.
L4= Axial concentricity error of calibration stylus.



POSITION OF MEASURING UNIT

When determining the position of the measuring unit for the calibration, the centre of the bottom edge of the pin (dimension L) must be set in the light beam (+/- 5 mm).

- The free machining plane G7 and axis rotation G92/G93 B4 must not be active

SPEED

Coolant will be thrown off by clockwise-anticlockwise-clockwise rotation.

The spindle is switched off with M5 at the end of the cycle.

Example

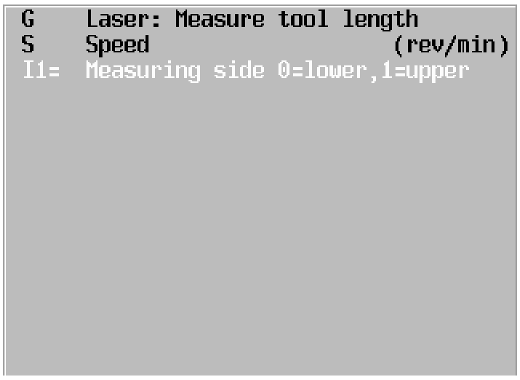
Example: calibration of laser measuring unit, determining concentricity error.

N... G600 X300 Y500 Z600 I1=1 S3000

Concentricity errors L4 and R4 are saved in the tool table, length is matched (I1=1).
The exact X, Y and Z positions are saved in the machine constants.

5.3 G601 laser system: Measuring length

{I1=}



SELECTING THE TOOL EDGE (I1=)

The lower edge or the upper edge of the tool can be measured.

I1= 0 measure lower edge (basic setting)
1 measure upper edge

SPEED

If the spindle is not first switched off (M5 or M19), then:

- Coolant will be thrown off by clockwise-anticlockwise-clockwise rotation.
- The spindle is switched off with M5 at the end of the cycle.

If the spindle is already switched off (M3 or M4), change of direction or spindle stop does not occur at the end of the cycle

TOOL MEMORY ADDRESSES

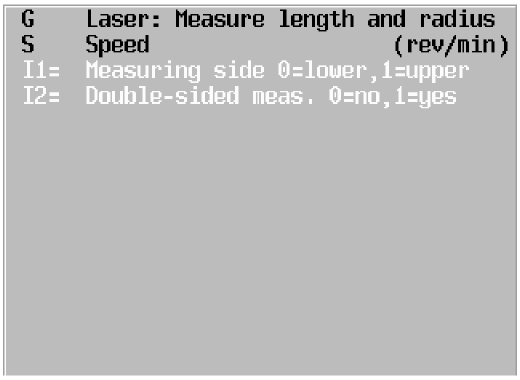
R6= Radius position for measuring length.

LENGTH MEASUREMENT

- If the tool radius is greater than MC373 and R6 is not programmed, the length is measured eccentrically.
- If R6 is programmed and R-R6 > MC373, an error message is issued

5.4 G602 laser system: Measuring length and radius

{I1=..} {I2=..}



SELECTING THE TOOL EDGE (I1=)

The lower edge or the upper edge of the tool can be measured.

I1= 0 measure lower edge (basic setting)

1 measure upper edge

SELECTING MEASUREMENT OF ONE OR BOTH EDGES (I2=)

One or both edges of the tool can be measured.

I2= 0 measure one side (basic setting)

1 measure both sides

When measuring both edges, temperature errors and tool obliquity have no influence on the measured radius.

SPEED

If the spindle is not first switched off (M5 or M19), then:

- Coolant will be thrown off by clockwise-anticlockwise-clockwise rotation.
- The spindle is switched off with M5 at the end of the cycle.

If the spindle is already switched off (M3 or M4), change of direction or spindle stop does not occur at the end of the cycle

LENGTH MEASUREMENT

- If the tool radius is greater than MC373 and R6 is not programmed, the length is measured eccentrically.
- If R6 is programmed and R-R6 > MC373, an error message is issued.

RADIUS MEASUREMENT

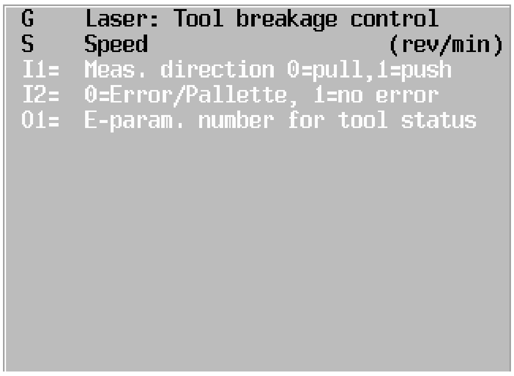
If L6 is greater than MC372, an error message is issued.

5.5 G603 laser system: Individual cutting tip monitoring

If I1+L6 is greater than MC372, an error message is issued

5.6 G604 laser system: Tool breakage monitoring

{I1=..}



MEASURING DIRECTION (I1=)

The measuring direction can be pushing or pulling.

I1= 0 pulling (basic setting)
1 pushing

The fast pulling measurement is preferred, but tools with pronounced concave grinding must be measured pushing, as otherwise the hollow grinding will be detected as a break.

ERROR EVALUATION (I2=)

If a break is detected, various actions can follow:

I2= 0 error message or reject pallet (basic setting)
1 no error message

If I2=0 is selected, function M105 (tool break detected) is issued in the case of tool breakage. The IPLC switches the laser off and the controller issues an error message.

If, however, a pallet system is present, the pallet is rejected if possible, the current program is interrupted and a new pallet is brought in.

If I2=1 is selected, no error message is issued on tool breakage. Every action must be programmed in the part program. To achieve this, the tool status (value E from the tool memory) can be written directly to an E parameter. See address O1.

TOOL STATUS OUTPUT TO E PARAMETER (O1=)

The tool status (definition E in the tool memory) is written to the specified E parameter. Based on this parameter, the program can determine whether a tool breakage has been detected (status –4). This is only meaningful if the error message has been switched off with I2=1.

SPEED

If the spindle is not first switched off (M5 or M19), then:
- Spindle is switched on clockwise (M3).
- The spindle is switched off with M5 at the end of the cycle.

If the spindle is already switched off (M3 or M4), spindle stop does not occur at the end of the cycle.

TOOL STATUS

The basic setting for tolerance B is entered in MC33. Only 1 or 2 mm is possible. The setting of MC133 is in mm even in inches mode.

BREAKAGE MEASUREMENT

If the tool radius is greater than MC373 and R6 is not programmed, the length is measured eccentrically.

If R6 is programmed and R-R6 > MC373, an error message is issued

6. Specific G functions for macros

6.1 G300 Programming error messages

D are general milling error messages (P), D1= are error messages (R) in turning mode (G38).

6.2 G302 Overwriting radius correction parameters

The G302 function overwrites the active tool parameters during execution. The tool parameters in the tool memory are not changed.

In this version, only the O parameter for tool orientation can be overwritten.

6.3 G303 M19 with programmable direction

M19 with programmable direction (CW or CCW).

Format

N... G303 M19 D... {I2=...}

Parameters

G	M19 with programmable direction
D	Angle oriented spindle stop
M	Machine function
I2=	Direction 3=CW 4=CCW

Notes and application

Only the M19 function can be programmed.
The basic setting for I2=3.

Example

Stopping spindle with M19.
N100 G303 M19 D75 I2=4

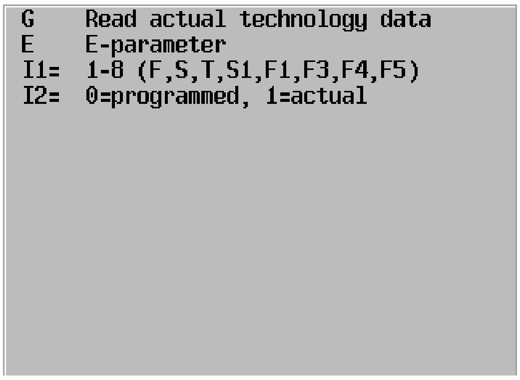
Explanation:
N100: Oriented spindle stop
Angle 75 degrees
CCW

6.4 G319 Query active technology

S1 (cutting speed/rotational speed) or T (tool number).

Format

N... G319 I1=.. E... {I2=..}



Selectable functions:

I1=4 Cutting speed/rotational speed (S1=) (turning only)
I1=5 Constant cutting feed (F1= with G41/G42)
I1=6 Plunge feed (F3=)
I1=7 Planar feed (F4=)
I1=8 Circular axis feed (F5=)

I2=0 Programmed value (basic setting)
I2=1 Current value.

Notes and application

READING ADDRESS WITHOUT VALUE

If the address does not exist, the E parameter is filled with –999999999.

6.5 G320 Query current G data

Query address value of current modal G function and save this value in the E parameter provided for this purpose.

Format

N... G320 I1=.. E...

Parameters

```
G    Read actual G data
E    E-parameter
I1=  Selection number
```

Type of function

Non-modal

Notes and application

BASIC SETTINGS

All values are initialised when the machine is started. Most values are set to zero.

READING ACTIVE MODAL G_FUNCTIONS

G324 can be used to query whether a G function is active.

Particular information can always be queried with G320.

UNIT OF RESULT

The unit of the result is mm or inches for positions. Degrees for angles.

SELECTION NUMBER

G function	I1= selection number		Result min—max.	Basic setting
G7	Swing machining plane			
	1	A axis solid angle	-180--180°	0
	2	B axis solid angle	-180--180°	0
	3	C axis solid angle	-180--180°	0
G8	Swing tool			
	4	A axis solid angle	-180--180°	0
	5	B axis solid angle	-180--180°	0
	6	C axis solid angle	-180--180°	0
G9	Pole point (define dimension reference point)			
	7	X axis polar co-ordinate		0
	8	Y axis polar co-ordinate		0
	9	Z axis polar co-ordinate		0
Result of G17, G18, G19, G180 and G182				
	10	First main axis	1--3	
	11	Second main axis	1--6	
	12	Tool axis	1--3	
			1=X, 2=Y, 3=Z, 4=A, 5=B, 6=C	
G25	Feed and spindle override effective			
	13	Feed and spindle override active	0	
G26	Feed and spindle override not effective			

	13	Feed and spindle override not active	1--3 1=F=100%, 2=S=100%, F and S=100%	
G27	Positioning functions			
	14	Feed motion (I3=)	0	
	15	Rapid motion (I4=)	0	
	16	Positioning logic (I5=0)	0	
	17	Acceleration reduction (I6=)	100%	
	18	Contour accuracy (I7=0)	MC765	
G28	Positioning functions			
	14	Feed motion (I3=)	0--1	
	15	Rapid motion (I4=)	0--1	
	16	Positioning logic (I5=0)	0--1	
	17	Acceleration reduction (I6=)	5—100%	
	18	Contour accuracy (I7=0)	0—10.000µm or MC765	
G39	Activate/deactivate allowance			
	19	Length allowance (L)		0
	20	Radius allowance)R)		0
G52	Pallet zero point displacement			
	21	Zero point displacement in X axis		0
	22	Zero point displacement in Y axis		0
	23	Zero point displacement in Z axis		0
	24	Zero point displacement in A axis		0
	25	Zero point displacement in B axis		0
	26	Zero point displacement in C axis		0
G54	Standard zero point displacement			
	27	Zero point displacement in X axis		0
	28	Zero point displacement in Y axis		0
	29	Zero point displacement in Z axis		0
	30	Zero point displacement in A axis		0
	31	Zero point displacement in B axis		0
	32	Zero point displacement in C axis		0
	33	Rotation angle	-180--180°	0
G92/G93	Incremental or absolute zero point displacement			
	34	Zero point displacement in X axis		0
	35	Zero point displacement in Y axis		0
	36	Zero point displacement in Z axis		0
	37	Zero point displacement in A axis		0
	38	Zero point displacement in B axis		0
	39	Zero point displacement in C axis		0
	40	Rotation angle	-180--180°	0
Total zero point displacement (G52 + G54 + G92/G93)				
	41	Zero point displacement in X axis		0
	42	Zero point displacement in Y axis		0
	43	Zero point displacement in Z axis		0
	44	Zero point displacement in A axis		0
	45	Zero point displacement in B axis		0
	46	Zero point displacement in C axis		0
	47	Rotation angle	-180--180°	0
G72	Activate mirroring and dimension factor			
	48	Plane scale (factor or %) (A4=)		1
	49	Tool axis scale (factor or %) (A4=)		1
	50	Mirroring in X axis		1
	51	Mirroring in Y axis		1
	52	Mirroring in Z axis		1
	53	Mirroring in A axis		1
	54	Mirroring in B axis		1
	55	Mirroring in C axis		1
G73	Activate mirroring and dimension factor			
	48	Plane scale (factor or %) (A4=)	0—100% or factor	
	49	Tool axis scale (factor or %) (A4=)	0—100% or factor depending on MC714	
		MC714	0= machining plane factor 1= machining plane percent 2= all linear axes factor 2= all linear axes percent	
	50	Mirroring in X axis	-1--1	
	51	Mirroring in Y axis	-1--1	

52	Mirroring in Z axis	-1--1
53	Mirroring in A axis	-1--1
54	Mirroring in B axis	-1--1
55	Mirroring in C axis	-1--1

System axis number determined by machine constants (MC103, MC105, etc.)

56	X axis	0--6 0= not active, 1—6 axis number For example, information about axis number 1 is located in the MC3100 and MC4700 row, etc.
57	Y axis	0--6
58	Z axis	0--6
59	A axis:	0--6
60	B axis	0--6
61	C axis	0—6

Examples

Querying current G data and saving the values in the E parameter.
N11 G320 I1=10 E11
N12 G320 I1=11 E12
N12 G320 I1=12 E13

Explanation:

N11: I1=10 Query first main axis
E11 contains the result
E11 =1 X axis is the first main axis.

N12: I1=11 Query second main axis
E12 contains the result
E12 =2 Y axis is second main axis

N13: I1=12 Query tool axis
E13 contains the result
E13 =1 Z axis is the tool axis.

6.6 G325 Query current modal M function

```
G    Read M-group
E    E-parameter
I1=  M-group (1,2,etc.)
```

Group division

Group	
11	off, M56, M57, M58
12	off, M72, M73
13	off, M1=..

SIGNIFICANCE OF M FUNCTIONS

Some of these M functions are basic M functions and are described in the M functions chapter. The others are machine-dependent M functions. Please refer to the machine builder's manual for a description.

6.7 G326 Query current axis position values

{I1=..} {I2=..}

Parameters

```
G    Read actual position
X7=  E-parameter for X-position
Y7=  E-parameter for Y-position
Z7=  E-parameter for Z-position
A7=  E-parameter for A-position
B7=  E-parameter for B-position
C7=  E-parameter for C-position
I1=  0=Workp. 1=Machine 2=RPF
I2=  0=programmed, 1=actual
```

I1=	0	Position to workpiece zero point (basic position)
	1	Position to machine zero point
	2	Position to reference point
I2=	0	Programmed value (basic setting)
	1	Current value.

Notes and application

QUERYING NON-EXISTENT AXES

If the axis does not exist, the E parameter is filled with –999999999.

6.8 G329 Query programmable kinematic elements

Query programmable kinematic elements and save these values in the E parameter provided for this purpose.

Format

N... G329 N1=.. E...

Parameters

G	Read offset from kinematic model
E	E-parameter
N1=	Progr.kinematic element nr.(1-10)

Notes and application

PROGRAMMABLE KINEMATIC ELEMENTS

A kinematic element is defined by a group of 4 machine constants.
The machine builder can enter whether a particular kinematic element is programmable. To do this, the third machine constant of the group (MC602, MC606, etc.) must have the value 2.
The values of these programmable kinematic elements can be programmed using G339. In doing this, the value of the fourth machine constant of the group (MC603, MC607, etc.) is overwritten.

PROGRAMMABLE KINEMATIC ELEMENT NUMBER

Defines the number of the programmable element of the kinematic model to be queried. The value lies between 1 and 10.

Examples

Read a programmable kinematic element and save the value in the E parameter.
N... G329 N1=1 E10

Explanation:

E10 contains the contents of programmable kinematic element 1 (in mm or inches).

6.9 G339 Write programmable kinematic elements

Write programmable kinematic elements.

Format

N... G339 N1=.. E... {I1=...}

Parameters

```
G    Write offset in kinematic model
E    E-parameter
N1=  Progr.kinematic element nr.(1-10)
I1=  Write mode         (0=inc, 1=abs)
```

Notes and application

PROGRAMMABLE KINEMATIC ELEMENTS

A kinematic element is defined by a group of 4 machine constants.
The machine builder can enter whether a particular kinematic element is programmable. To do this, the third machine constant of the group (MC602, MC606, etc.) must have the value 2.
The values of these programmable kinematic elements can be programmed using G339. In doing this, the value of the fourth machine constant of the group (MC603, MC607, etc.) is overwritten.

PROGRAMMABLE KINEMATIC ELEMENT NUMBER

Defines the number of the programmable element of the kinematic model to be written. The value lies between 1 and 10.

NOTATION

“Incremental” notation (basic setting) means that the programmed value is added to the existing value.
“Absolute” notation means that the existing value is overwritten by the programmed value.
The programmed values remain in the kinematic model and are not reset following M30, <Cancel program> or <Reset CNC>.

Example: Write a programmable kinematic element.

A circular table is defined in the kinematic model. This circular table is defined by two kinematic elements for each X axis. The first is defined by the machine builder and determines the position of the circular table. The second is a programmable element. Using this, the exact position can be corrected in the kinematic model after measurement.
N100 G145... (measure)
N105 (calculate all parameters)
N110 G339 N1=1 E10 I1=1

Explanation: N100 Measuring the exact position.
N110 The contents of E10 are written to the first programmable kinematic element.

6.10 Formatted write functions

Previously, only write functions to the internal memory were possible.

Using the formatted write function, it is now possible:
- to write to the screen
- to write to the file on the hard disc
- to fill an array
- to read a number from an array.

Configuration file

Configuration files are required for these actions to describe how and where to write or read. These configuration files are saved on the hard disk and are read when the system is started.

Two configuration files are possible.

1) File to define an array and fill it with basic settings.
   ARRnnnnn.CFG
   nnnnn is the file number from 1 to 99999.
2) File to define a print file.
   D:\STARTUP|CYCLES\FORMnnnn.CFG.
   nnnn is the file number from 1 to 9999.

6.10.1 File to define an array and fill it with basic settings.

An array is defined with a configuration file. This is activated when the system is started.
A maximum of 10 arrays can be defined. End users can define files themselves.
The maximum array size is 5000 elements.
An element can be read from the array with arrayread(nnnnn, row, column).
If an element is read that does not exist, the value 999999999 is returned.

Description of array configuration file:

```
;Comments start with ';'
;
;Sections:
;[element]                                  defines an element in the array
;row    =   row number                      where row number = [1|...|9999]
;col    =   column number                   where column number = [1|...|9999]
;                                           row * column <= 5000
;val    =   value                           where value = real number (double)
;
```

Example: Array configuration file:

```
ARRnnnnn.CFG

[element]
row     =   1
col     =   1
val     =   0                   ; element (1,1).=.0

[element]
row     =   3
col     =   66
val     =   397.01              ; element (3,66) = 397.01

 [element]
row     =   9999                ;maximum row size
col     =   9999
val     =   -123456789.123456789
```

E parameter array

Several arrays can be filled using a configuration file. These arrays can be read with E parameters during execution. For unbalance detection, the calibration curves are read and interpolated in this manner.

arrayread (arraynumber, row, column)
Where:

arraynumber	is the number of the array Every array has its own configuration file. Arraynumber between 1 and 89999.
Row	is the row number in the array that is to be read. Row between 1 and 999999.
Column	is the position in the row of the array that is to be read. Column between 1 and 999999.

Fixed arrays can be read with the arrayread function. The arrays are filled from a configuration file D:\STARTUP\CYCLES\ARRnnnnn.CFG).

Empty 'elements' in the array have the value <–999999999>.

Example arrayread
E300 = arrayread(100,1,2)
E300 has the value of array 100, row 1, column 2.

6.10.2 Configuration file to define a file or window (display/input)

A print file is defined with a configuration file. This is activated when the system is started.
A maximum of 10 files can be defined. End users can define files themselves.
The file size is unlimited.

Description of print file configuration file:

```
:Commentary starts with a ';'
;
;Sections:
Only for one window:
;[window]                                defines present window
;number=             windowId            where windowId =   1 = output, centre, 5x40
;                                                           2 = input, centre, 1x40;
;                                                           3 = graphic, above dashboard

;[file]                                  defines file (only for G351')
;name      =         file name           where filename is 8.3 ASCII characters
;                                        The directory is always D:\STARTUP\
;

;[string]                                defines position and content of the block
;line      =         line number         where line number = [1|...|n]   basic setting = 1
;position=           position number     where      position number = [1|...|n]   basic setting = 1
;gb        =         "string"            where block is <n> ASCII characters
;d=                  "string"
;                                        Texts are defined for various languages
;                                        Code gb=, d=, f= .. or defined independent of language with: txt =
;
;[value]                                 defines position, format and E parameter of the value
;line      =         line number
;position=           position number
;eparam=             E parameter         where E parameter number = [1|...|1250]
;form      =         digit decimal       where digit decimal = <digits>.<decimals>
;sign      =         yesNo               where yesNo =      y = space for sign
;                                                           n = no space for sign
;
Only for input window:
;[input]                                 defines position, format and E parameter before an input field
;                                        only for G350 and windowId = 2
;                                        only one [input] section is allowed
;line      =         line number
;position=           position number
;eparam=             E parameter number        where E parameter number = [1|...|1250]
;form      =         digitDecimal              where digitDecimal = <digits>.<decimals>
;sign      =         yesNo                     where yesNo =      y =space for sign
;                                                                 n = no space for sign
```

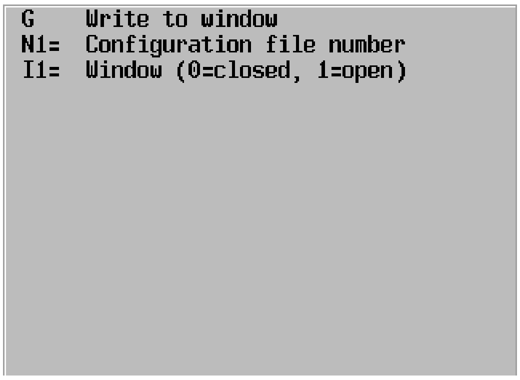
6.10.3 G350 Writing to a window

Specific lines and values can be written to a window using E parameters and a configuration file. In addition, a particular input can be expected. For unbalance detection, the result can be displayed to the operator in this way.

Format

N... G350 N1=.. {I1...}

Parameters



N1= Defines the configuration file 'D:\STARTUP\CYCLES'\FORMnnnn.CFG> that is used for the format, lines and E parameters that are written. File number between 1 and 8999.

I1= 0 = window not visible. Setting on switch-on:
1 = window visible.

Notes and application

G350 can be used to make a previously defined window visible. The texts in the window are fixed, the values are continuously updated according to the defined E parameters.
When an input field is defined, the program waits until the entry is made and <Start> is pressed. Only one entry window can be active at any one time.

Up to now, 3 windows are defined:

Number	Window type	Mode	Position	Size
1	Display	Manual Automatic	Right side of screen Top 'Dashboard'	14 lines, 35 characters
2	Input	Manual Automatic	Right side of screen Top 'Dashboard'	1 line, 35 characters
3	Graphics	Manual Automatic	Up to machine function keys Top 'Dashboard'	14 lines, 70 characters

Please see also the configuration file.

The window also appears in graphics, but not during block search.
the window becomes invisible following M30 and <Cancel program>.

Example Writing to a window

```
N1 E300=8            Hole number
N2 E301=5            Number
N3 E350=6            Suggested input value

N10.. G350 N1=6789   Write to window
                     File D:\STARTUP\CYCLES\FORM6789.CFG is used
N

; The window is:
```

```
+----------------------------------+
¦Drilling pattern                  ¦
¦----------------------------------¦
¦Holes    8   Number   5           ¦
¦                                  ¦
¦Restart at number      ¦¦¦        ¦
+----------------------------------+
```

E350 is changed following user input.

Display window configuration file
FORM6789.CFG

```
[window]
number    =      1          ;Uses window number 1 of the available windows.

[string]
line      =      1
gb        =      "drilling pattern"
d         =      "Bohrbild"

[string]
line      =      2
gb        =      '---------------------------------'
d         =      '---------------------------------'

[string]
line      =      3
gb        =      "holes"
d         =      "Locher"

[value]
line      =      3
position  =      8          ;Print value in field at position 8 and onwards
eparam    =      300        ;E parameter E300 is given the value
form      =      3.0        ;format 3 figures and 0 decimals
sign             n          ;No space reserved for sign

[string]
line      =      3
position  =      17         ;Print value at position 17 and onwards
gb        =      "number"
d         =      "Nummer"

[value]
line      =      3
position  =      26         ;Print value in field at position 26 and onwards
eparam    =      301        ;E parameter E301 is given the value
form      =      3.0        ;format 3 figures and 0 decimals
sign      =      n          ;No space reserved for sign

[string]
line      =      4
gb        =      '---------------------------------'
d         =      '---------------------------------'

[string]
line      =      5
gb        =      "restart at number:"
d         =      "Neustart bei Nummer:"

[input]
line      =      5
position  =      26         ;Define input field at position 26 and onwards
eparam    =      350        ;E parameter E350 suggests an input value
                            ;and later received the operator input
form      =      3.0        ;format 3 figures and 0 decimals
sign      =      n          ;No space reserved for sign
```

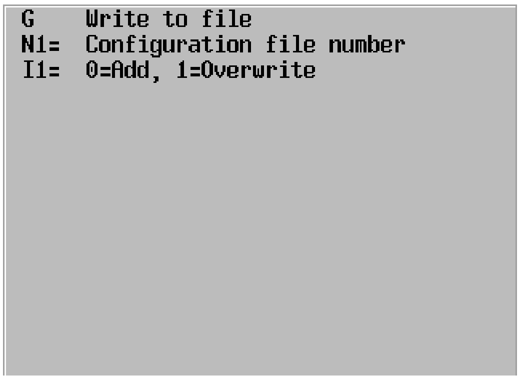
6.10.4 G351 Writing to a file

Specific lines and values can be written to a text file in D:\Startup\ using E parameters and a configuration file. This can be used to create the calibration curves for unbalance detection.

Format

N... G351 N1=.. {I1=...}

Parameters



N1= Defines the configuration file <'Directory'\FORMnnnn.CFG> that is used for the format, lines and E parameters that are written. File number between 1 and 9999.

The directory can be any 'Cycle Design' directory.

The configuration file is the same as for writing to a window, but ‘section’ [window] and [input] are ignored.

I1= States whether the data is to be inserted at the end of an existing file or whether a file that may exist is to be overwritten. Basic setting <0> for insertion.

Notes and application

G351 writes the lines and values of the configuration file and E parameters to the hard disk.
A maximum of 50 lines of 255 characters can be written at the same time.
The file is not written during graphics or block search.

Example Listing measurement data and writing to a file.

The radius of a pocket is measured in the program

The following data available in the E parameters must be listed:
N10 (measurement programmed in blocks N12 to N16)
N11 (in this case as example of just the results from e.g. measurement cycle G145)

N12 E50=34.1	(setpoint)	(entered)
N13 E51=34.05	(lower tolerance)	(entered)
N14 E52=34.15	(upper tolerance)	(entered)
N15 E53=34.108	(actual value)	(measured)
N16 E54=0.008	(difference)	(calculated)

N20 G351 N1=0002 I1=0 (write file)
File D:\STARTUP\CYCLES\FORM0002.CFG is used.
I1=0 is insert

The file messdat.txt is:

Radius
Setpoint = 34.1
Lower tolerance =34.5
Upper tolerance =34.5

Actual value = 34.108
Difference = 0.008
****

Configuration file for listing measurement data

FORM0002.CFG

```
;*******************************************************************
; CFG file for writing measurement data
;*******************************************************************

;---- Name of file to be written to startup\ --------
[file]
name            = Messdat.txt

;---- Type of measurement ------------------------------
[string]
line            = 1
position        = 1
d               = Radius

;---- Setpoint --------------------------
[string]
line            = 2
position        = 1
d               = Sollwert =

[value]
line            = 2
position        = 20
eparam          = 50
form            = 4.3
sign            = y

;---- Lower tolerance --------------------------
[string]
line            = 3
position        = 1
d               = Untere Toleranz =

[value]
line            = 3
position        = 20
eparam          = 51
form            = 4.3
sign            = y

;---- Upper tolerance --------------------------
[string]
line            = 4
position        = 1
d               = Obere toleranz =

[value]
line            = 4
position        = 20
eparam          = 52
form            = 4.3
sign            = y

;---- Actual value --------------------------
[string]
line            = 5
position        = 1
d               = Istwert =

[value]
line            = 5
position        = 20
eparam          = 53
form            = 4.3
sign            = y
```

```
;---- Difference --------------------------
[string]
line            = 6
position        = 1
d               = Differenz =

[value]
line            = 6
position        = 20
eparam          = 54
form            = 4.3
sign            = y

;---------------------------------------
[string]
line            = 7
d               = *****************************************************
```

7. Turning

7.1 Introduction

The turning mode has been developed for machines with a C axis that can turn continuously. In this way, turning operations can be carried out on a milling machine.

The C axis can be switched to turning mode. The C axis is then programmed as a turning spindle via S1= and M1=. The turning tools are mounted in the milling spindle and clamped at the desired orientation.

In special cases, the milling spindle can be programmed parallel to the turning spindle via S and M. A second milling spindle is not possible on machines with turning mode.

Notes and application

AVAILABILITY

Machine and CNC must be prepared for turning mode by the machine builder. If your machine is not equipped with all the G functions described here, please refer to your machine manual.

GRAPHICS

The graphic is not displayed symmetrically to the rotation.

DISPLAY

If G36 is active, the display of the C axis position changes to display S1=.
The machining status is expanded with G36/G37.
The spindle power display shows the power of the second spindle even if both spindles are active.

REFERENCE POINT

When the controller runs up, it is always in milling mode G37. The C axis can only be switched to turning mode after the reference points have been approached.

ZERO POINT

In turning mode, the workpiece zero point in X should lie in the centre of rotation of the S1 axis. It is recommended that the workpiece zero point in Y should also lie in the centre of rotation of the S1 axis.

SPINDLE OVERRIDE

Spindle override is effective for both spindles in turning mode (G36).

7.2 Machine constants

New machine constants

MC 314 Turning mode (0=off, 1=on)
Activated:
- G functions G36 and G37
- Turning cycles
- Machine constants MC2600 - MC27xx, MC45xx

MC 450 Balancing: measurement axis (1=X, 2=Y, 3=Z)
This MC determines the axis on which the rotary table is installed. Unbalance is easiest to measure in this axis. Normally, 2 = Y axis
The MC is used in the ‘unbalance calibration’ (installation), G691 ‘unbalance detection’ and G692 ‘unbalance checking’ cycles.

MC 451 Balancing: maximum amplitude [µm]
This MC specifies the permissible residual amplitude in the measuring axis. The measurement is cancelled if the measured amplitude is greater than MC451 at a particular speed. Normally 5 [µm].
The MC is used in the ‘unbalance calibration’ (installation), G691 ‘unbalance detection’ and G692 ‘unbalance checking’ cycles. The C1 parameter can be superimposed on this in the G691 and G692 cycles.

MC 452 Balancing: initial radial position [µm]
This MC specifies the radial position (distance from centre point) of the rotary table (S1 axis) at which a balancing mass is normally mounted to compensate for unbalance.
The MC is used in the G691 ‘unbalance detection’ cycle.

MC 453 Balancing: rotary table displacement [mGrad]
This MC specifies the 0 position of the rotary table and the position (door) where the operator fits the mass to compensate (and calibrate) the unbalance.
The MC is used in the ‘unbalance calibration’ (installation) and G691 ‘unbalance detection’ cycles.

Existing machine constants

MC 268 second spindle (0=no, 1=yes)
MC2600 - MC2799, MC4500 - MC4599 second spindle

Remark: The inverter output of the 2nd spindle (MC2600) should be equal to the inverter output of the C axis (MC3400).

7.3 G36/G37 Switching turning mode on and off

G36 Switches the machine from milling mode on the C axis to turning mode with turning spindle S1.
G37 Terminates turning mode. Switches the machine back to milling mode.

Format

N... G36 or N... G36

Parameters

none.

Type of function

modal

Notes and application

G36

The CNC switches the C axis to turning mode.
In turning mode, the circular axis is programmed as a second spindle using S1= and M1=. C parameters can no longer be programmed.
The display of C (setpoint and actual value) on the screen is switched to S1. If the turning spindle is stationary, the position (0-359.999 degrees) is displayed.
G95 is active, assigned to the second spindle.
All G functions can be programmed, but not all the G functions are meaningful. For instance, a pocket has no meaning in turning mode. The C parameters and certain other parameters can no longer be programmed in certain G functions.
The effect of G36 remains active until it is cancelled by G37, runup or <CNC reset>. G36 is not cancelled by M30 or <Cancel program>.

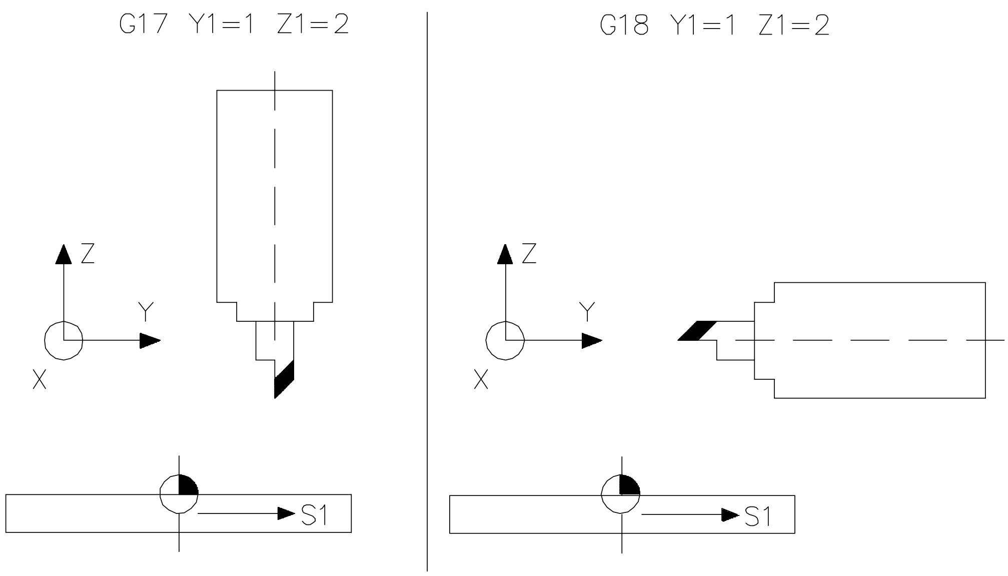
G37

The CNC switches the C axis on again.
If the rotary spindle is still turning at the start of G37, it is first stopped.
The position of the circular axis is displayed on the screen with a value between 0 and 359.999 degrees.
G94 becomes active.
The effect of G37 remains active until it is cancelled by G36. G37 is not cancelled by M30 or <Cancel program>. G27 is always active following runup or <CNC reset>.

7.4 G17 Plane for turning mode (G17 Y1=1 Z1=2)

In turning mode, the machine is in G17 or G18. The determines the direction of tool length correction (Z direction in G17, Y direction in G18).

In turning mode, machining takes place in the YZ plane. These planes are automatically taken into account in the turning cycles. The special machining plane is indicated in G17 by Y1=1 (first main axis) and Z1=2 (second main axis).



Angles (positive) and circular directions (CW) are defined from the Y axis to the Z axis.

Remark: The special plane must be reset at the end of turning mode by programming a ‘normal’ G17 or G18 without addresses.

7.5 G94/G94 Expanded choice of feed unit

Informs the CNC how to evaluate the programmed speed (S).
This function is expanded for turning mode.
The spindle and the circular table must be programmed for turning.

Notes and application

In addition, the rotary table (second spindle) must be programmed with S1= and M1= for turning.

In milling mode (G37): N... G95 F.. {S..} {M..}
In turning mode (G36): N... G95 F.. {S1=..} {M1=..}

S and M refer to the spindle
S1= and M1= refer to the second spindle

PRIORITY

The active spindle speed is either S or S1=. If S and S1= are both programmed, S1 is used.

MAXIMUM SPEED

The value of the second spindle speed (S1=) lies between 0 and ‘Max. output voltage speed’ (MC2691).

MACHINE FUNCTION

Second spindle machine functions:

M1=3	second spindle clockwise
M1=4	second spindle anticlockwise
M1=5	second spindle stop

Positioning of the second spindle (M1=19) is not possible. Positioning takes place in milling mode.

The S1= and M1= addresses can also be programmed in the following G functions: G0, G1, G2, G3, G94.

The G95 function calculates the feed in [mm/min (inches/min)] based on the programmed feed in [mm/rev], [inches/rev] and the active spindle speed.

7.6 G96/G97 Constant cutting speed

G96 Programming constant cutting speed.
G97 Switching off constant cutting speed.

Format

N... G96 F.. D.. {S..} {M..} {S1=..} {M1=..}
N... G97 F.. {S..} {M..} {S1=..} {M1=..}

Parameters

```
G    Konstante Schnittgeschwindigkeit
D    Obere Drehzahlgrenze (U/min)
F    Vorschub
S    Schnittgeschwindigkeit (m(f)/min)
M    Maschinenfunktion
S1=  Schnittgeschwindigkeit (m(f)/min)
M1=  Maschinenfunktion
```

G96

```
G    Spindeldrehzahl
S    Spindeldrehzahl (U/min)
M    Maschinenfunktion
S1=  Spindeldrehzahl (U/min)
M1=  Maschinenfunktion
```

G97

S and M refer to the spindle
S1= and M1= refer to the second spindle (rotary table)

Type of function

modal

Notes and application

MAXIMUM SPEED (D)

The value of the second spindle speed lies between 0 and ‘Max. output voltage speed’ (MC2691).

MACHINE FUNCTION

Second spindle machine functions:

M1=3 second spindle clockwise
M1=4 second spindle anticlockwise
M1=5 second spindle stop

Positioning of the second spindle (M1=19) is not possible. Positioning takes place in milling mode.
The G96 function calculates the feed in [mm/min (inches/min)] based on the programmed feed in [mm/rev], [inches/rev] and the active spindle speed.
The active spindle speed is either S or S1=. If S and S1= are both programmed, S1 is used.

7.7 Defining turning tools in the tool table

Tool correction and orientation

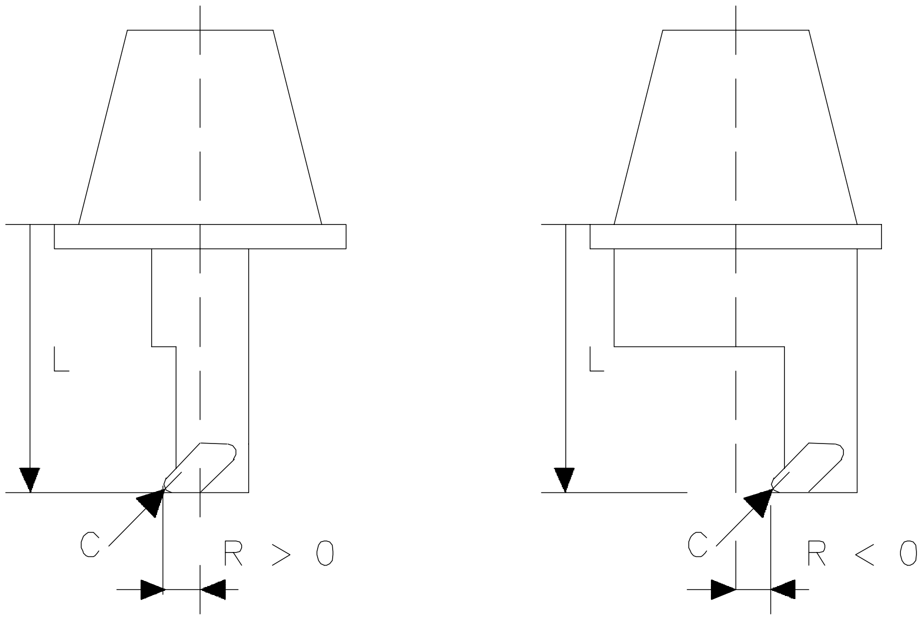
The tool dimensions are stored as tool length L, tool radius R and tool corner radius C. The tool radius correction refers to the corner radius C and the necessary tool orientation is entered in address O in the tool memory.

Tool memory

The turning tools can be placed in any position in the tool magazine. The tool is marked as a turning tool by parameter Q3= 'tool type'. This also locks the spindle.

Q3= 'tool type' = 8xx turning tool.

Q3 is taken into account by the PLC. Please see the machine manual for further information.



Tool dimensions for turning tools.

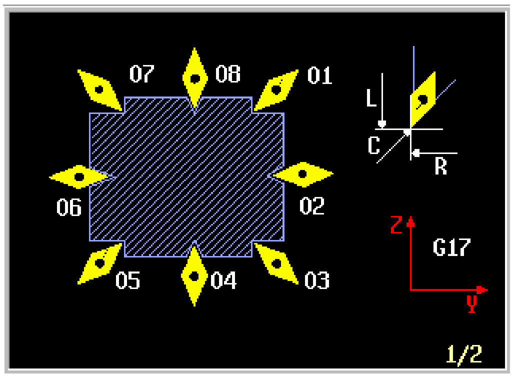
If turning mode is active (or in general, when the main plane is parallel to the tool axis), the radius R is regarded as a displacement. The radius correction is also calculated in this case using the tool corner radius C and the orientation O. If the orientation is in the negative direction, the tool radius is also calculated as a negative displacement.

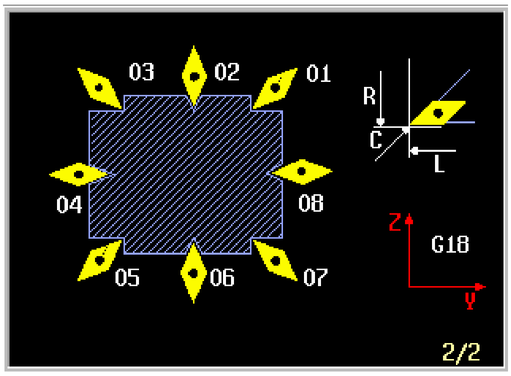
Level	**Orientation**	**Radius correction**	**Radius as displacement**
G17	not active	R	not active
G17 Y1=1 Z1=2	1, 2, 3, 4, 8	C and O	R in negative Y direction
G17 Y1=1 Z1=2	5, 6, 7	C and O	R in positive Y direction
G18	not active	R	not active
G18 Y1=1 Z1=2	1, 2, 6, 7, 8	C and O	R in negative Z direction
G18 Y1=1 Z1=2	3, 4, 5	C and O	R in positive Z direction

The orientation must be defined in the tool memory for G17 Y1=1 Z1=2 (vertical operation). The controller adapts the active orientation when G18 Y1=1 Z1=2 (horizontal operation) is activated.

7.8 G302 Superimposing tool data

The G302 function determines the tool orientation during execution. The tool parameters in the tool memory are not changed.

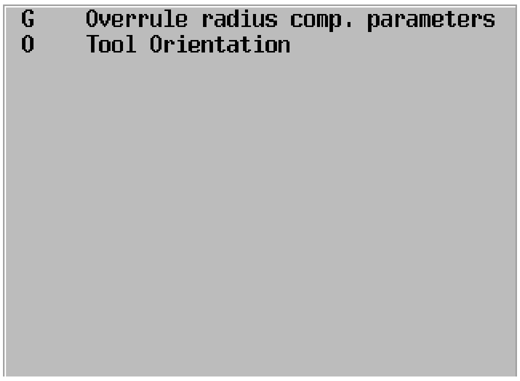




Format

N.. G302 O

Parameters



O Defines the tool orientation used during execution.
The value lies between 0 and 8.

Type of function

Non-modal

Notes and application

Remarks:

If the active tool orientation is overwritten, the direction of the R displacement may also change.
In G18, the active tool orientation is already changed by the CNC. See chapter 'Tool correction'.

USE

The G302 function should be used if. for example, the main spindle has been turned through 180 degrees with M19 D90. In this case, the orientation is mirrored compared with the status with M19 D90. The orientation should also be mirrored when turning takes place 'across the centre'.
Remark: In these cases, the direction of rotation of the 2nd spindle should also be reversed.

DELETING

G302 is switched off again with G302 without parameter, set plane (G17, G18, G19), tool change, M30 and <Cancel program>.

7.9 Unbalance cycles

7.9.1 General information

To machine workpieces to be turned on an FP machine, both the machine (rotary table) and workpiece must be balanced, otherwise the life of the machine, the quality of the workpiece or even the safety of the operator cannot be guaranteed.

First, the unbalance properties of the rotary table must be determined. Usually, this unbalance calibration takes place when the machine is handed over or during servicing.

To determine the unbalance of the clamped workpiece, a new cycle has been introduced: **G691 unbalance detection**.
This cycle can be called up directly in manual mode under the FST menu.
The result is a suggestion for compensating for the measured unbalance: what mass should be attached at what radial position from the turning centre. The rotary table is automatically turned to the position where the mass should be attached.
The radial position for an available compensating mass can be calculated in the dialog window. The relationship between mass and position are shown graphically.

To ensure that no turning operations take place in automatic mode with too great an unbalance, a new G function can be called in the program: **G692 unbalance check**.
This G function checks the unbalance present against the permissible unbalance. If this is exceeded, an error message is issued, following which the operator can cancel the automatic mode and carry out a new unbalance detection with correction in manual mode.

7.9.2 Description of unbalance

When working in turning mode, centrifugal forces occur if the clamped part (e.g. a pump housing) has an unbalance. This influences concentric accuracy because the second spindle (= circular axis C) is configured on the Y axis.

Unbalance U = m . R

where:

m	= mass	[g]
R	= distance form centre of mass to centre of table	[mm]

The unbalance is given in [gmm} (grammes*mm). This means that 500 [grammes] at 300 [mm] (=150000 [gmm]) has the same effect as 1000 [grammes] at 150 [mm].

The centrifugal force is proportional to the unbalance and rises quadratically with rising speed.

Centrifugal force Fc = m . R : 1000000 . (S . 2 . PI : 60) ^ 2

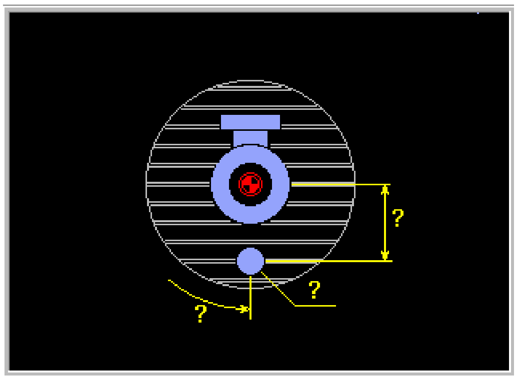
where:

Fc	= centrifugal force	[N]
m	= mass	[g]
R	= distance form centre of mass to centre of table	[mm]
S	= speed	[rpm]

The unbalance must be compensated by a balance weight. For this purpose, the available measuring systems of the circular axis C and the linear axis Y are used to detect the unbalance that exists.

7.9.3 G691 Measure unbalance

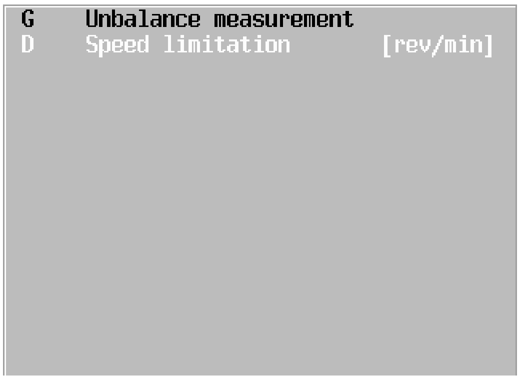
This cycle calculates the instantaneous unbalance. It gives the operator a suggestion how to compensate for the unbalance. This cycle should be called after each clamping operation and after milling mode.



Format

N.. G691 {D..}

Parameters



D	Maximum speed for terminating the measurement
Basic setting MC2691 ‘maximum speed’
Minimum value 50 [rpm]
The speed limit should be at least as high as the programmed speed for turning machining.

Type of function

Non-modal

Associated functions

G692

Notes and application

When detecting unbalance, the position error of the linear axis is measured with rising speed. The speed is increased in steps of 25 rpm. When the position error has reached the maximum value (MC451) or the maximum speed has been reached, the measurement is terminated. The unbalance is calculated from the measured error and the stored calibration data.
The unbalance (gmm) and compensation position (degrees) are displayed. This position is approached at the end of the cycle.

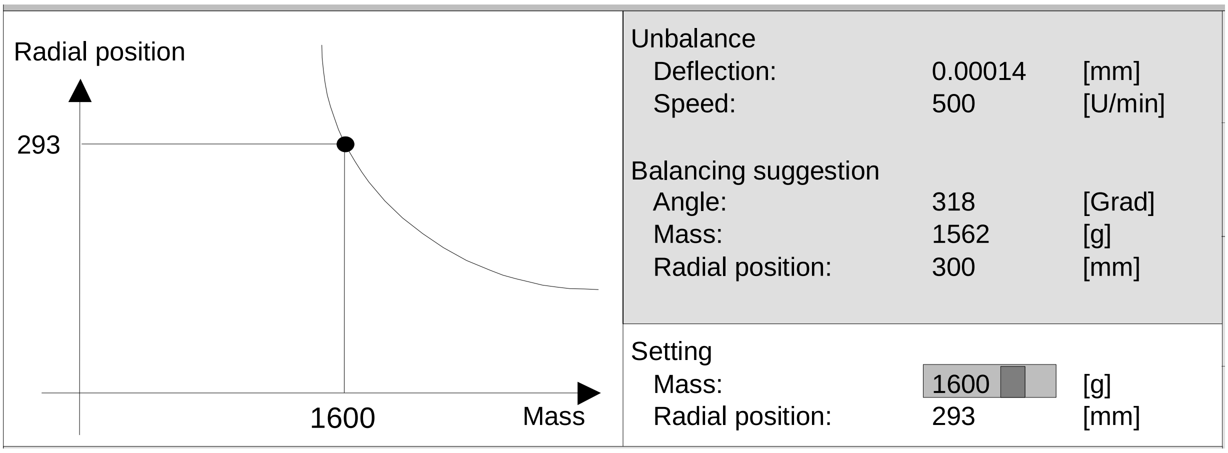
Example: **Balancing a workpiece**

G691 D500
Explanation:

1 Start balancing cycle with maximum speed of 500 rpm.
2 Unbalance is measured. Calculated mass and radial position (distance and angle) are shown in the window. The balance position is automatically positioned.
3 Enter the weight of an available mass in the window.
4 The CNC displays the new radial distance for the available mass.
5 Fit the mass at the radial position (distance and angle). Terminate with start.
6 Check the balance quality by repeating the balancing cycle G691. The unbalance mass must be very small. If necessary, balance again with the displayed mass.

Representation of measurement results

Once the unbalance detection measurement is terminated, the measurement results are displayed instead of the input and support fields. This image is created by G350.



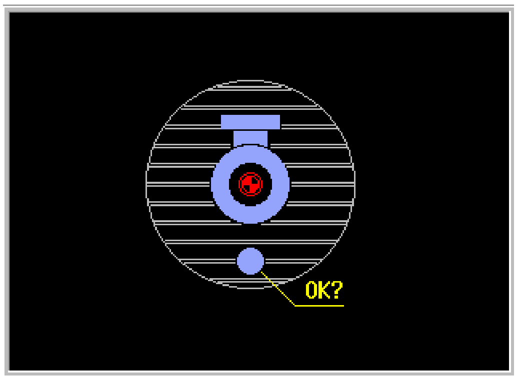
Left:
The relationship between mass and position are shown graphically.
Top right:
The measured unbalance causes a deflection at the speed displayed. This unbalance can be compensated in accordance with the balancing suggestion.

Bottom right:
The radial position for a selected mass is calculated in the dialog window. The calculation takes place after pressing the <ENTER> key. The START key terminates the cycle and closes this window.

In **automatic mode**, the left graphical window is not shown so that the program pointer remains visible.

7.9.4 G692 Unbalance checking

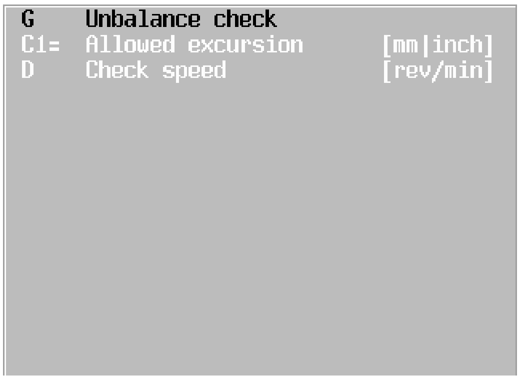
This cycle checks that the unbalance does not exceed a particular value. It should be called at the start of every turning operation to ensure that the concentric error does not exceed the desired tolerance or the specified limit.



Format

N... G692 {C1=..} {D..}

Parameters



C1= Maximum unbalance for message
Basic setting MC451 “maximum deflection”.

D Programmed speed for checking
Basic setting MC2691 “maximum speed”

Type of function

modal

Associated functions

G691

Notes and application

When checking unbalance, the deflection of the linear axis is measured at a specified speed. If the deflection reaches the value C1=, an error message is issued.

Example: Checking unbalance.

G692 C1=0.003 D500

Explanation:
The CNC detects whether the deflection of the table is within the limit of 0.003 mm at a speed of 500 rpm. If the deflection is greater than the value entered (C1=), the program is stopped.

7.9.5 Unbalance example

```
N1 G691 D500
N2 G691 D500
N...  milling
N30 G37
N31 G692 D500
N...  turning
```

Explanation:
N1 G691 D500

1 Start balancing cycle with maximum speed of 500 rpm.
2 Unbalance is measured. Calculated mass and radial position (distance and angle) are shown in the window. The balance position is automatically located.
3 Enter the weight of an available mass in the window.
4 The CNC displays the new radial distance for the available mass.
5 Fit the mass at the radial position (distance and angle). Terminate with start.

N2 G691 D500
Check the balance quality by repeating the balancing cycle G691. The unbalance mass must be very small. If necessary, balance again with the displayed mass.

N...milling
Unbalance may change due to milling processes or changes in the clamping.
N30 G37
Start turning mode
N31 G692 D500
Check whether unbalance is still correct.

7.10 Turning cycles

AVAILABILITY

Machine and CNC must be prepared for turning operations by the machine builder. If your machine is not equipped with all the G functions described here, please refer to your machine manual.

The tuning cycles are executed as macros, every block can be seen in the display and each block is processed as a single block.

General notes and application

STARTING POINT

The starting point determines the place where the tool starts machining. The cutting steps start from this position. If the tool is a long distance away, several cutting steps take place. If the tool is between Y1= and Y2=, cutting will start there and the cutting may not all be carried out.
If the co-ordinate of the starting point Y is smaller than the co-ordinate of the machining starting point Y1, the machine first travels to co-ordinate Z1.

TOOL MEMORY ADDRESSES

The following addresses are used in the tool memory:

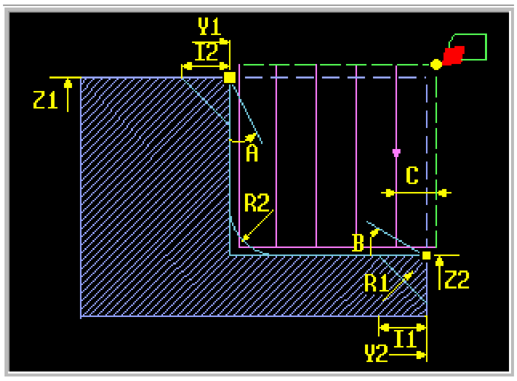
C	Tool tip radius
O	Tool orientation

If no O is entered in the tool memory, a standard orientation is assumed depending on the machining.

RADIUS COMPENSATION

Tool tip radius compensation is carried out automatically in this G function.

7.10.1 G822 Longitudinal cut



Format

N... G822 Y.. Z.. Y1=.. Z1=.. Y2=.. Z2=.. C.. {A..} {B..} [{I1=..}|{R1=..}] [{I2=..}|{R2=..}] {S1=..} {F}

Parameters

G	Clearance axial (E)
Y	Startpoint
Z	Startpoint
Y1=	Beginpoint contour
Z1=	Beginpoint contour
Y2=	Endpoint contour
Z2=	Endpoint contour
C	Cutting depth
A	(Angle 1)
B	(Angle 2)
I1=	(Chamfer length 1)
R1=	(Radius 1)
I2=	(Chamfer length 2)
R2=	(Radius 2)

S1=	((Cutting) Speed)
F	(Feed)

Y Starting point. (Recommended value: safety distance up to Y2=)
Position of tool in radial direction. This position is the starting point for machining. Y is reduced with C until Y1= is reached.

Z Starting point. (Recommended value: safety distance up to Z1=)
Position of tool in axial direction. This position is the starting point for machining. Machining starts at Z until Z2 is reached.

Y1= Contour starting point
Starting point of the contour to be machined.

Z1= Contour starting point
Starting point of the contour to be machined.

Y2= Contour end point
End point of the contour to be machined.

Z2= Contour end point
End point of the contour to be machined.

C Radial feed depth (incremental):
Dimension by which the tool is fed in the radial direction in each case. The depth does not have to be a multiple of the feed depth.

A Angle: Basic setting A=0.
Angle (>0) at contour starting point. Angle A or B must be chosen so that the tool does not undercut.

B Angle: Basic setting B=0.
Angle (>0) at contour end point.

I1= Chamfer length: Basic setting I1=0.

Chamfer length at contour end point.
Only I1= or R1= may be programmed.

R1= Rounding: Basic setting R1=0.
Rounding at contour end point.

I2= Chamfer length: Basic setting I2=0.
Chamfer length at contour starting point.

R2= Rounding: Basic setting R2= tool tip radius.
Rounding between angles A and B.

F Feed rate: Traverse speed of tool in mm/min

S1 Spindle speed

Type of function

Non-modal

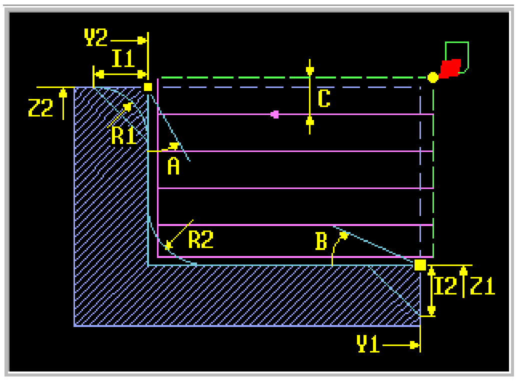
Associated functions

G827 for finish machining

Notes and application

Cutting takes place first, then finish machining.
Tool orientation may only be 4, 5 or 6.
The tool path is corrected for the tip radius.

7.10.2 G823 Cutting plan



Format

N... G823 Y.. Z.. Y1=.. Z1=.. Y2=.. Z2=.. C.. {A..} {B..} [{I1=..}|{R1=..}] [{I2=..}|{R2=..}] {S1=..} {F}

Parameters

G	Clearance radial (E)
Y	Startpoint
Z	Startpoint
Y1=	Beginpoint contour
Z1=	Beginpoint contour
Y2=	Endpoint contour
Z2=	Endpoint contour
C	Cutting depth
A	(Angle 1)
B	(Angle 2)
I1=	(Chamfer length 1)
R1=	(Radius 1)
I2=	(Chamfer length 2)
R2=	(Radius 2)

S1=	((Cutting) Speed)
F	(Feed)

Y Starting point. (Recommended value: safety distance up to Y1=)
Position of tool in radial direction. This position is the starting point for machining. Machining starts at Y until Y2 is reached.

Z Starting point. (Recommended value: safety distance up to Z2=)
Position of tool in axial direction. This position is the starting point for machining. Z is reduced with C until Z1= is reached.

Y1= Contour starting point
Starting point of the contour to be machined.

Z1= Contour starting point
Starting point of the contour to be machined.

Y2= Contour end point
End point of the contour to be machined.

Z2= Contour end point
End point of the contour to be machined.

C Radial feed depth (incremental):
Dimension by which the tool is fed in the axial direction in each case. The depth does not have to be a multiple of the feed depth.

A Angle: Basic setting A=0.
Angle (>0) at contour starting point.
Angle A or B must be chosen so that the tool does not undercut.

B Angle: Basic setting B=0.
Angle (>0) at contour end point.

I1= Chamfer length: Basic setting I1=0.
Chamfer length at contour end point.
Only I1= or R1= may be programmed.
R1= Rounding: Basic setting R1=0.
Rounding at contour end point.
I2= Chamfer length: Basic setting I2=0.
Chamfer length at contour starting point.
R2= Rounding: Basic setting R2= tool tip radius.
Rounding between angles A and B.
F Feed rate: Traverse speed of tool in mm/min
S1 Spindle speed

Type of function

Non-modal

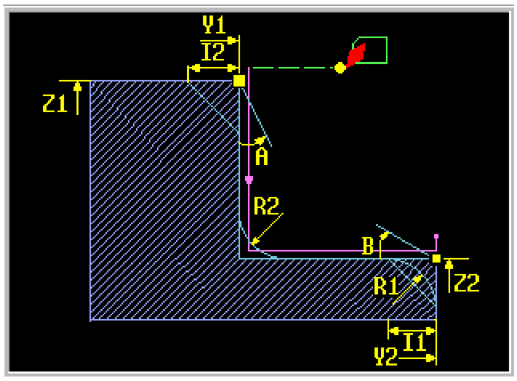
Associated functions

G827 for finish machining

Notes and application

Cutting takes place first, then finish machining.
Tool orientation may only be 4, 5 or 6.
The tool path is corrected for the tip radius.

7.10.3 G826 Finish machining longitudinal cut



Format

N... G826 Y.. Z.. Y1=.. Z1=.. Y2=.. Z2=.. {A..} {B..} [{I1=..}|{R1=..}] [{I2=..}|{R2=..}] {S1=..} {F}

Parameters

```
G    Clearance axial finishing (E)
Y    Startpoint
Z    Startpoint
Y1=  Beginpoint contour
Z1=  Beginpoint contour
Y2=  Endpoint contour
Z2=  Endpoint contour
A    (Angle 1)
B    (Angle 2)
I1=  (Chamfer length 1)
R1=  (Radius 1)
I2=  (Chamfer length 2)
R2=  (Radius 2)
S1=  ((Cutting) Speed)
```

```
F    (Feed)
```

Y Starting point. (Recommended value: safety distance up to Y1=)
Position of tool in radial direction. This position is the starting point for finish machining.

Z Starting point. (Recommended value: safety distance up to Z1=)
Position of tool in axial direction. This position is the starting point for finish machining. Finish machining starts at Y.

Y1= Contour starting point
Starting point of the contour to be machined.

Z1= Contour starting point
Starting point of the contour to be machined.

Y2= Contour end point
End point of the contour to be machined.

Z2= Contour end point
End point of the contour to be machined.

A Angle: Basic setting A=0.
Angle (>0) at contour starting point.
Angle A or B must be chosen so that the tool does not undercut.

B Angle: Basic setting B=0.
Angle (>0) at contour end point.

I1= Chamfer length: Basic setting I1=0.
Chamfer length at contour end point.
Only I1= or R1= may be programmed.

R1= Rounding: Basic setting R1=0.
Rounding at contour end point.
I2= Chamfer length: Basic setting I2=0.
Chamfer length at contour starting point.
R2= Rounding: Basic setting R2= tool tip radius.
Rounding between angles A and B.
F Feed rate: Traverse speed of tool in mm/min
S1 Spindle speed

Type of function

Non-modal

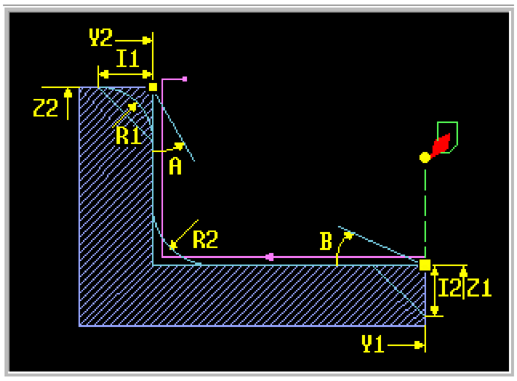
Associated functions

G822 for rough machining

Notes and application

Finish machining goes from Y1/Z1 to Y2/Z2.
Tool orientation may only be 4, 5 or 6.
The tool path is corrected for the tip radius.

7.10.4 G827 Cutting plan, finish machining



Format

N... G827 Y.. Z.. Y1=.. Z1=.. Y2=.. Z2=.. {A..} {B..} [{I1=..}|{R1=..}]]{I2=..}|{R2=..}] {S1=..} {F}

Parameters

G	Clearance radial finishing (E)
Y	Startpoint
Z	Startpoint
Y1=	Beginpoint contour
Z1=	Beginpoint contour
Y2=	Endpoint contour
Z2=	Endpoint contour
A	(Angle 1)
B	(Angle 2)
I1=	(Chamfer length 1)
R1=	(Radius 1)
I2=	(Chamfer length 2)
R2=	(Radius 2)
S1=	((Cutting) Speed)

F	(Feed)

Y Starting point. (Recommended value: safety distance up to Y1=)
Position of tool in radial direction. This position is the starting point for finish machining. Finish machining starts at Y until Y2 is reached.

Z Starting point. (Recommended value: safety distance up to Z2=)
Position of tool in axial direction. This position is the starting point for finish machining.

Y1= Contour starting point
Starting point of the contour to be machined.

Z1= Contour starting point
Starting point of the contour to be machined.

Y2= Contour end point
End point of the contour to be machined.

Z2= Contour end point
End point of the contour to be machined.

A Angle: Basic setting A=0.
Angle (>0) at contour starting point.
Angle A or B must be chosen so that the tool does not undercut.

B Angle: Basic setting B=0.
Angle (>0) at contour end point.

I1= Chamfer length: Basic setting I1=0.
Chamfer length at contour end point.
Only I1= or R1= may be programmed.

R1= Rounding: Basic setting R1=0.
Rounding at contour end point.
I2= Chamfer length: Basic setting I2=0.
Chamfer length at contour starting point.
R2= Rounding: Basic setting R2= tool tip radius.
Rounding between angles A and B.
F Feed rate: Traverse speed of tool in mm/min
S1 Spindle speed

Type of function

Non-modal

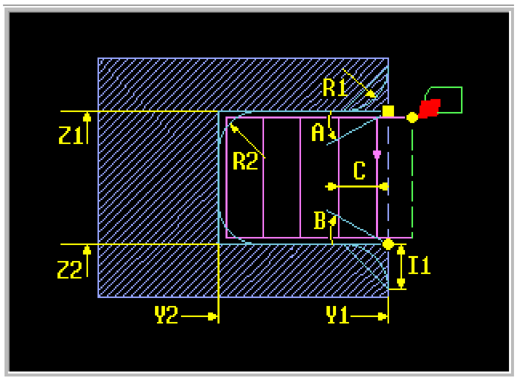
Associated functions

G823 for rough machining

Notes and application

Finish machining goes from Y1/Z1 to Y2/Z2.
Tool orientation may only be 4, 5 or 6.
The tool path is corrected for the tip radius.

7.10.5 G832 Longitudinal reverse boring



Format

N... G832 Y.. Z.. Y1=.. Z1=.. Y2=.. Z2=.. C.. {A..} {B..} [{I1=..}|{R1=..}] {R2=..} {S1=..} {F}

Parameters

G	Roughing axial (E)
Y	Startpoint
Z	Startpoint
Y1=	Beginpoint contour
Z1=	Beginpoint contour
Y2=	Endpoint contour
Z2=	Endpoint contour
C	Cutting depth
A	(Angle 1)
B	(Angle 2)
I1=	(Chamfer length 1)
R1=	(Radius 1)
R2=	(Radius 2)
S1=	((Cutting) Speed)

F	(Feed)

Y Starting point. (Recommended value: safety distance up to Y1=)
Position of tool in radial direction. This position is the starting point for machining. Machining starts at Y and is reduced with C until Y2= is reached.

Z Starting point. (Recommended value: safety distance up to Z1=)
Position of tool in axial direction. This position is the starting point for machining. Machining starts at Z1= until Z2= is reached.

Y1= Contour starting point
Starting point of the contour to be machined.

Z1= Contour starting point
Starting point of the contour to be machined.

Y2= Contour end point
End point of the contour to be machined.

Z2= Contour end point
End point of the contour to be machined.

C Radial feed depth (incremental):
Dimension by which the tool is fed in the radial direction in each case. The depth does not have to be a multiple of the feed depth.

A Angle: Basic setting A=0.
Angle (>0) at contour starting point. (Z1=)
Angles A and B must be chosen so that the tool does not undercut.

B Angle: Basic setting B=0.
Angle (>0) at contour end point. (Z2=)

I1= Chamfer length: Basic setting I1=0.
Chamfer length at start and end of contour.
Only I1= or R1= may be programmed.
R1= Rounding: Basic setting R1=0.
Rounding at start and end of contour.
R2= Rounding: Basic setting R2= tool tip radius.
Rounding at the bottom of the contour.
F Feed rate: Traverse speed of tool in mm/min
S1 Spindle speed

Type of function

Non-modal

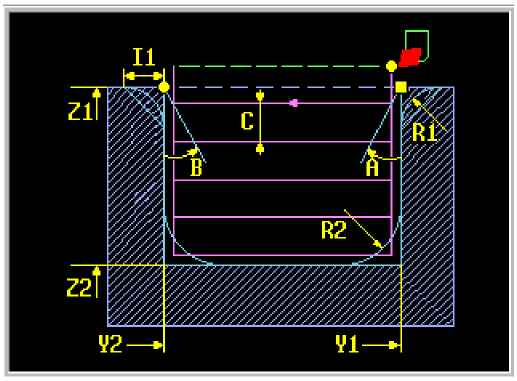
Associated functions

G837 for finish machining

Notes and application

Rough cutting takes place first, then finish machining.
Tool orientation may only be 3, 4 or 5.
The tool path is corrected for the tip radius.

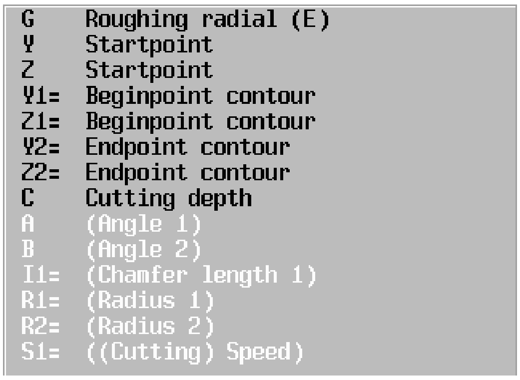
7.10.6 G833 Reverse boring plan



Format

N... G833 Y.. Z.. Y1=.. Z1=.. Y2=.. Z2=.. C.. {A..} {B..} [{I1=..}|{R1=..}] {R2=..} {S1=..} {F}

Parameters



Y Starting point. (Recommended value: safety distance up to Y1=)
Position of tool in radial direction. This position is the starting point for machining. Machining starts at Y1= until Y2= is reached.

Z Starting point. (Recommended value: safety distance up to Z1=)
Position of tool in radial direction. This position is the starting point for machining. Machining starts at Z and is reduced with C until Z2= is reached.

Y1= Contour starting point
Starting point of the contour to be machined.

Z1= Contour starting point
Starting point of the contour to be machined.

Y2= Contour end point
End point of the contour to be machined.

Z2= Contour end point
End point of the contour to be machined.

C Radial feed depth (incremental):
Dimension by which the tool is fed in the axial direction in each case. The depth does not have to be a multiple of the feed depth.

A Angle: Basic setting A=0.
Angle (>0) at contour starting point. (Y1=)
Angles A and B must be chosen so that the tool does not undercut.

B Angle: Basic setting B=0.
Angle (>0) at contour end point. (Y2=)

I1= Chamfer length: Basic setting I1=0.

Chamfer length at start and end of contour.
Only I1= or R1= may be programmed.

R1= Rounding: Basic setting R1=0.
Rounding at start and end of contour.

R2= Rounding: Basic setting R2= tool tip radius.
Rounding at the bottom of the contour.

F Feed rate: Traverse speed of tool in mm/min

S1 Spindle speed

Type of function

Non-modal

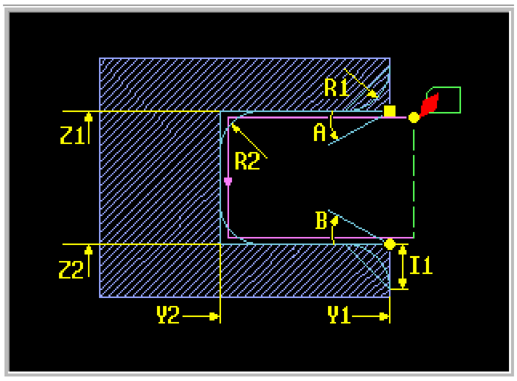
Associated functions

G837 for finish machining

Notes and application

Rough cutting takes place first, then finish machining.
Tool orientation may only be 5, 6 or 7.
The tool path is corrected for the tip radius.

7.10.7 G836 Finish machining longitudinal reverse boring



Format

N... G836 Y.. Z.. Y1=.. Z1=.. Y2=.. Z2=.. {A..} {B..} [{I1=..}|{R1=..}] {R2=..} {S1=..} {F}

Parameters

G	Roughing axial finishing (E)
Y	Startpoint
Z	Startpoint
Y1=	Beginpoint contour
Z1=	Beginpoint contour
Y2=	Endpoint contour
Z2=	Endpoint contour
A	(Angle 1)
B	(Angle 2)
I1=	(Chamfer length 1)
R1=	(Radius 1)
R2=	(Radius 2)
S1=	((Cutting) Speed)
F	(Feed)

Y Starting point. (Recommended value: safety distance up to Y1=)
Position of tool in radial direction. This position is the starting point for finish machining.

Z Starting point. (Recommended value: safety distance up to Z1=)
Position of tool in axial direction. This position is the starting point for finish machining. Finish machining starts at Z1= until Z2= is reached.

Y1= Contour starting point
Starting point of the contour to be machined.

Z1= Contour starting point
Starting point of the contour to be machined.

Y2= Contour end point
End point of the contour to be machined.

Z2= Contour end point
End point of the contour to be machined.

A Angle: Basic setting A=0.
Angle (>0) at contour starting point. (Z1=)
Angles A and B must be chosen so that the tool does not undercut.

B Angle: Basic setting B=0.
Angle (>0) at contour end point. (Z2=)

I1= Chamfer length: Basic setting I1=0.
Chamfer length at start and end of contour.
Only I1= or R1= may be programmed.

R1= Rounding: Basic setting R1=0.
Rounding at start and end of contour.
R2= Rounding: Basic setting R2= tool tip radius.
Rounding at the bottom of the contour.
F Feed rate: Traverse speed of tool in mm/min
S1 Spindle speed

Type of function

Non-modal

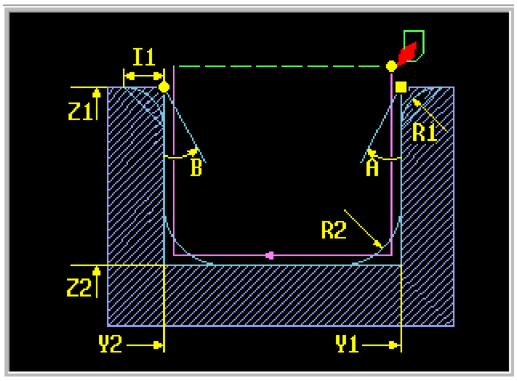
Associated functions

G832 for finish machining

Notes and application

Finish machining goes from Y1/Z1 to Y1/Z2.
Tool orientation may only be 3, 4 or 5.
The tool path is corrected for the tip radius.

7.10.8 G837 Reverse boring plan, finish machining



Format

N... G837 Y.. Z.. Y1=.. Z1=.. Y2=.. Z2=.. {A..} {B..} [{I1=..}|{R1=..}] {R2=..} {S1=..} {F}

Parameters

G	Roughing radial finishing (E)
Y	Startpoint
Z	Startpoint
Y1=	Beginpoint contour
Z1=	Beginpoint contour
Y2=	Endpoint contour
Z2=	Endpoint contour
A	(Angle 1)
B	(Angle 2)
I1=	(Chamfer length 1)
R1=	(Radius 1)
R2=	(Radius 2)
S1=	((Cutting) Speed)
F	(Feed)

Y Starting point. (Recommended value: safety distance up to Y1=)
Position of tool in radial direction. This position is the starting point for finish machining. Finish machining starts at Y1= until Y2= is reached.

Z Starting point. (Recommended value: safety distance up to Z1=)
Position of tool in radial direction. This position is the starting point for finish machining.

Y1= Contour starting point
Starting point of the contour to be machined.

Z1= Contour starting point
Starting point of the contour to be machined.

Y2= Contour end point
End point of the contour to be machined.

Z2= Contour end point
End point of the contour to be machined.

A Angle: Basic setting A=0.
Angle (>0) at contour starting point. (Y1=)
Angles A and B must be chosen so that the tool does not undercut.

B Angle: Basic setting B=0.
Angle (>0) at contour end point. (Y2=)

I1= Chamfer length: Basic setting I1=0.
Chamfer length at start and end of contour.
Only I1= or R1= may be programmed.

R1= Rounding: Basic setting R1=0.
Rounding at start and end of contour.
R2= Rounding: Basic setting R2= tool tip radius.
Rounding at the bottom of the contour.
F Feed rate: Traverse speed of tool in mm/min
S1 Spindle speed

Type of function

Non-modal

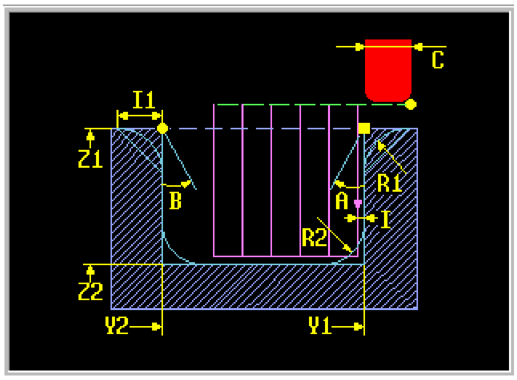
Associated functions

G833 for finish machining

Notes and application

Finish machining goes from Y1/Z1 to Y1/Z2.
Tool orientation may only be 5, 6 or 7.
The tool path is corrected for the tip radius.

7.10.9 G842 Axial plunge cutting



Format

N... G842 Y.. Z.. Y1=.. Z1=.. Y2=.. Z2=.. C.. {A..} {B..} [{I1=..}|{R1=..}] {R2=..} {I..} {S1=..} {F}

Parameters

G	Grooving axial (E)
Y	Startpoint
Z	Startpoint
Y1=	Beginpoint contour
Z1=	Beginpoint contour
Y2=	Endpoint contour
Z2=	Endpoint contour
C	Tool width
A	(Angle 1)
B	(Angle 2)
I1=	(Chamfer length 1)
R1=	(Radius 1)
R2=	(Radius 2)
I	(Finishing)

S1=	((Cutting) Speed)
F	(Feed)

Y Starting point. (Recommended value: safety distance up to Y1=)
Position of tool in radial direction. This position is the starting point for machining. Machining starts at Y1= with the feed width until Y2= is reached.

Z Starting point. (Recommended value: safety distance up to Z2=)
Position of tool in axial direction. This position is the starting point for machining.

Y1= Contour starting point
Starting point of the contour to be machined.

Z1= Contour starting point
Starting point of the contour to be machined.

Y2= Contour end point
End point of the contour to be machined.

Z2= Contour end point
End point of the contour to be machined.

C Chisel width
Width of tool. The feed width is C minus twice the tip radius

A Angle: Basic setting A=0.
Angle (>0) at contour starting point. (Y1=)

B Angle: Basic setting B=0.
Angle (>0) at contour end point. (Y2=)

I1= Chamfer length: Basic setting I1=0.
Chamfer length at start and end of contour.
Only I1= or R1= may be programmed.

R1= Rounding: Basic setting R1=0.

Rounding at start and end of contour.

R2= Rounding: Basic setting R2= tool corner radius.
Rounding at the bottom of the contour.

I Finish machining allowance: basic setting I=0.

F Feed rate: Traverse speed of tool in mm/min

S1 Spindle speed

Type of function

Non-modal

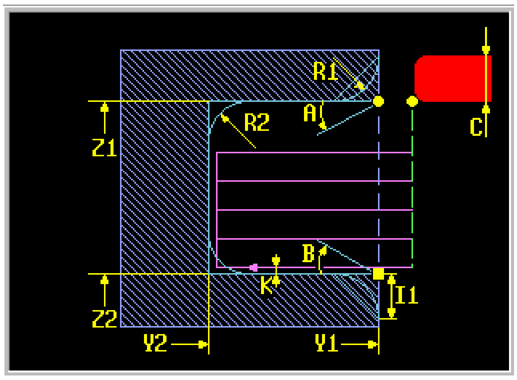
Associated functions

G846 for finish machining

Notes and application

Rough cutting takes place first, then finish machining.
Tool orientation may only be 5, 6 or 7.
The tool path is corrected for the tip radius.

7.10.10 G843 Radial plunge cutting



Format

N... G843 Y.. Z.. Y1=.. Z1=.. Y2=.. Z2=.. C.. {A..} {B..} [{I1=..}|{R1=..}] {R2=..} {K..} {S1=..} {F}

Parameters

G	Grooving radial (E)
Y	Startpoint
Z	Startpoint
Y1=	Beginpoint contour
Z1=	Beginpoint contour
Y2=	Endpoint contour
Z2=	Endpoint contour
C	Tool width
A	(Angle 1)
B	(Angle 2)
I1=	(Chamfer length 1)
R1=	(Radius 1)
R2=	(Radius 2)
K	(Finishing)

S1=	((Cutting) Speed)
F	(Feed)

Y Starting point. (Recommended value: safety distance up to Y1=)
Position of tool in radial direction. This position is the starting point for machining. Machining starts at Y until Y2 is reached.

Z Starting point. (Recommended value: safety distance up to Z2=)
Position of tool in axial direction. This position is the starting point for machining. Machining starts at Z2= with the feed width until Z1= is reached.

Y1= Contour starting point
Starting point of the contour to be machined.

Z1= Contour starting point
Starting point of the contour to be machined.

Y2= Contour end point
End point of the contour to be machined.

Z2= Contour end point
End point of the contour to be machined.

C Chisel width
Width of tool. The feed width is C minus twice the tip radius

A Angle: Basic setting A=0.
Angle (>0) at contour starting point. (Z1=)

B Angle: Basic setting B=0.
Angle (>0) at contour end point. (Z2=)

I1= Chamfer length: Basic setting I1=0.
Chamfer length at start and end of contour.
Only I1= or R1= may be programmed.

R1= Rounding: Basic setting R1=0.
Rounding at start and end of contour.
R2= Rounding: Basic setting R2= tool tip radius.
Rounding at the bottom of the contour.
K Finish machining allowance: basic setting K=0.
F Feed rate: Traverse speed of tool in mm/min
S1 Spindle speed

Type of function

Non-modal

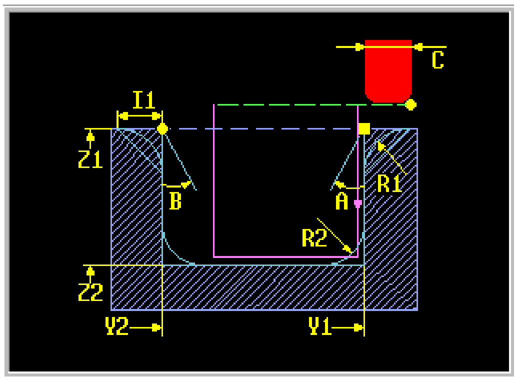
Associated functions

G847 for finish machining

Notes and application

Rough cutting takes place first, then finish machining.
Tool orientation may only be 3, 4 or 5.
The tool path is corrected for the tip radius.

7.10.11 G846 Axial plunge cutting, finish machining



Format

N... G846 Y.. Z.. Y1=.. Z1=.. Y2=.. Z2=.. C.. {A..} {B..} [{I1=..}|{R1=..}] {R2=..} {I..} {S1=..} {F}

Parameters

G	Grooving axial finishing (E)
Y	Startpoint
Z	Startpoint
Y1=	Beginpoint contour
Z1=	Beginpoint contour
Y2=	Endpoint contour
Z2=	Endpoint contour
C	Tool width
A	(Angle 1)
B	(Angle 2)
I1=	(Chamfer length 1)
R1=	(Radius 1)
R2=	(Radius 2)
I	(Finishing)

S1=	((Cutting) Speed)
F	(Feed)

Y Starting point. (Recommended value: safety distance up to Y1=)
: Position of tool in radial direction. This position is the starting point for machining. Machining starts at Y until Y2 is reached.

Z Starting point. (Recommended value: safety distance up to Z1=)
: Position of tool in axial direction. This position is the starting point for machining. Starts at Z2= until Z1= is reached.

Y1= Contour starting point
: Starting point of the contour to be machined.

Z1= Contour starting point
: Starting point of the contour to be machined.

Y2= Contour end point
: End point of the contour to be machined.

Z2= Contour end point
: End point of the contour to be machined.

C Chisel width
: Width of tool. The feed width is C minus twice the corner radius

A Angle: Basic setting A=0.
: Angle (>0) at contour starting point. (Y1=)

B Angle: Basic setting B=0.
: Angle (>0) at contour end point. (Y2=)

I1= Chamfer length: Basic setting I1=0.
: Chamfer length at start and end of contour.
: Only I1= or R1= may be programmed.

R1= Rounding: Basic setting R1=0.
Rounding at start and end of contour.
R2= Rounding: Basic setting R2= tool tip radius.
Rounding at the bottom of the contour.
I Finish machining allowance: basic setting I=0.
F Feed rate: Traverse speed of tool in mm/min
S1 Spindle speed

Type of function

Non-modal

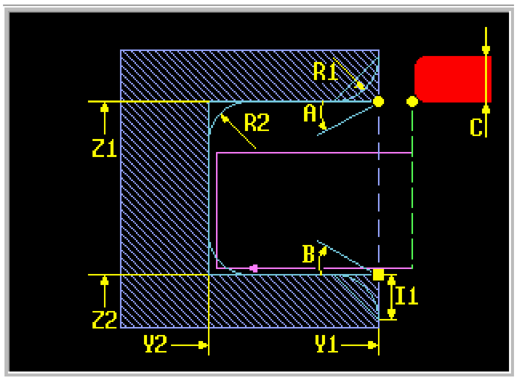
Associated functions

G842 for finish machining

Notes and application

Finish machining goes from Y1/Z1 to Y1/Z2.
Tool orientation may only be 5, 6 or 7.
The tool path is corrected for the tip radius.

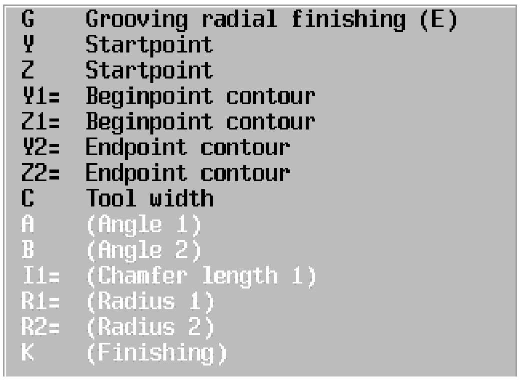
7.10.12 G847 Radial plunge cutting, finish machining



Format

N... G847 Y.. Z.. Y1=.. Z1=.. Y2=.. Z2=.. C.. {A..} {B..} [{I1=..}|{R1=..}] {R2=..} {K..} {S1=..} {F}

Parameters



Y Starting point. (Recommended value: safety distance up to Y1=)
Position of tool in radial direction. This position is the starting point for finish machining. Finish machining starts at Y until Y2 is reached.

Z Starting point. (Recommended value: safety distance up to Z2=)
Position of tool in axial direction. This position is the starting point for finish machining.

Y1= Contour starting point
Starting point of the contour to be machined.

Z1= Contour starting point
Starting point of the contour to be machined.

Y2= Contour end point
End point of the contour to be machined.

Z2= Contour end point
End point of the contour to be machined.

C Chisel width
Width of tool. The feed width is C minus twice the corner radius

A Angle: Basic setting A=0.
Angle (>0) at contour starting point. (Z1=)

B Angle: Basic setting B=0.
Angle (>0) at contour end point. (Z2=)

I1= Chamfer length: Basic setting I1=0.
Chamfer length at start and end of contour.
Only I1= or R1= may be programmed.

R1= Rounding: Basic setting R1=0.
Rounding at start and end of contour.
R2= Rounding: Basic setting R2= tool tip radius.
Rounding at the bottom of the contour.
K Finish machining allowance: basic setting K=0.
F Feed rate: Traverse speed of tool in mm/min
S1 Spindle speed

Type of function

Non-modal

Associated functions

G843 for rough machining

Notes and application

Finish machining goes from Y1/Z2 to Y1/Z1.
Tool orientation may only be 3, 4 or 5.
The tool path is corrected for the tip radius.

7.11 Examples

Example 1:

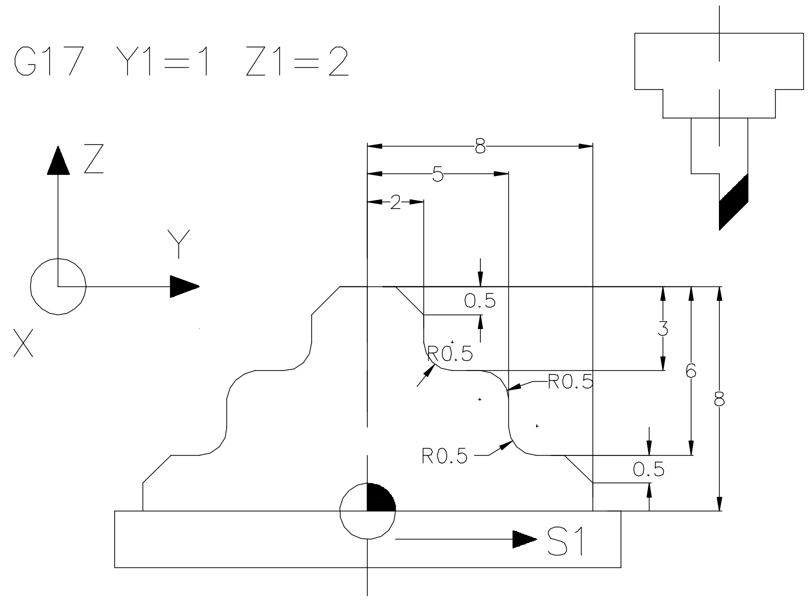
```
N9999
N1 G17
N2 G37
N3 M54
N4 T1 M6
N5 S1000 F1000 M3
N...

N100 G17 Z1=1 Y1=2
N101 G36
N102 T7 M6
N103 S1=100 M1=3
N104 G0 X0 Y100 Z100
N105 G822 ....
N...

N200 G17
N201 G37
N203 T1 M6
N204 S1000 M3
N205 ....
N300 M30
```

Explanation:

N1	Set planes for milling. Length compensation in Z direction.
N2	Milling mode
N3	Head is in the Z direction
N4	Insert milling tool
N5	Start Spindle
N...	
N...	Milling
N100	Set planes for turning. Main axis 1 is Z, main axis 2 is Y. Radius correction in ZY plane.
N101	Turning
N102	Insert tool
N103	Start rotary table for continuous turning
N104	Position turning tool
N105	Start longitudinal cutting cycles
N...	
N...	Turning
N200	Set planes for milling. Length compensation in Z direction.
N201	Milling mode
N202	Insert milling tool
N103	Start Spindle
N104	Position milling tool
N...	
N...	Milling
N300	Program end

Example 2:

```
N9999
N1 G17
N2 G37
N3 G54 I1 Z8
N4 G36 (continuous turning)
N5 M54
N6 G17 Z1=1 Y1=2
N7 G195 X-1 Y-1 Z1 I2 J12 K-11
N8 G199 X0 Y0 Z0 B4 C2
N9 G198 I1=14 X0 Y8 Z0
N10 G2 X0 Y8 I0 J0
N11 G1 X0 Y8 Z-8
N12 G2 X0 Y8 I0 J0
N13 G196
N14 T1 M6 (L100 R5 C0.3 Q3=800)
N15 S1=1000 M1=3
N16 G0 X0 Y8 Z3 F1000 S1000 M5
N17 (rough machining)
N18 G823 Y8 Z0.3 Y1=8 Z1=-3 Y2=2 Z2=0 I1=0.5 R2=0.5 C0.2
N19 G823 Y8 Z-2.7 Y1=8 Z1=-6 Y2=5 Z2=-3 R1=0.5 I2=0.5 R2=0.5 C0.2
N20 (fine machining)
N21 G827 Y8 Z-6.7 Y1=8 Z1=-6 Y2=5 Z2=-3 R1=0.5 I2=0.5 R2=0.5
N22 G827 Y8 Z-2.7 Y1=8 Z1=-3 Y2=2 Z2=0 I1=0.5 R2=0.5
N23 G0 Z10
N24 T0 M6
N25 G37
N26 G53
N27 M30
```



Explanation:

N1 Set planes for milling. Length compensation in Z direction.
N2 Milling mode
N3 Zero point displacement for Z direction. Upper edge of material is zero.
N4 Turning
N5 Head is in the Z direction

N6 Set planes for turning.
Main axis 1 is Z, main axis 2 is Y.
Radius correction in ZY plane.
N7 Set graphics window
N8 Start of material graphical contour description. B4 means automatic drawing.
N9 Start of contour description. I1=14 is light blue colour
N10 Upper circle of cylinder.
N11 Line
N12 Lower circle of cylinder.
N13 End of graphical contour description
N14 Insert turning tool (length, radius, corner radius and type)
N15 Start rotary table for continuous turning
N16 Position turning tool
N18 G823 start cutting plan cycles. Turn upper part

Y8	Starting point
Z0.3	Starting point
Y1=8	Contour starting point
Z1=-3	Contour starting point
Y2=2	Contour end point
Z2=0	Contour end point
C0.2	Feed depth
I1=0.5	Chamfer length 1
R2=0.5	Radius 2

N19 G823 start cutting plan cycles. Turn lower part

Y8	Starting point
Z-2.7	Starting point
Y1=8	Contour starting point
Z1=-6	Contour starting point
Y2=5	Contour end point
Z2=0	Contour end point
C0.2	Feed depth
R1=0.5	Radius 1
I2=0.5	Chamfer length 2
R2=0.5	Radius 2

N21 G827 start finish machining cutting plan cycles. Finish machine lower part

Y8	Starting point
Z-6.7	Starting point
Y1=8	Contour starting point
Z1=-6	Contour starting point
Y2=5	Contour end point
Z2=-3	Contour end point
R1=0.5	Radius 1
I2=0.5	Chamfer length 1
R2=0.5	Radius 2

N22 G827 start finish machining cutting plan cycles. Finish machine upper part

Y8	Starting point
Z-2.7	Starting point
Y1=8	Contour starting point
Z1=-3	Contour starting point
Y2=2	Contour end point
Z2=0	Contour end point
I1=0.5	Chamfer length 1
R2=0.5	Radius 2

N23 Move tool clear
N24 Reset tool
N25 Milling mode
N26 Deactivate zero point displacement

N27 Program end

Control system manual V410

HEIDENHAIN NUMERIC B.V.
Eindhoven (NL)
Tel: 31.40.250 13 00
Fax: 31.40.250 13 01

11/07/2001

First edition Software Version V410 20000710

Xxx xxx-20 DER6-32.8-2607_000

1. Introduction

These instructions are intended to assist you in operating and programming the controller.

Please read the information in this manual carefully before you start your new machine. It contains important information on machine operation and safety to enable you to use your machine safely and effectively.

The following advice is important for your safety:

This manual is essential for safe use of the machine.
Please ensure that it is in the vicinity of the machine.

The machine should not be operated, even for a short period, by anyone who has not received suitable training, either in the company, at an Institute of Further Education or in one of the Training Centres.

Please read the general safety regulations issued by your professional association.
If they are not on display in the company, contact your appointed safety representative.

Observe the instructions for proper use of the machine.

The controller and the machine are coordinated using machine constants. Some of these constants are accessible to the user. Caution!
The meaning and function of the constants must be fully understood before any changes are made to these constants. If in doubt, please consult our service department.

The controller is fitted with a backup battery that safeguards the memory content for up to three years after the machine is switched off (but only if the battery is serviceable).

The user should always save the programs and specific data (e.g. technology data, machine constants, etc.) to a PC or to diskette. This will avoid the data becoming irrecoverably lost if the system or backup battery becomes defective.

We reserve the right to make changes to the design, equipment and accessories in the interest of further development. No liability will be accepted for any errors in the data, illustrations or descriptions.

1.1 MillPlus software and functions

This manual describes functions available in MillPlus (VME and LE4xx hardware) for the following or higher software versions:
- V410 (VME, LE4xx) Software number 341 482-xx
- V420 (VME, LE4xx) Software number 344 198-xx

The machine builder adapts the versatile capability of MillPlus to the machine in question by means of machine parameters. That is why some functions described in this manual are not available with every version of MillPlus.
MillPlus functions that are not available on every machine include, for example:
- Digitising option
- Tool measurement with laser system or with TT120/TT130
- DNC Plus (DNET) interface
- Ethernet interface (TCP/IP)
- Autostart (warm machine startup program)
- Milling/turning

Please contact the machine builder for individual support for the particular machine being controlled.

1.2 Software version V420

Note

The V420 software only functions in 16 Mbyte DRAM systems.

Operation:

Expanded calculation operations

- The number of possible addresses in Cycle Design is expanded and OPTIONAL and ACTIVE have been added.
- Calculations for E parameters expanded with the floor(E2), ceil(E2), round(E2,n), mod(E2,E3), sign(E2), atan(E2,E3), acos(E2,E3) and asin(E2,E3) functions.

New G functions:

Dynamic TCPM (Tool Centre Point Management) (G141)
Tool withdrawal movement (G174)
M19 with programmable direction (G303)
Query current G data (G320)
Query programmable kinematic elements (G329)
Write programmable kinematic elements (G339)
Writing to a window (G350)
Writing to a file (G351)

Milling/turning

G17	expanded
G36 and G37	turning
G94 and G95	expanded
G96 and G97	constant cutting speed
G302	overwrite tool orientation radius correction
G691	unbalance detection
G692	unbalance checking
G822, G823, G826 and G827	machining
G832, G833, G836 and G837	hollow boring
G842, G843, G846 and G847	plunge cutting

Expanded G functions:

Expansions: tool measurement cycles for laser measuring system

G600 laser system: calibration
G601 laser system: measuring length
G602 laser system: measuring length and radius
G603 laser system: individual cutting tip monitoring
G604 laser system: tool breakage monitoring

G8 selectable tool radius correction
Start of contour description (G198) expanded with I1 address.
Start of graphic contour description (G199) expanded with B=4.
Query active technology **(G139)** expanded with S1 address.
Query tool table **(G321)** expanded with O address.
Query current modal G function **(G324)** expanded.
Writing to the tool table **(G331)** expanded with O address.

1.3 Software-Version V410

Hinweis

Die V410 Software funktioniert allein in 16Mbyte DRAM Systemen.

Bedienung:

Dateien markieren
Easy Operate Taschen und Nuten mit besserer Technologie
Autostart Warmlaufprogramm
Werkzeugkorrektur

Neue G-Funktionen:
-G8 Werkzeugrichtung schwenken
-G327 Abfragen aktive Betiebsart
-G600-G609 Werkzeugmeßzyklen für Lasermessung und TT120/TT130
-Zyklen Design

Geänderte G-Funktionen:
-G0/G1 Berechneter Rundachsenradius
-G28 Programmierbare Konturgenauigkeit
-G84 Gewindebohren bei aktivem G7 in der Hauptebene mit offenem Regelkreis
-G93 Rücksetz Funktion
-G94 Aktivieren Rundachsenberechnung
-G324 Erweitert mit I1=28

1.4 Software-Version V400

Neue G-Funktionen:
-G7 Freie Bearbeitungsebene
-G319 Lese aktuelle F, S oder T
-G321 Werkzeugtabelle abfragen
-G324 Abfragen G-Gruppe
-G325 Abfragen M-Gruppe
-G331 Schreiben nach Werkzeugtabelle
-G332 Schreiben in Maschinenkonstantenspeicher

Geänderte G-Funktionen:
-Erweiterte Werkzeugtabelle
-Vorschub-Override wirksam/nicht wirksam G25/G26 erweitert mit Spindeloverride
-Gewindeschneid Zyklus G84 ist geändert in Synchron Bewegung (Rigid Tapping)
-Der Meßzyklus G145 ist erweitert mit des Meßposition der Spindel (S1=)
-Meßtasterstatus G148 ist erweitert
-Abfrage Werkzeug-Länge und Radius G149 an Aufmaß angepaßt
-Ändern Werkzeug-Länge und Radius G150 an Aufmaß angepaßt

B. Software version V410

Note Software version V410 will only run on 16 Mbyte DRAM systems.

Operation:
Select files
Easy Operate pockets and keyways with better technology
Autostart or warm start program
Tool correction

New G functions:
-G8 Swivel tool direction
-G327 Query active operating mode
-G600-G609 Tool measuring cycles for laser measurement and TT120/TT130
-Cycle design

Modified G functions:
-G0/G1 Calculated radius of rotary axes
-G28 Programmable contour precision

- -G84 Tapping with active G7 at top level with open control loop
- -G93 Reset function
- -G94 Activate rotary axes calculation
- -G324 Extended with I1=28

C. Software version V400

New functions:

- -G7 Free operating planes
- -G319 Read current F, S or T
- -G321 Query the tool table
- -G324 Query the G group
- -G325 Query the M group
- -G331 Write to the tool table
- -G332 Write to the machine settings memory

Changed functions:

- -Extended tool table
- -Advance override active/inactive G25/G26 enhanced with spindle override
- -Tapping cycle G84 has been changed to synchronous motion (rigid tapping)
- -Measurement cycle G145 has been enhanced by the spindle measuring position (S1=)
- -Measuring key status G148 has been enhanced
- -Query tool length and radius G149 adapted for oversize
- -Change tool length and radius G150 adapted for oversize

2. Safety

Meaning of symbols and notices:

Signifies immediate danger to persons

"LIVE COMPONENTS" Access through authorized personnel only! Indicates
danger due to live components, which must be isolated prior to commencing
repairs.

Applies to operating or plant procedures which have to be followed precisely to avoid danger or injury to persons and damage to the installation.

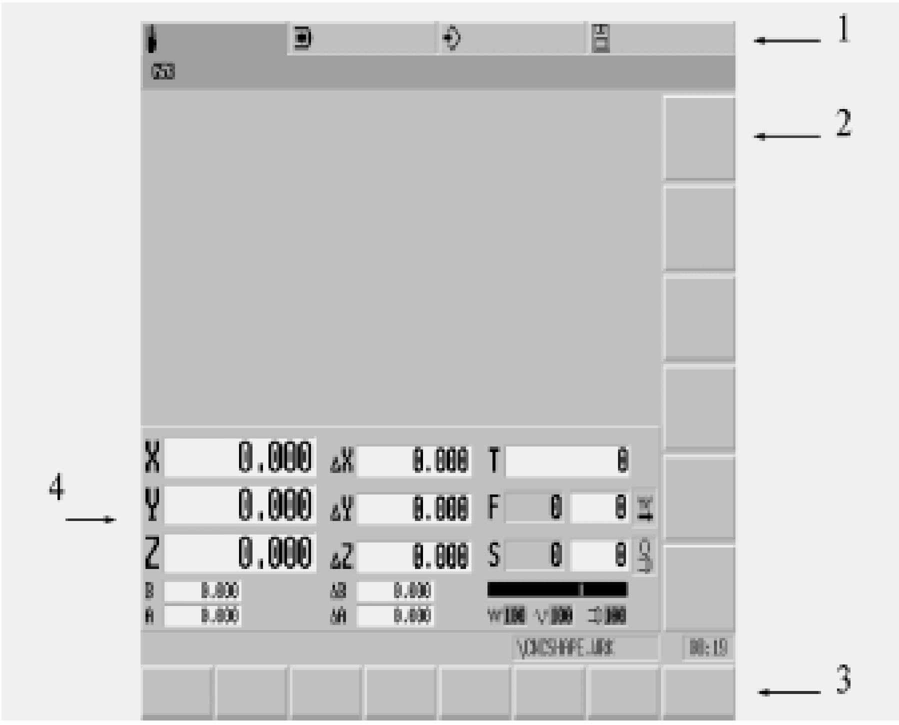
Applies to situations which may pose a danger to persons.

For special technical features which the user must note.

General safety and accident prevention regulations must be heeded as well as the advice given in the operating instructions.

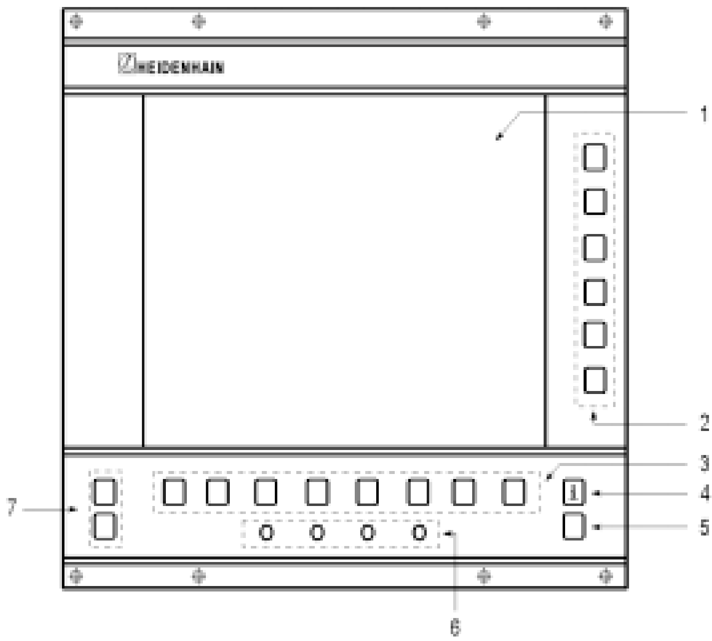
3. Keyboard and screen layout

3.1 Screen display



1 Process level
2 Machine function softkeys
3 Softkeys
4 Machine information

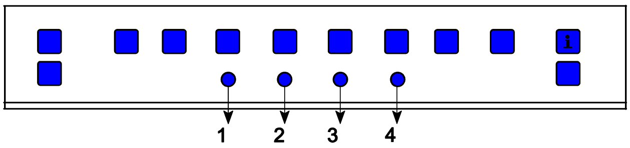
3.2 Screen and keyboard of LE412



3 Softkeys
4 Information key
5 Key without function
6 VGA monitor controls
7..Keys without function

3.2.1 Screen adjusting keys

The screen adjusting keys have different functions depending on the selected mode.



No function selected yet:

1 Demagnetising
2-4 Accessing set-up menu

Set-up menu on screen:

1 Exiting set-up menu (new settings are saved)
2 Moving highlighting of submenu downward (when bottom line of menu 1 is reached and the key is pressed again, menu 2 is selected automatically)
3 Moving highlighting of submenu upward (when the top line of menu 2 is reached and the key is pressed again, menu 1 is selected automatically)
4 Activating the highlighted submenu

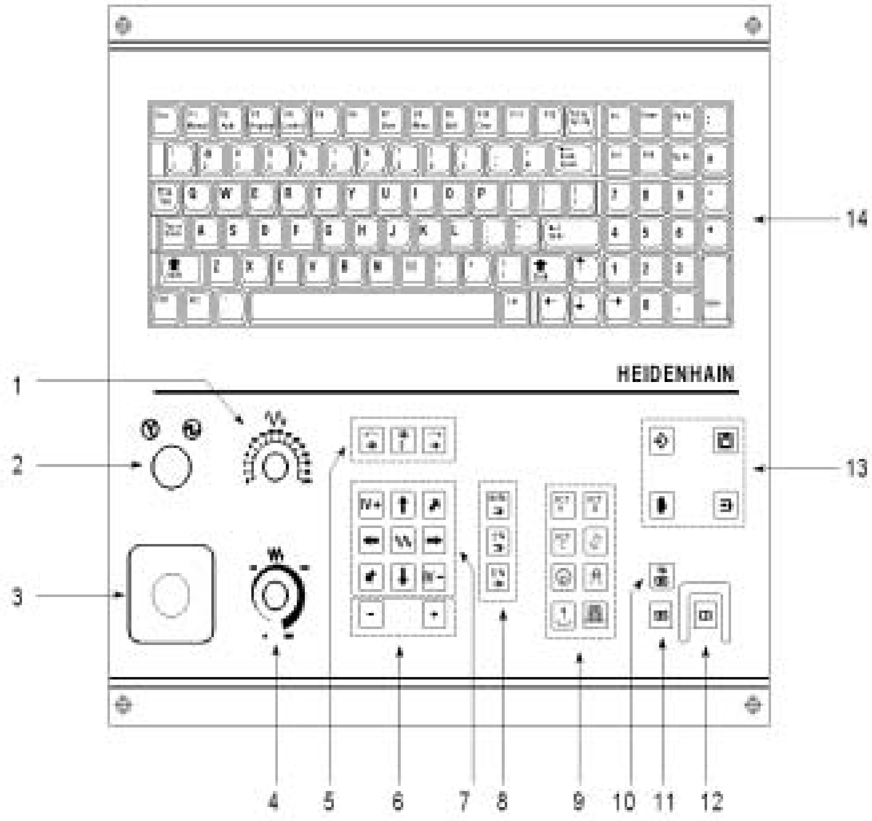
Submenu on screen:

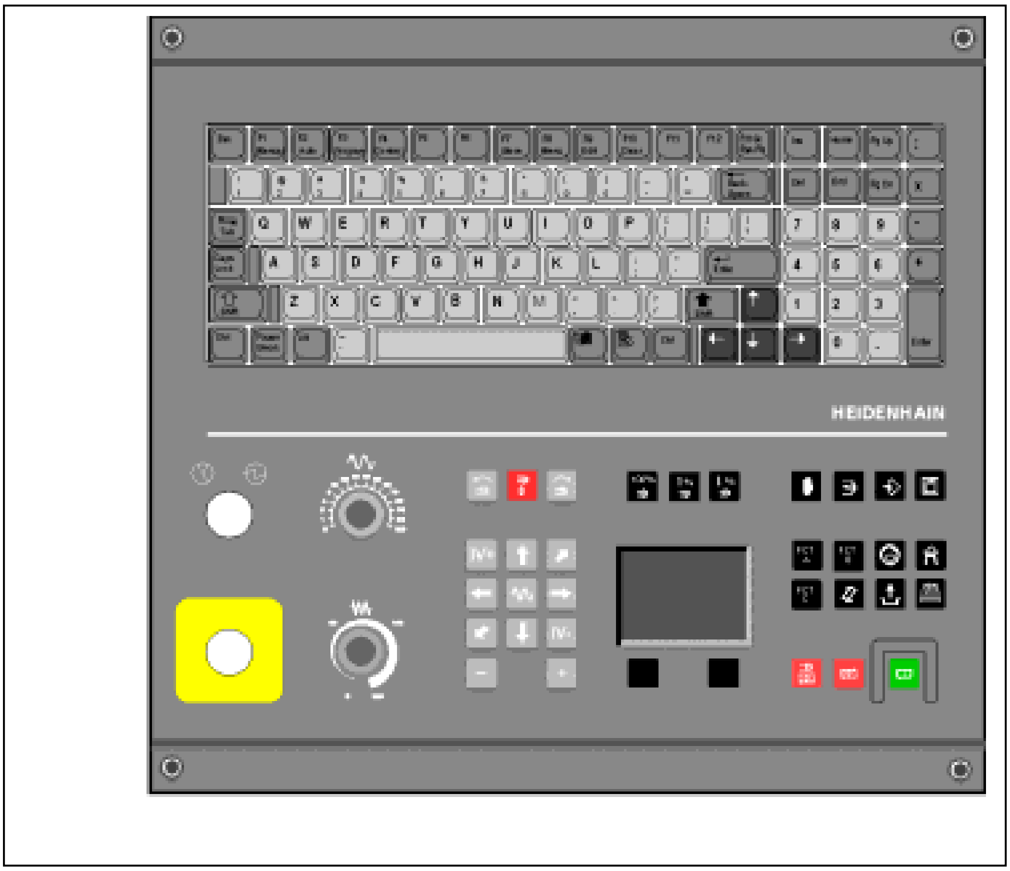
1 Exiting set-up menu (new settings are saved)
2 Reducing the value means
Image moves to the left, or downward
3 Increasing the value means
Image moves to the right, or upward
4 Back to menu 1 or menu 2 (new settings are saved)

Screen settings (preferences)

CONTRAST	Adjusting the contrast
BRIGHTNESS	Adjusting the brightness
H-POSITION	Adjusting horizontal image position
H-SIZE	Adjusting image size
V-POSITION	Adjusting vertical image position
V-SIZE	Adjusting image height
SIDE-PIN	Pin-cushion correction
TRAPEZOID	Correcting trapezium (keystone) distorsion
ROTATION	Correcting image rotation
COLORTEMP	Adjusting the colour temperature
R-GAIN	Adjusting red colour intensity
B-GAIN	Adjusting blue colour intensity
RECALL	Not used

3.3 Machine control panel





1 Rapid traverse rate control.
2 Machine ON
3 EMERGENCY STOP
4 Feed rate control
5 Spindle On Clockwise Rotation, Stop, On Anticlockwise Rotation
6 Axial movement keys for other axes
7 Axial movement keys and rapid traverse
8 Spindle speed control
9 Machine function keys; the function of the keys is determined by the machine tool builder. Please refer to your machine tool manual
10 Feed and spindle STOP
11 Feed STOP
12 START
13 Main modes of operation
14 Standard PC-keyboard

Note The keys (F11, F12, Prt Sc Sys Rq, Pause Break) must not be activated, because no function has been assigned to them.

3.4 Hand wheel HR410 (HCU)

1. Emergency stop button

2. Hand wheel
3. Safety keys

4. Axis selection keys

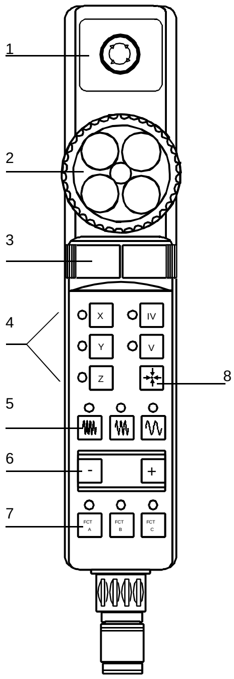
5. Keys for setting the feed (slow, medium, fast); feed rates are defined by the machine manufacturer

6. Direction into which the CNC moves the selected axis

7. Machine function keys
(defined by the machine manufacturer)

8. Key for taking over the actual position
- setting the actual value
- tool measurement
- Program Editor

The red LED displays indicate the axis and feed you selected.



3.4.1 Selecting/deselecting the hand wheel

The hand wheel is selected by pressing the left safety key. In the right top of the display appears HCU. For deselection let go the left safety key.

Hinweis
Die Bedienung wird vom Maschinenhersteller festgelegt. Beachten Sie Ihr Maschinenhandbuch.

3.5 Exiting a function

To exit a function or a mode, press "Menu" again,

or

3.6 Return to previous softkey level

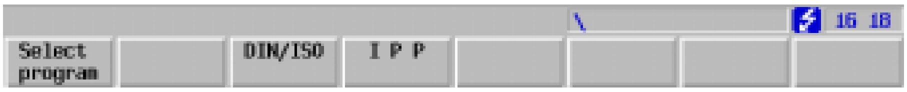
Press to return to previous softkey group (if one exists).

3.7 Superimposition of softkey groups

In addition to the current softkey group, other softkey groups may be active in the same mode.

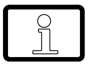
User softkey group for editing DIN/ISO programs
Press a mode key twice:

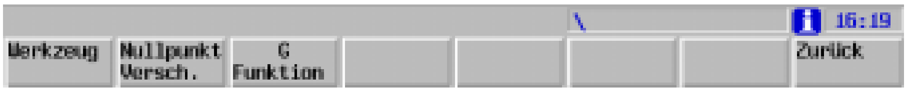
Example



Edit softkey group

Info softkey group



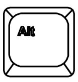


Indication of the tools entered in the tool table.

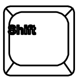
Indication of the zero offset table.

Indication of the list of G functions.

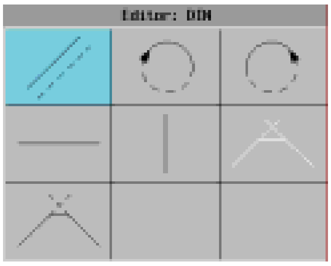
3.8 Switching between upper and lower case characters

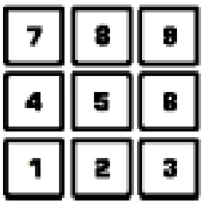


with



3.9 Making selections in the Easy Operate, ICP and IPP menus





1. Use the cursor keys to move left, right, up and down through the menu.
   To choose a menu item, press ENTER

2. or press one of the number keys 1-9. The ENTER key is not used in this case.

3.10 Quick mode selection

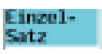
Two-digit mode number. (first digit: menu position, second digit: mode position)

Example: Select clock

3.11 Softkey Status

The status indicator of the softkeys shows the actual condition. For instance:

Softkey blue (Softkey active)



Softkey grey (Softkey not active)

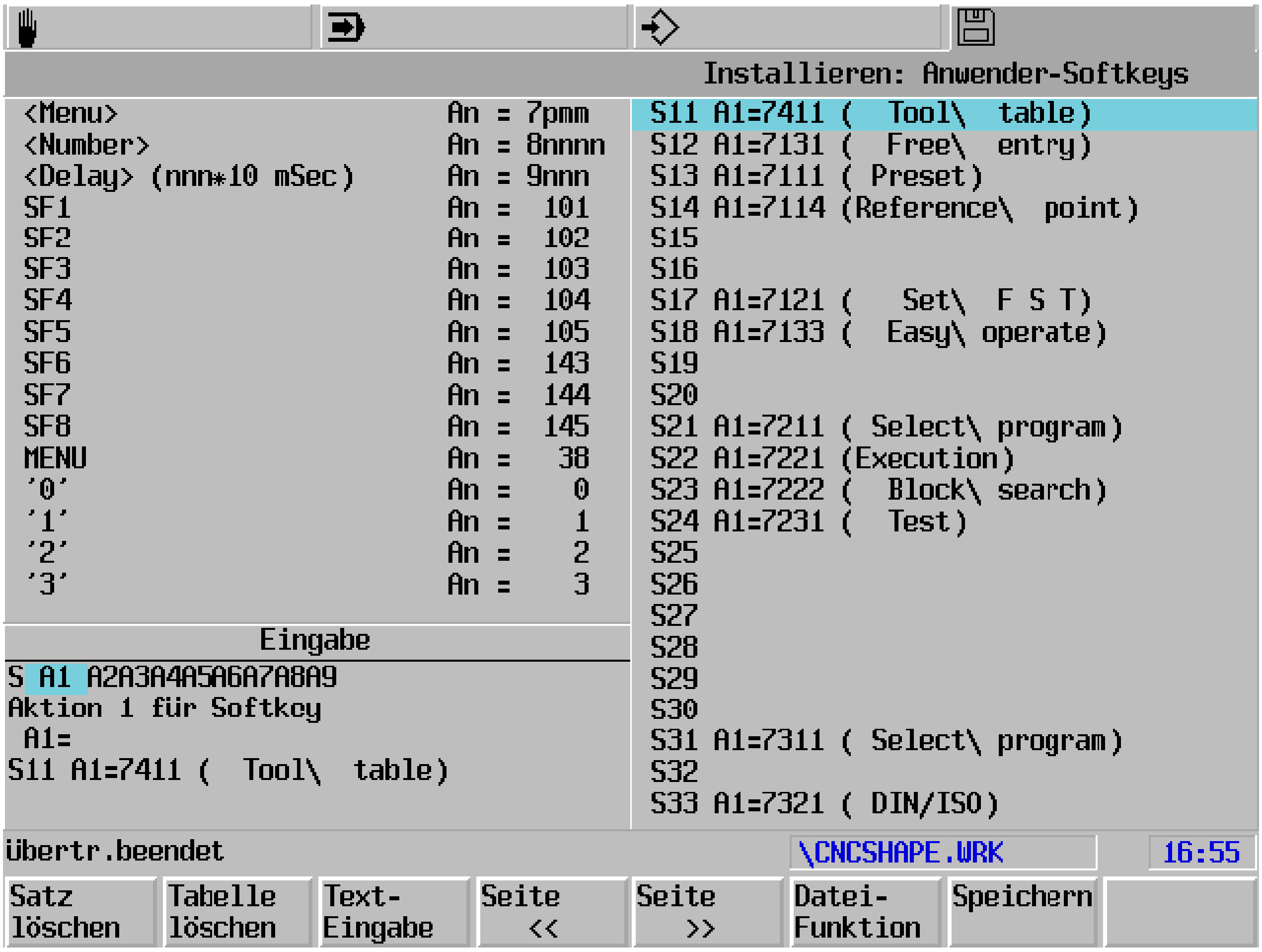
3.12 User softkeys

The user softkeys are used to initiate the most common functions quickly.

The user softkeys appear when the current process mode key is pressed a second time.

Pressing the key again causes the user softkeys to disappear. The previous softkey level becomes active again.

3.12.1 Defining the user softkeys




Search auxiliary window

Table with key

Key command	Action value t	Key command	Action value t
direct menu command	7000-7499	<-- (Cursor left)	49
number command	80000-89999	^ (Cursor Up)	51
Delay command	9000-9999	v (Cursor Down)	52
hor. softkey 1	101	--> (Cursor right)	50
hor. softkey 2	102	clear	15
hor. softkey 3	103	escape	166
hor. softkey 4	104	back space	154
hor. softkey 5	105	key pad "."	39
hor. softkey 6	143	key pad "="	40
hor. softkey 7	144	key pad "+"	45
hor. softkey 8	145	key pad "-"	46
menu	38	key pad "/"	47
number "0"	0	key pad "*"	48
number "1"	1	help	153

Key command	Action value t	Key command	Action value t
number "2"	2	store/select	53
number "3"	3	tab	171
number "4"	4	ASCII "("	1044
number "5"	5	ASCII ")"	1045
number "6"	6	ASCII "*"	1046
number "7"	7	ASCII "+"	1047
number "8"	8	ASCII ","	1048
number "9"	9	ASCII "-"	1049
process manual	139	ASCII "."	1050
process automatic	162	ASCII "/"	1051
process program	140	ASCII "0" | ASCII "9"	1052 | 1061
process control	141		
store	53	ASCII "A" | ASCII "Z"	1068 | 1094
enter	54		
insert	168	ASCII "a" | ASCII "z"	1101 | 1127
home	176		
page Up	170		
delete	163		
end	165		
page Down	169		

Process level Manual: S11 to S18 (Softkey 1-8)
Process level Automatic: S21 to S28 (Softkey 1-8)
Process level Program: S31 to S38 (Softkey 1-8)
Process level Monitor: S41 to S48 (Softkey 1-8)

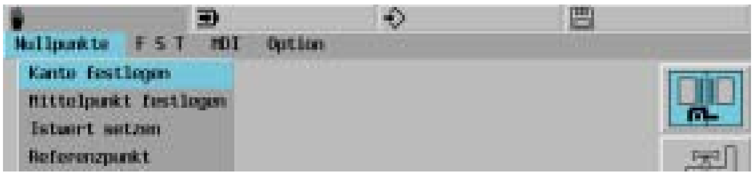
Entering softkey text:

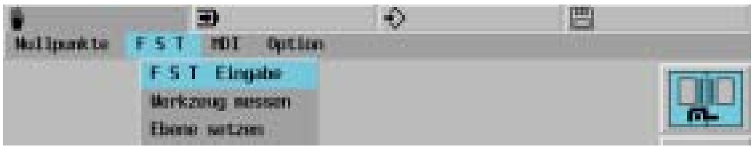
- The softkey text should be in brackets.
- 2 lines, not more than 9 characters per line.
- Character "\" defines the line break.

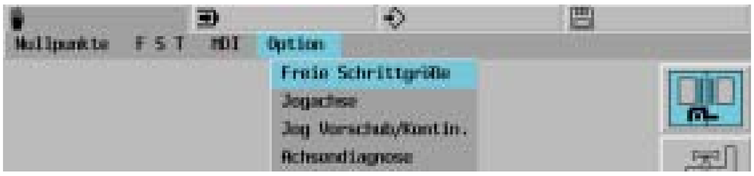
Examples

SF1:	S31 A1=38 A2=1 A3=1	Select file/program
SF3:	S33 A1=38 A2=2 A3=1	DIN/ISO input

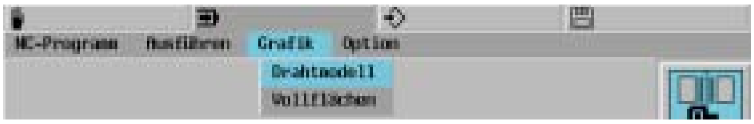
3.13 Process level Manual



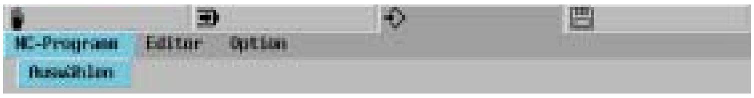


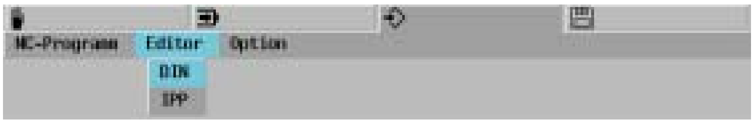


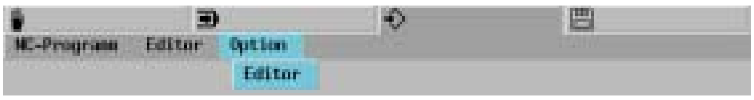
3.14 Process level Automatic



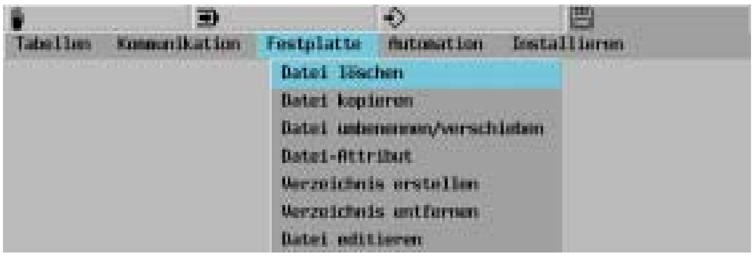
3.15 Process level Program

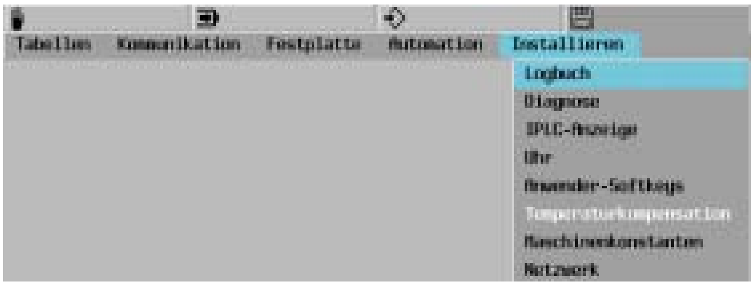






3.16 Process level Monitor





3.17 Starten und schließen der Software auf ein Doppel-Prozessor-System

Bei eine erstmalige Installation beachten Sie Ihr Technisches Handbuch, Installieren der Software auf ein Doppel-Prozessor-System.

3.17.1 Steuerungsbetrieb starten

Die Steuerungssoftware kann gestartet werden durch Doppelklicken auf die neue Ikone auf dem

Desktop 

3.17.2 Steuerungsbetrieb schließen

Drücken Sie vorab auf **Not Aus**, damit sichergestellt wird daß die Motoren-Hardware ausgeschaltet wurde!

Klicken Sie die rechten Maustaste auf einer beliebigen Stelle im Bildschirm.

Beim Cursor erscheint: 

Klicke 

Bemerkung
Falls Sie die Steuerung gerade einschalten, brauchen Sie nicht zu warten bis die Steuerungs-Software aufgestartet ist. Sobald der Aufstartvorgang läuft, können Sie bereits Ctrl/Esc drücken wodurch auch nachvolgende Vorgang erreicht wird.

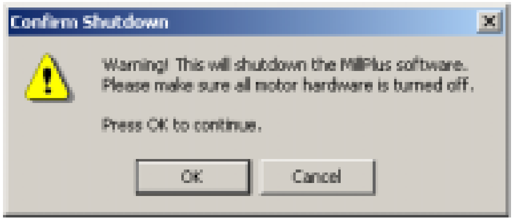
Rechtsunten erscheint auf dem Bildschirm: 

 Klicken

Auf dem Bildschirm erscheint:

Klicken

Klicke →
Bestätigungsfenster erscheint:



Bestätigung Schließen

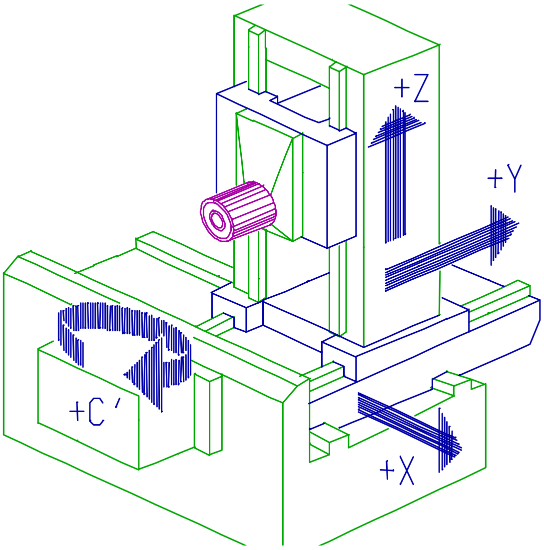
Warnung! Die MillPlus-Software wird abgeschlossen. Bitte überzeugen Sie sich daß alle Motoren-Hardware ausgeschaltet wurde.

Drücke "OK" um fortzufahren.

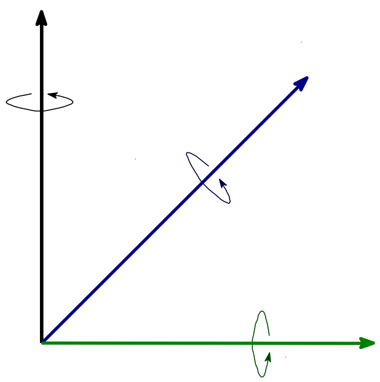
Klicken

4. Workpiece coordinates

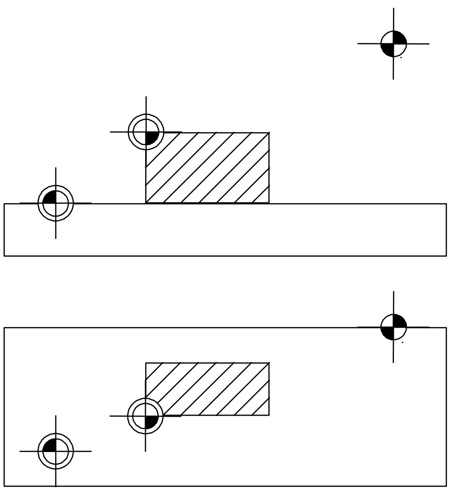
4.1 Coordinate system and direction of movement



4.2 Axes

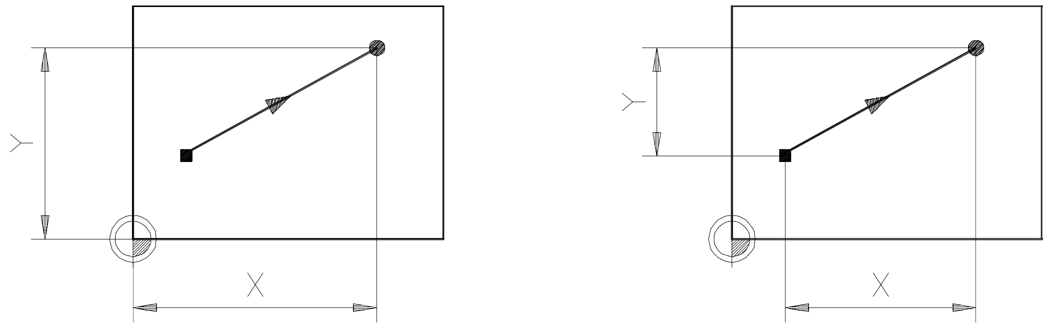


4.3 Zero points



R Referenzpunkt
M Maschinennullpunkt
W Werkstücknullpunkt

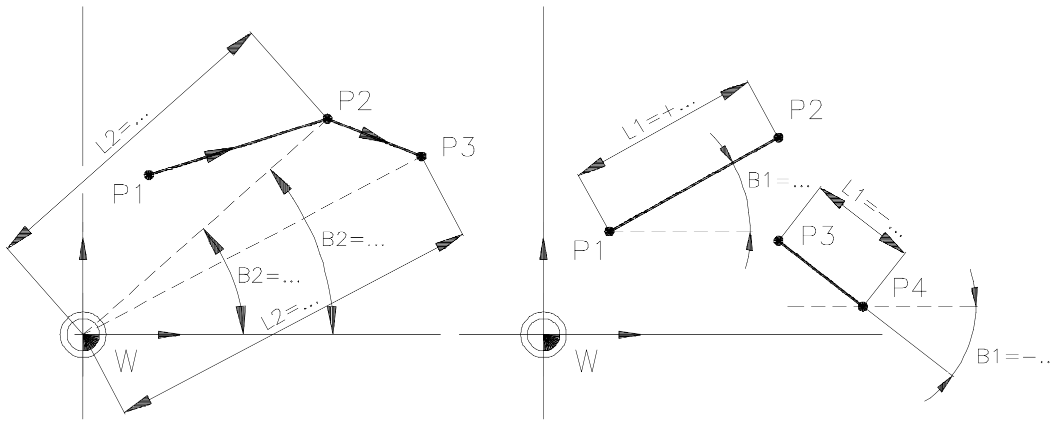
4.4 Cartesian coordinates



Absolute coordinates (G90) Incremental coordinates (G91)

Wordwise absolute/incremental programming (X90,X91,Y90...) does not depend on the modally valid G90/G91 system of measurement.

4.5 Polar coordinates



Absolute coordinates (G90) Incremental coordinates (G91)

Programming in polar coordinates is not affected by wordwise absolute/incremental programming.

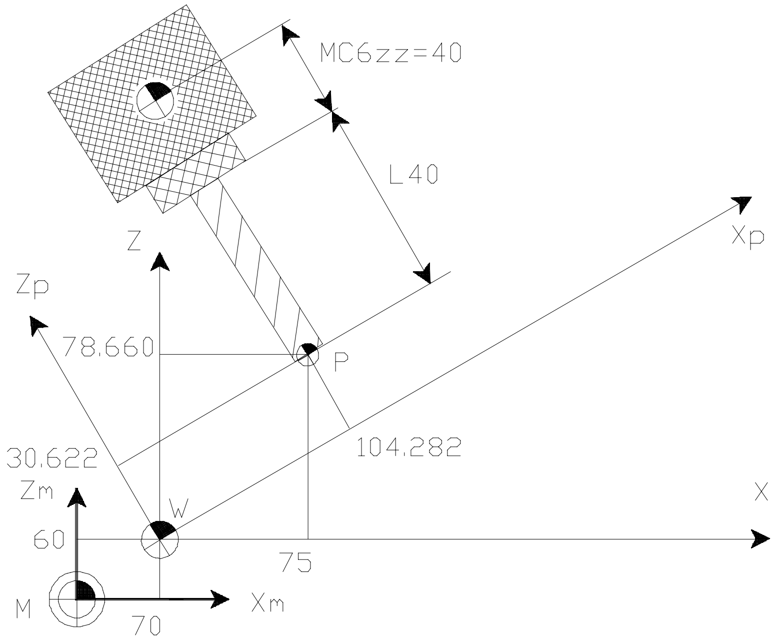
Note

If a pole point has been programmed (see G9), program blocks that use polar programming (angle and length) no longer refer to the zero point, but to the most recently programmed pole point.

4.5.1 Assignment of polar coordinates

Polar coordinates	Angle reference axis	Movement B1=+
X Y G17	+X	+X to +Y
Z X G18	+Z	+Z to +X
Y Z G19	+Y	+Y to +Z

4.6 FSP coordinates



The position display on the screen can change between the position in the G7 plane (Xp,Zp) or in machine coordinates (X,Z).
Both are based on the active null point G52 + G54 + G92/G93.

5. Start machine / reference point

5.1 Start machine (example)

Main switch ON

Power supplied to controller and measuring system.

Danger! High voltage!

Do not touch any exposed components in the switchgear cubicle as they may be live.

Before starting or operating the machine, ensure that noone is likely to be endangered as a result.

Ensure that only authorised personnel operate the machine!zzz

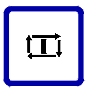
Release the EMERGENCY STOP switch.

Machine ON (keep key depressed) and press CLEAR.

5.2 Approach reference points

Selection of

one or more axes



Approach reference point (RPF)

Note

Beware of collision!
The software limit switches are not active prior to "Approach reference points", and the axis slides are able to run up to the mechanical end stop.

Before "Approach reference points", the machine operator should ensure that no collision with the machine will occur when approaching the reference points.

5.3 Select level

The active plane can be selected by using the softkey. The functions G17, G18 or G19 are decisive in the machining program and the softkey setting is overwritten.

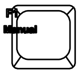
Selection level

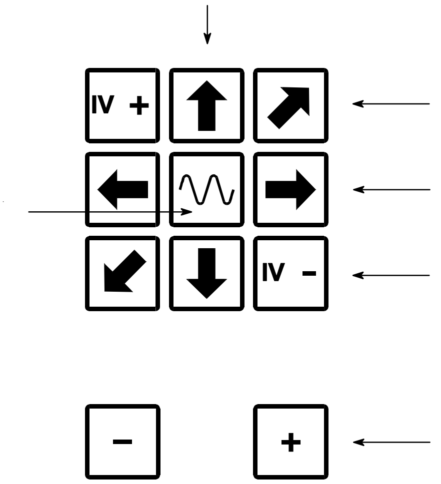
6. Manual operation

The machine axes can be moved continuously and manually by adjustable movement steps. The speed of movement can be regulated using the feed override. It is also possible to move two axes simultaneously. The work spindle may also be moved manually. Other axes, e.g. the fifth axis or spindle, must first be selected.

6.1 Move axes

The axes are moved using the axis movement keys.





1. Z-axis   2 Y-axis
3 X-axis   4 Axis 4
5 Axis 5   6 Rapid traverse

Note

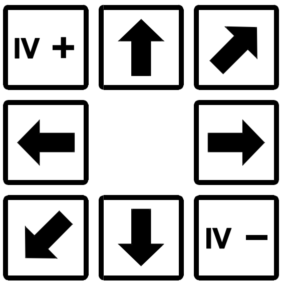
Select axis 4 with mc153.
Select axis 5 with mc154.

6.1.1 Step movement, continuous movement

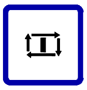
It is determined whether the machine axis moves stepwise or continuously when the axis movement key is depressed.

6.1.2 Continuous movement

Kontinuierlich verfahren mit Achsenbewegungstaste und Start. Die Achse verfährt bis sie angehalten wird.

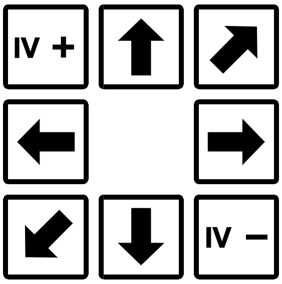


Press at the same time as

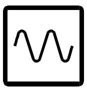


-Feed from MC
-A maximum of 2 axes can be moved at the same time.
-Stop using 'Feed STOP' or 'Feed and Spindle STOP' keys

6.1.3 Rapid traverse motion

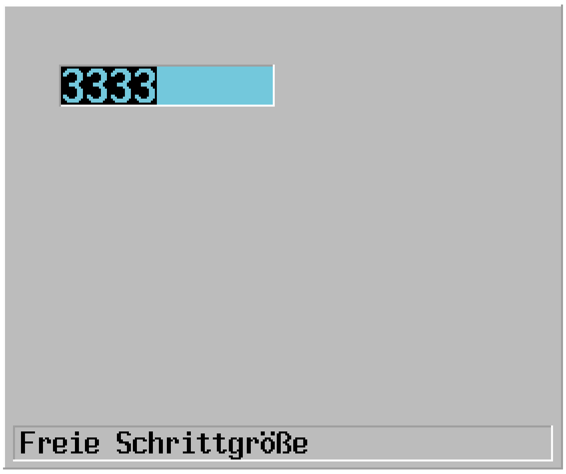


Press at the same time as



6.1.4 Free step size

The free increment allows you to set the appropriate increment for your machine.



Use free step size:

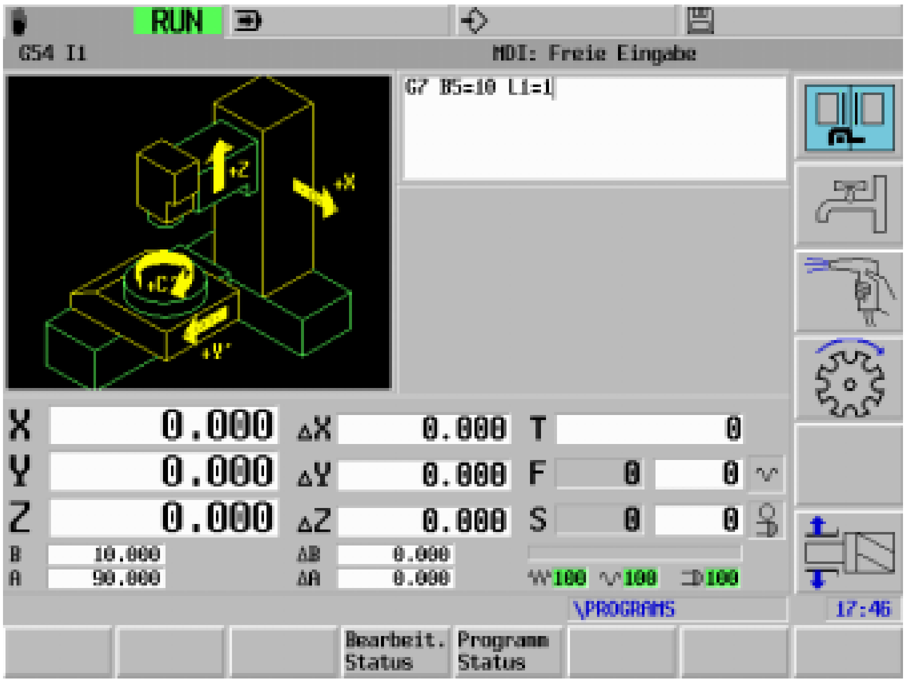
6.1.5 Move spindle and other axes

−   +

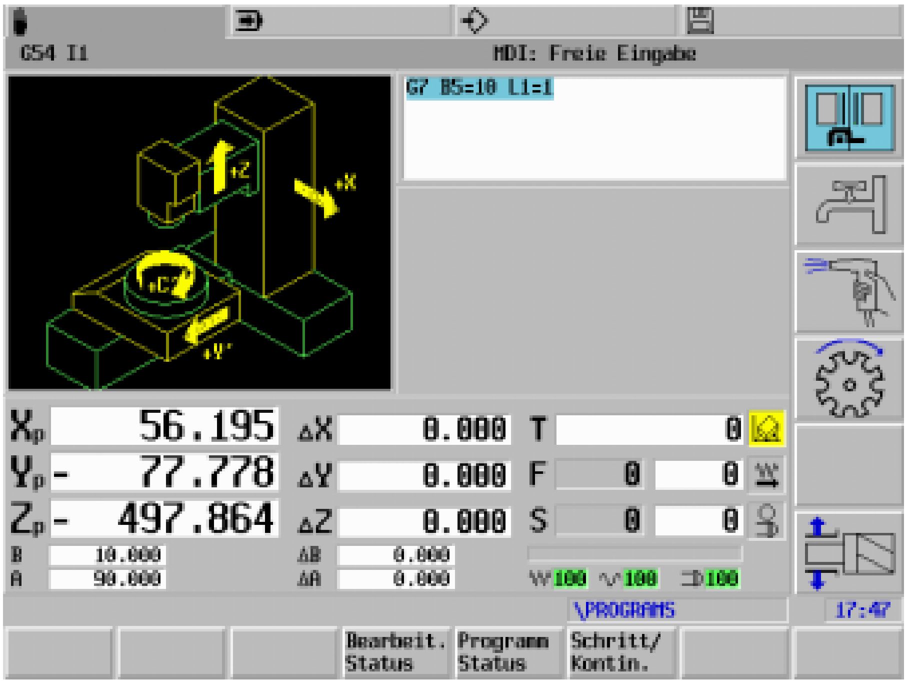
6.2 Procedure in FSP

It is possible to proceed on the FSP level or in the machine axes after enabling the "Free process level"

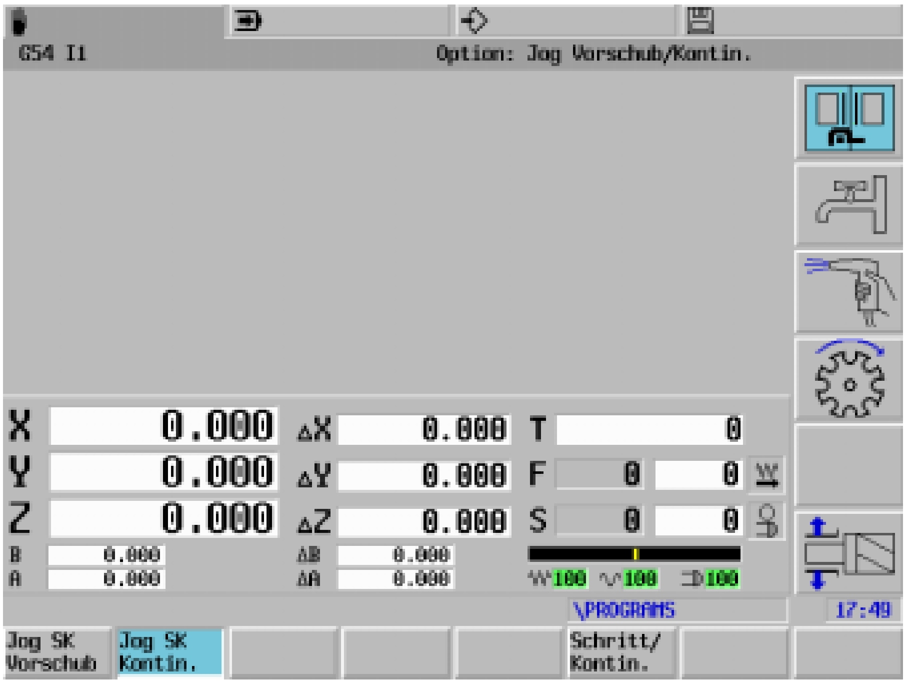
Procedure on the free process level.



Procedure in the machine axes.

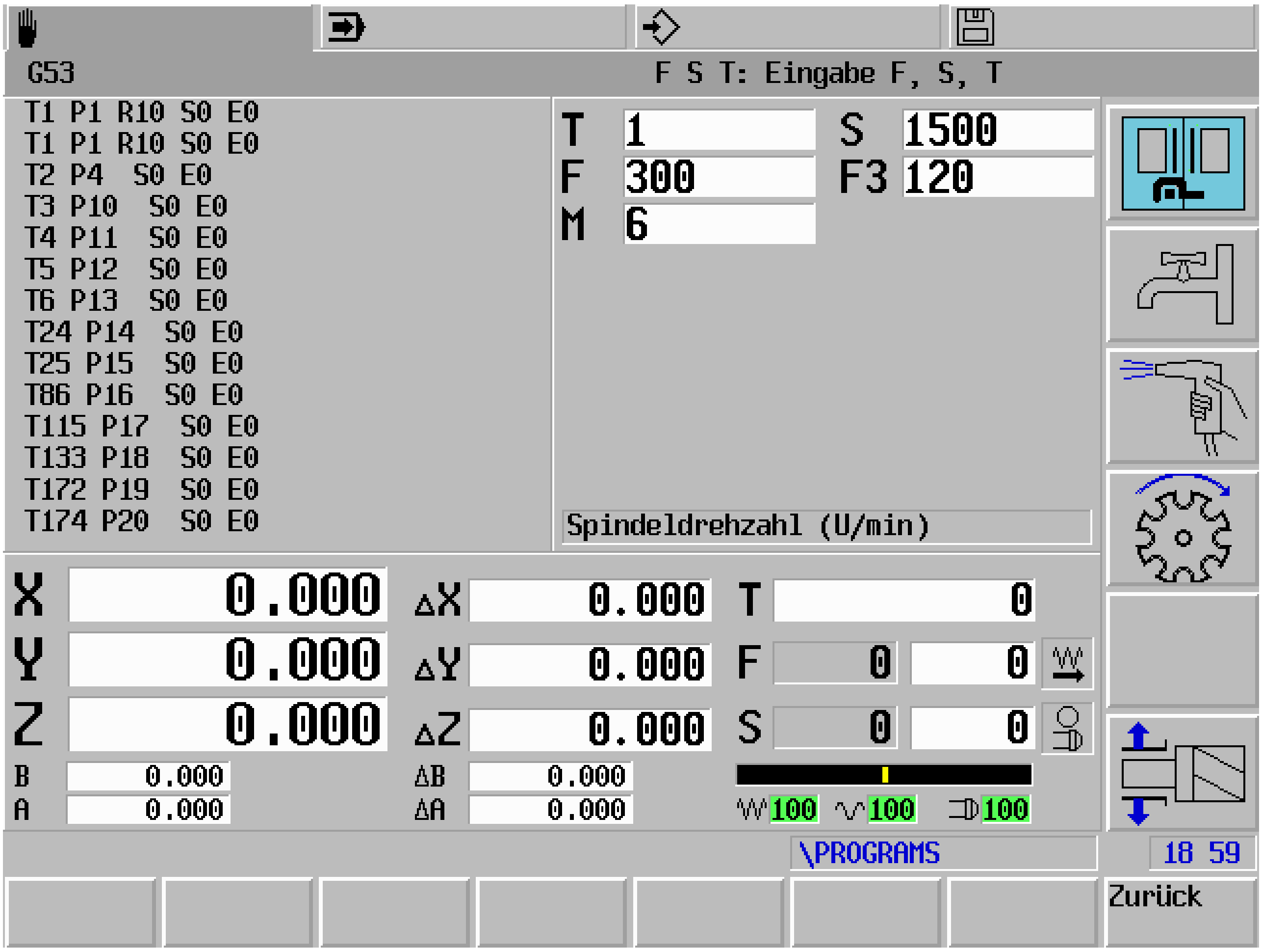

G54 I1
MDI: Freie Eingabe
G7 B5=10 L1=1
+Z
+X
+C2
+Y
Xp 56.195
Yp - 77.778
Zp - 497.864
B 10.000
A 90.000
ΔX 0.000
ΔY 0.000
ΔZ 0.000
ΔB 0.000
ΔA 0.000
T 0
F 0 0
S 0 0
100 100 100
\PROGRAMS
17:47
Bearbeit. Status
Programm Status
Schritt/ Kontin.

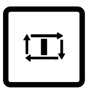
6.3 Switch over rate of advance/continue procedure



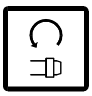
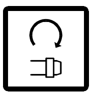
6.4 F, S, T input

Entry of tool number, spindle speed, feed and M-function.





Activate tool change

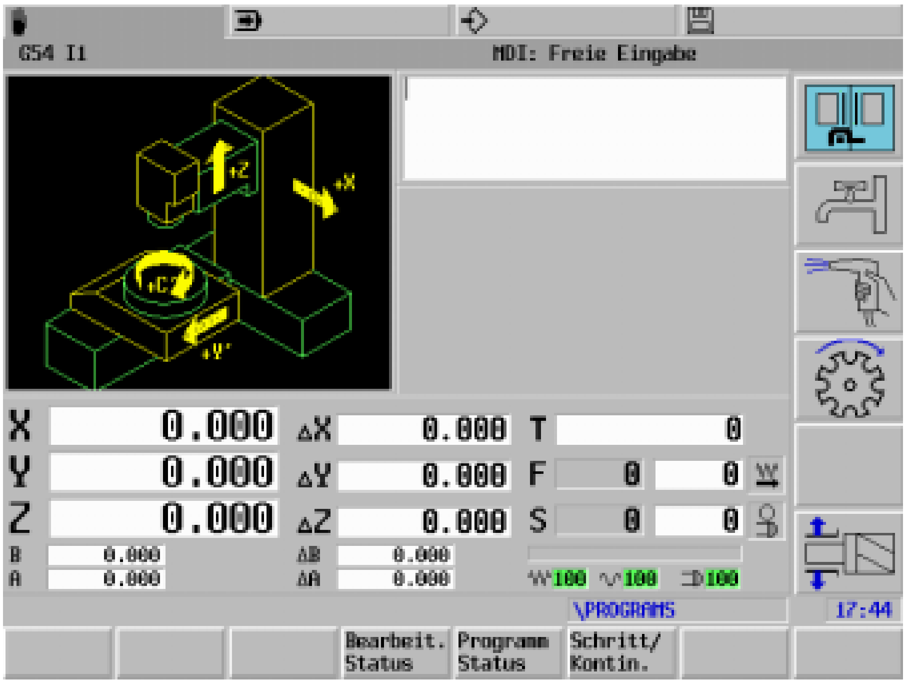



Start spindle (M3 or M4)

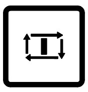
7. Free entry (MDI)

7.1 Free entry

Entry of an instruction in the command line followed by execution.



Enter address and address value from keyboard.



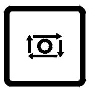
Execute program block.

When execution of the block has been completed, the Free Entry mode remains active.

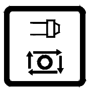
Note

Please refer to the chapter Easy Operate.

7.2 Cancel block (MDI)



or



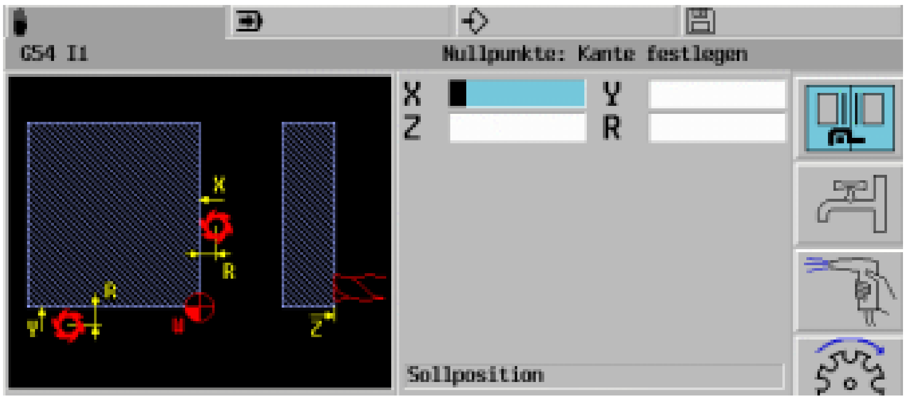
Interrupt program block run

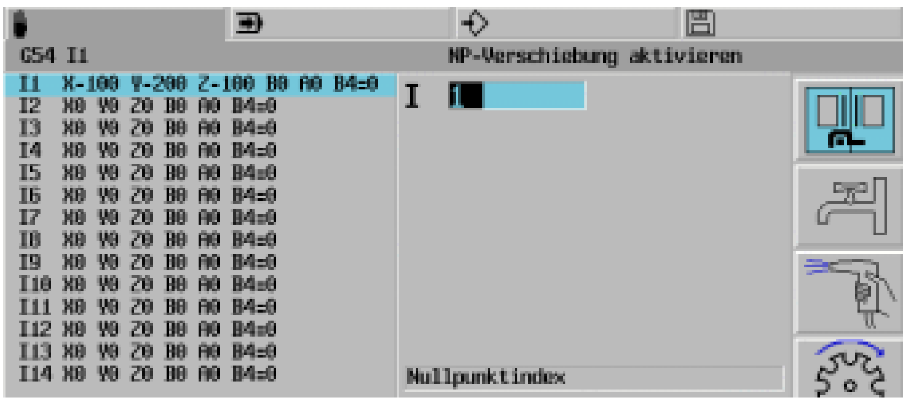
The current block is interrupted.

8. Set axis value

With "Touch side", "Determine centre" and "preset axes" it is possible, after selection of softkey "Select zeropoint", to undo the current zero offset.

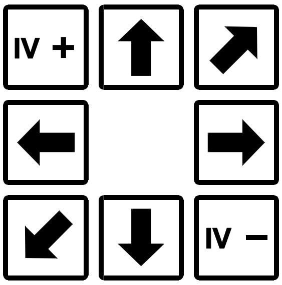
8.1 Determine side





Input W activate zero offset

Aktiviere
NPV-Nr.



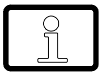
Approach side

Enter offset value (X, Y, Z, R)

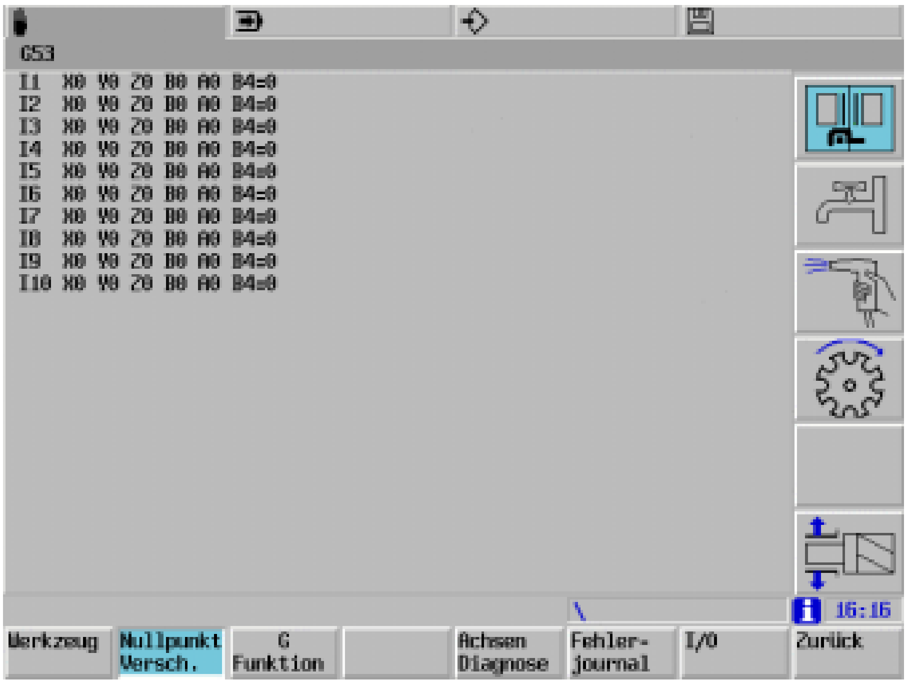
Press the softkey to indicate from which direction the side was approached. The zero offset for the selected axis and direction is calculated and stored in the zero offset memory. The offset value in the current axis screen is also updated.

to

Display of zero offset memory.



Nullpunkt
Versch.



8.2 Determine centre

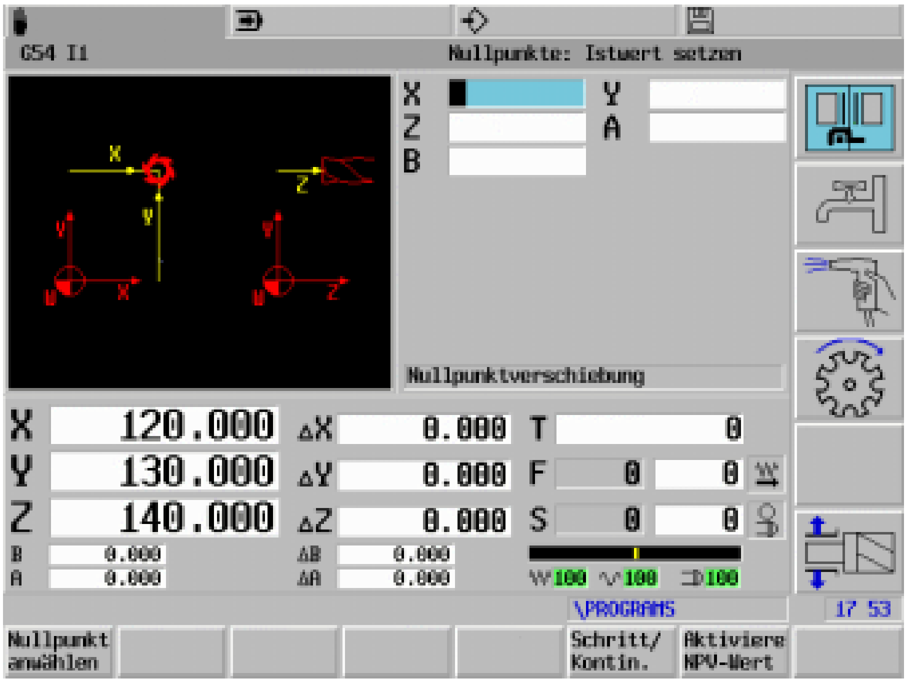
Procedure: as for Determine side.

Activate values in main plane

Activate values in tool axis

8.3 Set actual value

To machine a workpiece, the machine zero points must be synchronised with each other. The workpiece zero point is determined by the machine operator and passed to the controller via the zero offset.



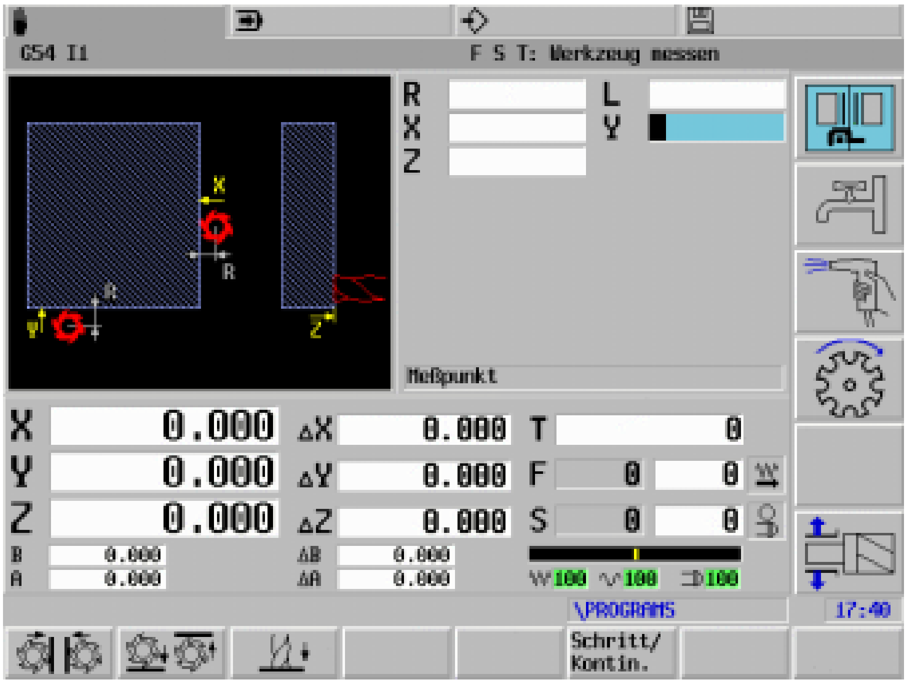
- Select zero point
- Approach position using axis movement keys
- Enter the actual axis values

Update the axis display with the present axis values and add the zero point to the zero offset table.

8.4 Measure tool

The Measure tool function is used to determine tool offset values (radius and length) for the active tool. The offset values are added to the Tool table.

Example: tool length measurement.
- Activate machining levels (e.g. G17)
- Activate zero offset (e.g. G54 or G54 I10)
- Change tool in the spindle (e.g. T1)



The actual tool values are shown under R and L
Radius measurement:
- Enter reference position (e.g. X20).
- Approach reference position.
- Establish tool radius, using softkeys

Length measurement:
- Enter reference position (e.g. Z0).
- Approach reference position.
- Establish tool length, using softkey

Note

Please refer to the chapter Tools.

Note

Please refer to the chapter Tools.

9. Data input / output and file management

9.1 Data transfer

9.2 Coordinate controller with peripheral device

Note

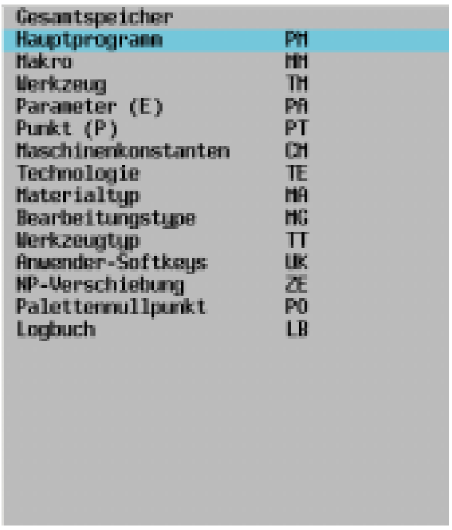
Machine constants for units:

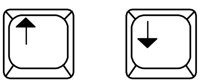
900-908	910-918	920-928	780-783 930-936	790-795	797-799

Block number > 9000, refer to user machine constant list (MC772-774).

9.3 Reading

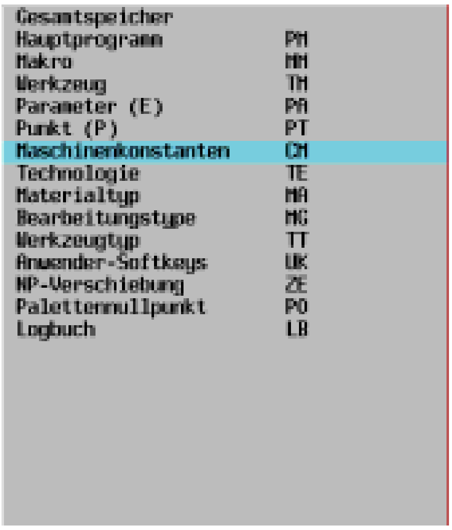
9.3.1 Reading in the program (PM,MM)

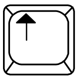
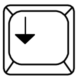




Select the main program or macro from the list.

9.3.2 Reading in tables (TM..PO)



Select a table from the list.

Note

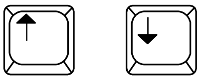
When the technology tables have been read in, they must be saved on the hard disk enabling them to be activated after the controller has been switched on/off (the CNC always saves in the startup directory).

9.4 Output program

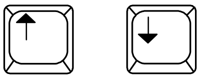
9.4.1 Data back-up

The user should regularly save his programs (PM and MM) and important data (e.g. technology data, machine constants, tools, etc.) onto his PC or onto a floppy to prevent irretrievable data loss.

9.4.2 Reading out the program (PM,MM)

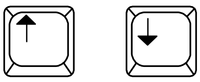


Select the main program or macro from the list.



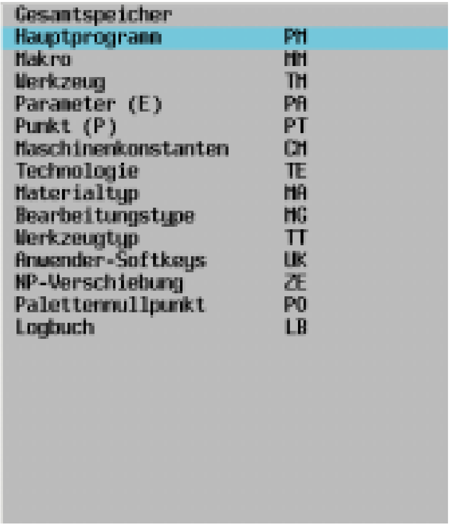
Select a program

9.4.3 Reading out a table (TM-LB)



Select a table from the list.

9.5 Memory name abbreviations



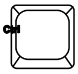
Note

- At mc84=0 the zero offset identifier is ZO.ZO and at mc84>0 ZE.ZE.

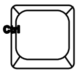
9.6 Mini-PC

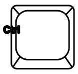
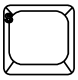
3,5" disk drive

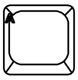
9.7 Select files

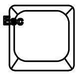
 with Select a file

or

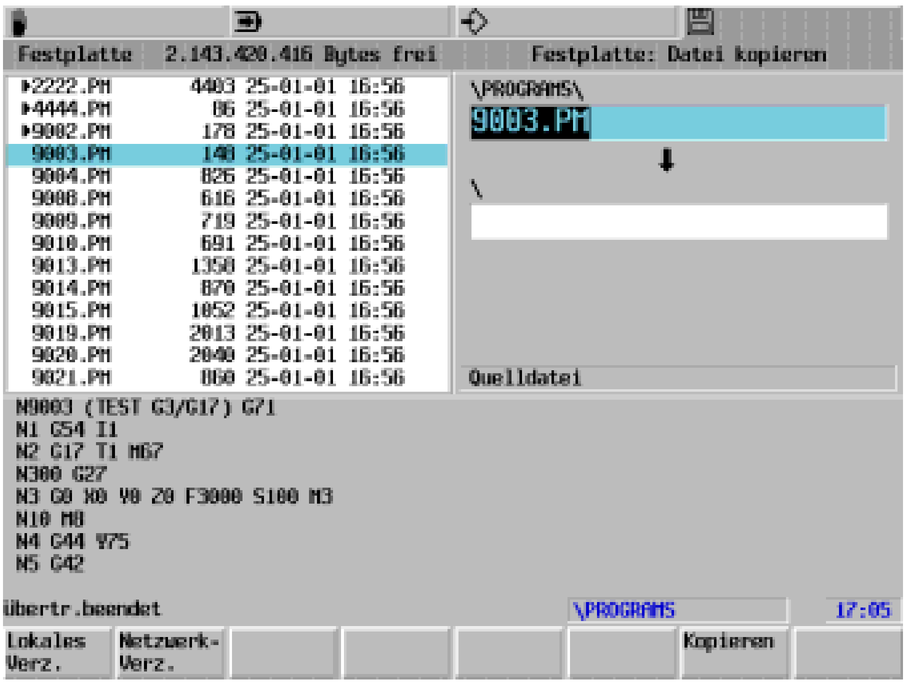
 with Select a file. A toggle function (between <selected>/<not selected>) for the file where the cursor is currently positioned; the cursor moves to the next file.

  with All files in the directory are selected



 The current selections are cleared

Files that have been selected have a ! character in front of their name.



Files can be selected in this way in the following menus:

Fixed disk:

- Delete file
- Copy file
- Rename/move file
- File properties

Communication:

- Upload
- Download

function is then applied to all the selected files.

Notes:

A destination file that is entered when more than one source file has been selected will be ignored. The destination is then assumed to be the destination directory.
ted file at the current cursor position will not be included in the operation.

Copy and Rename/Move have the same format. If, during a Copy or Move operation, one of the selected files already exists in the destination directory, a new softkey group, as shown below, is displayed.

lways appears when a file exists until the

key is pressed or all files have been processed. If **F4** is pressed, the remaining steps are performed automatically and no further confirmation will be required.
Selections can always be deleted by pressing the **<Esc>** key at any time; this also has the effec of cancelling the processing of any more files.

9.8 File management

Speicherstruktur

CNC Speicher

Festplatte (Massenspeicher)
- *.PM
- *.MM
- Verzeichnisse

Bearbeiten
DRAM
- 1.PM
- *.MM

DNC
Remote

Dateiverwaltung
- Verzeichnis
- Kopie usw.
- Ethernet

Editieren
DRAM
- 1.PM/MM

Datenübertragung

A hierarchical directory structure is present on the hard disk supplied. The structure looks like this:

```
\─┬──── STARTUP
 └──── WORK
 └──── TEMP
```

The technology tables and subprograms in the startup directory are loaded into the CNC DRAM when the CNC is initialised.

Executing a faulty program can lead to dangerous situations.

In the Automatic and Edit operating modes, the programs are always selected from the harddisk. The directory can be changed in the modes of operation.
Selected programs are loaded into the working memory (DRAM).

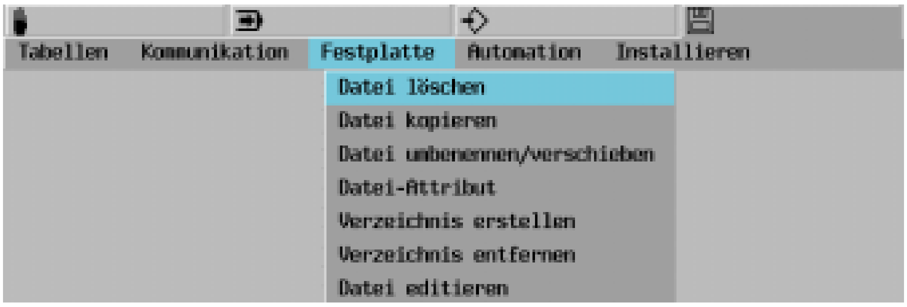
Notes

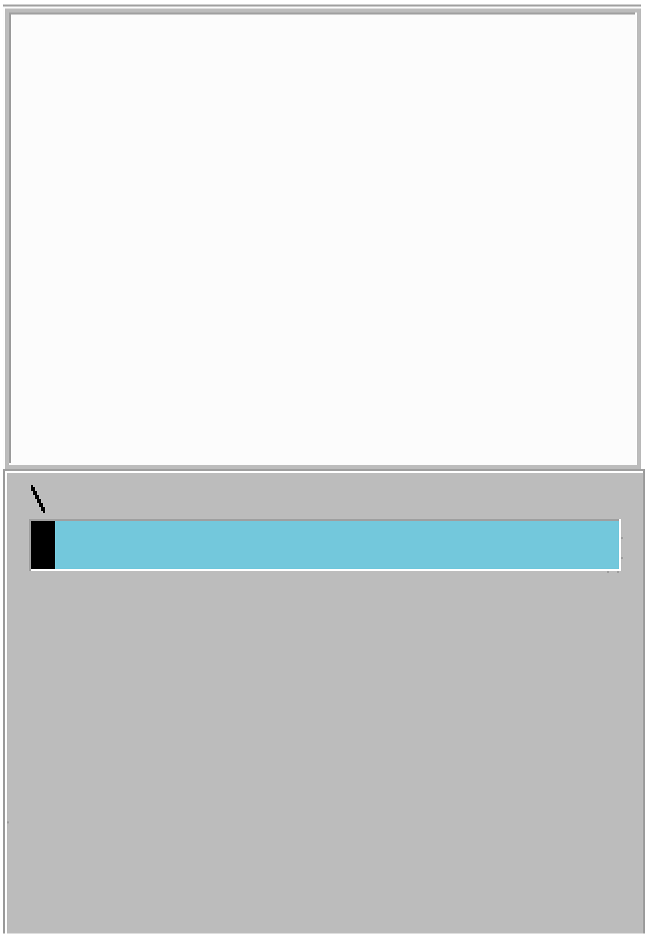
- If a faulty file is found during loading, loading is stopped.
- Programs are checked as they are loaded. If an error occurs during loading, an error message (in brackets) is appended to the relevant program block.
  Example: N.. G301 (O... "Original block contents incorrect")
- The startup directory contains the technology tables and the IPP setup macro. We recommend not to store other programs in the startup directory. The only exceptions are e.g. subprograms which are invoked in several main programs.
- During copying, renaming or loading, the program number in the first program block is adapted to the file name, provided the name of the file matches a valid program number.

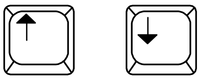
- Main programs (invoked with G23) and subprograms (invoked with G22) have to be in the directory of the active main program.
- When leaving the editor, the program will ask whether the changes are to be stored. Changes in the active main program and in the accessory subprograms are stored automatically.
- Large programs that do not fit into the working memory have to be executed with softkey "CAD-Betrieb". However, when in a program that is not executed in "CAD-Betrieb", it is still possible to invoke and execute a large program via G23.

9.8.1 Delete file

Only programs in the current directory can be deleted.
When deleting a complete directory (*.*), the contents of the directory are deleted. The directory itself is not deleted.





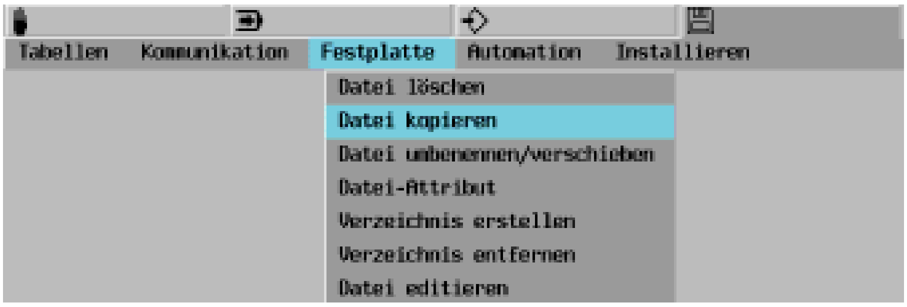

Select program or enter program number

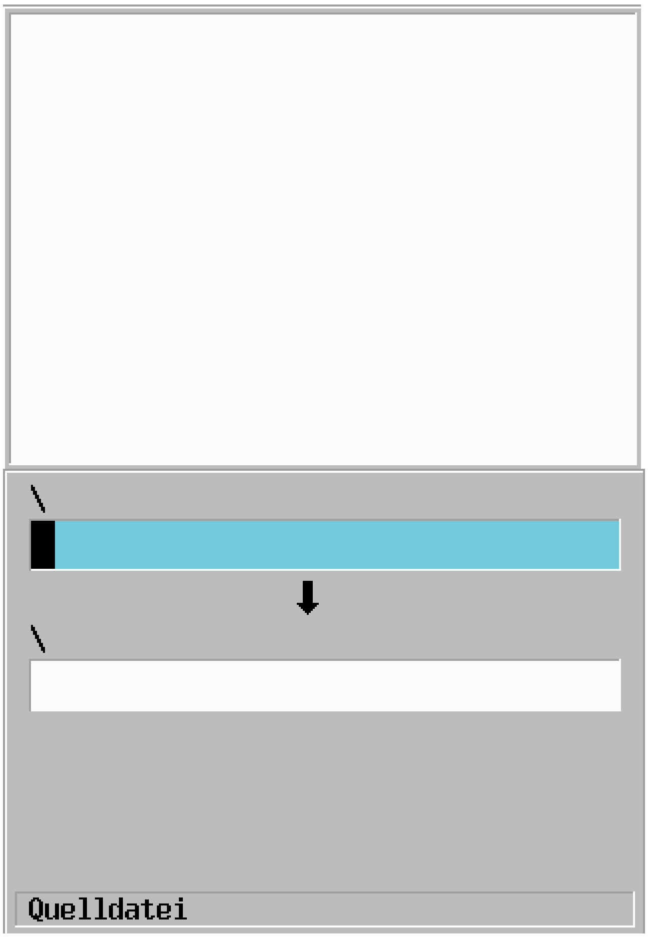
Delete program No or Yes.

9.8.2 Copy file

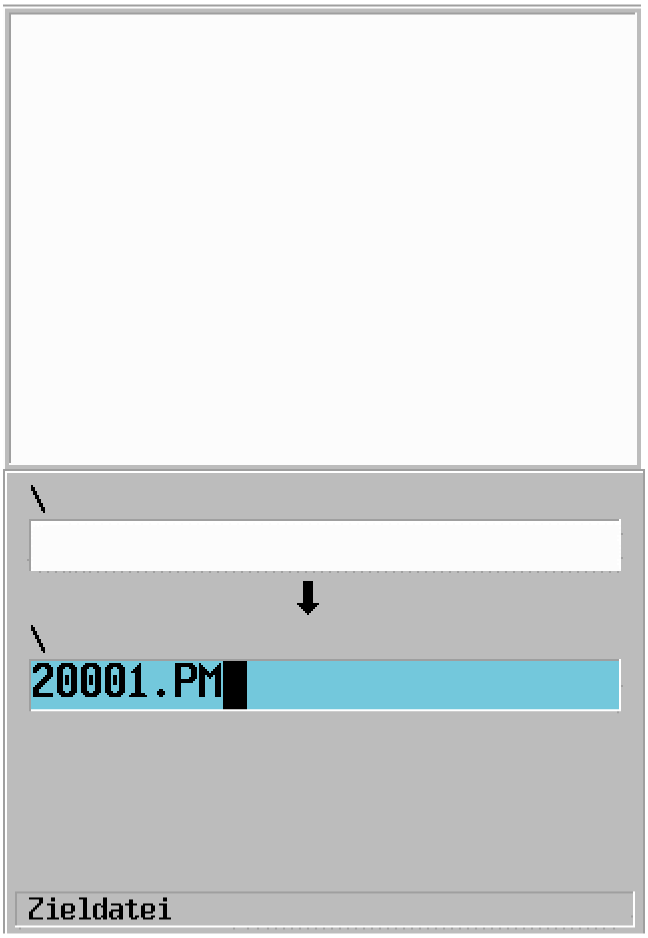
The <File: copy> function is identical regardless of whether the file is being copied across the Ethernet or from one location to another on the local hard disk. The choice of source or target directory determines whether the Ethernet is used or not.

Copy to current directory:



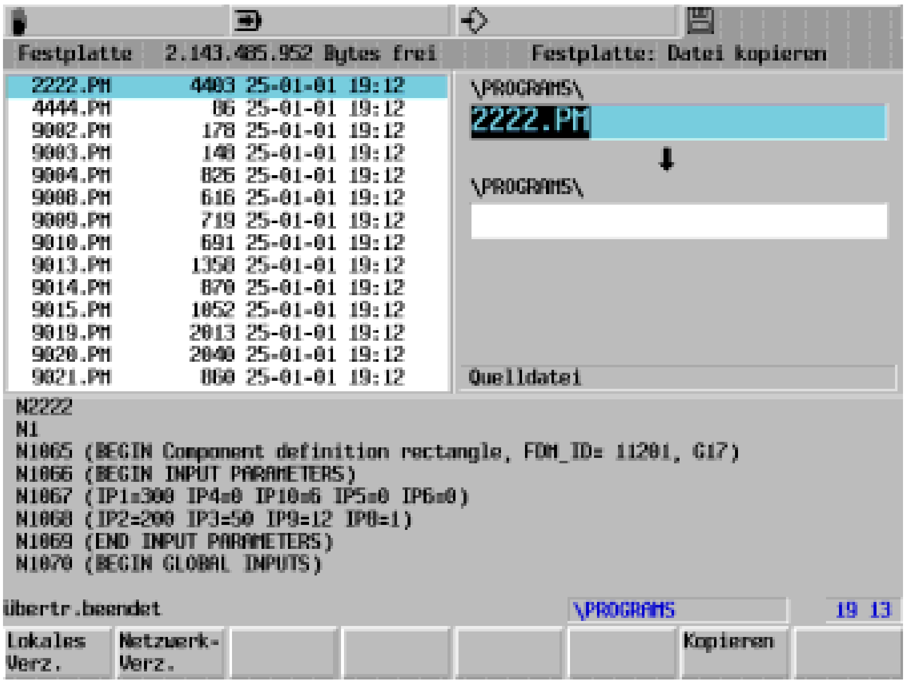


Enter name of target file (e.g. 20001.PM):

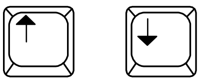

\
\
20001.PM
Zieldatei


Kopieren

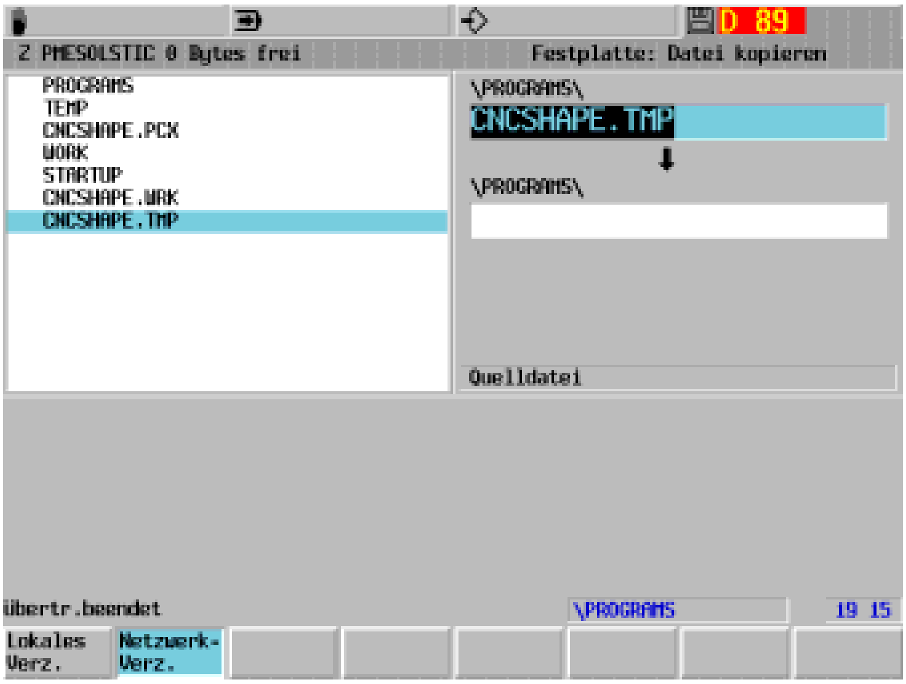
Copy across the Ethernet:



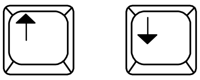
source file:



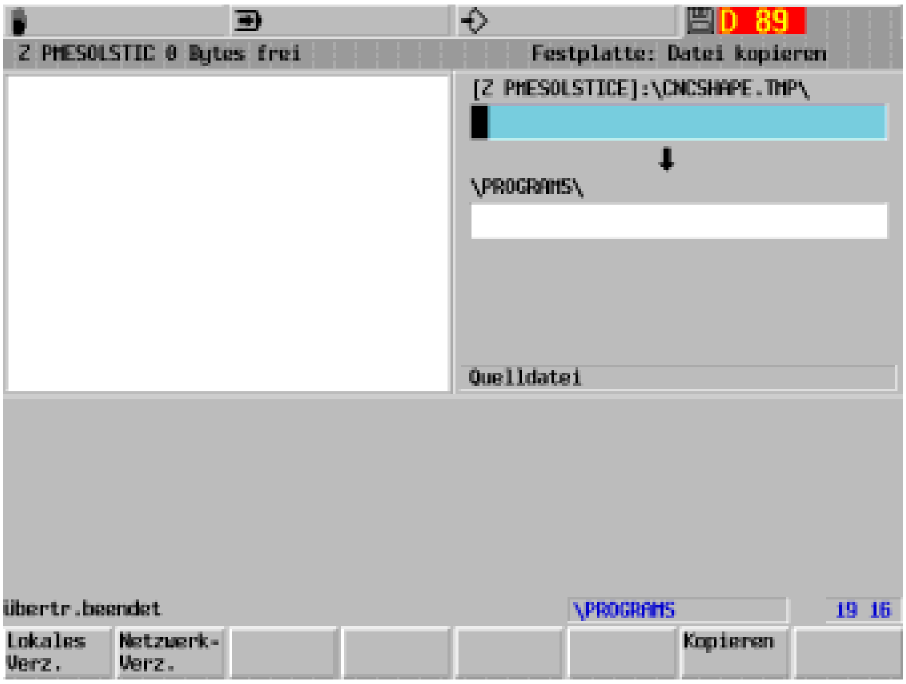
Select directory

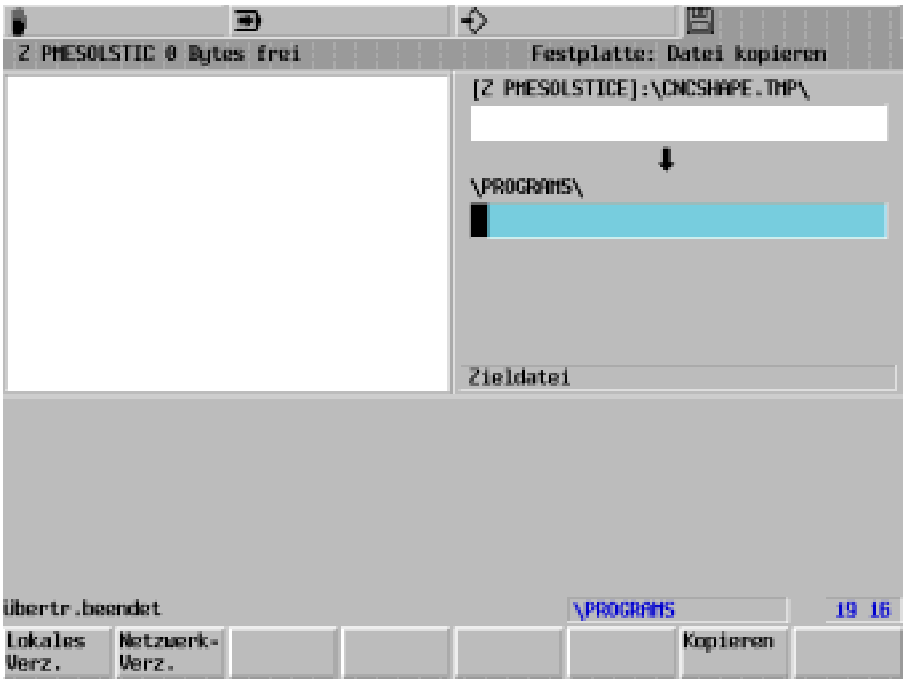


Open directory

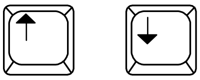


Select program

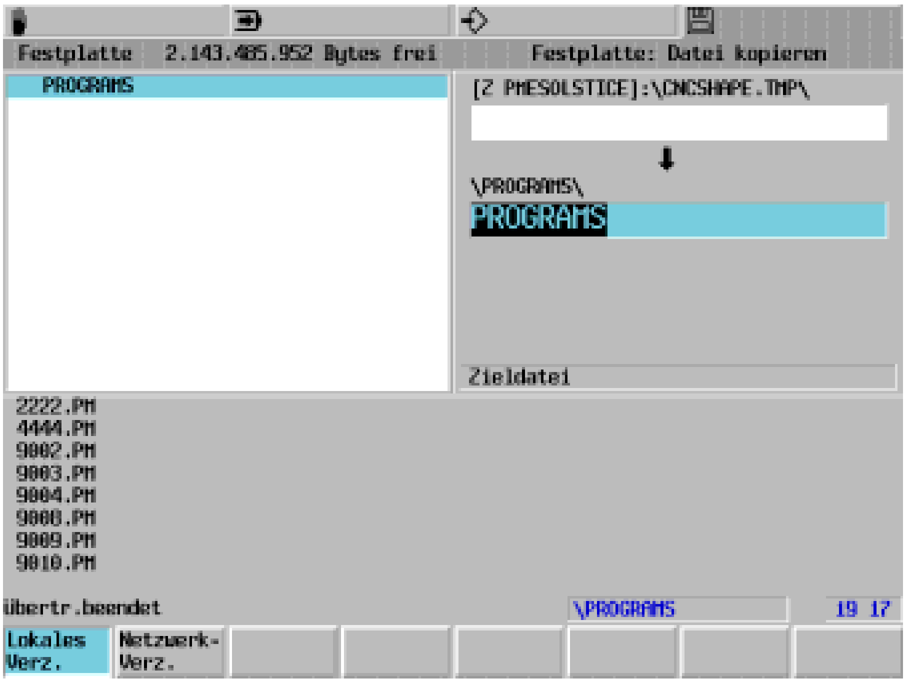




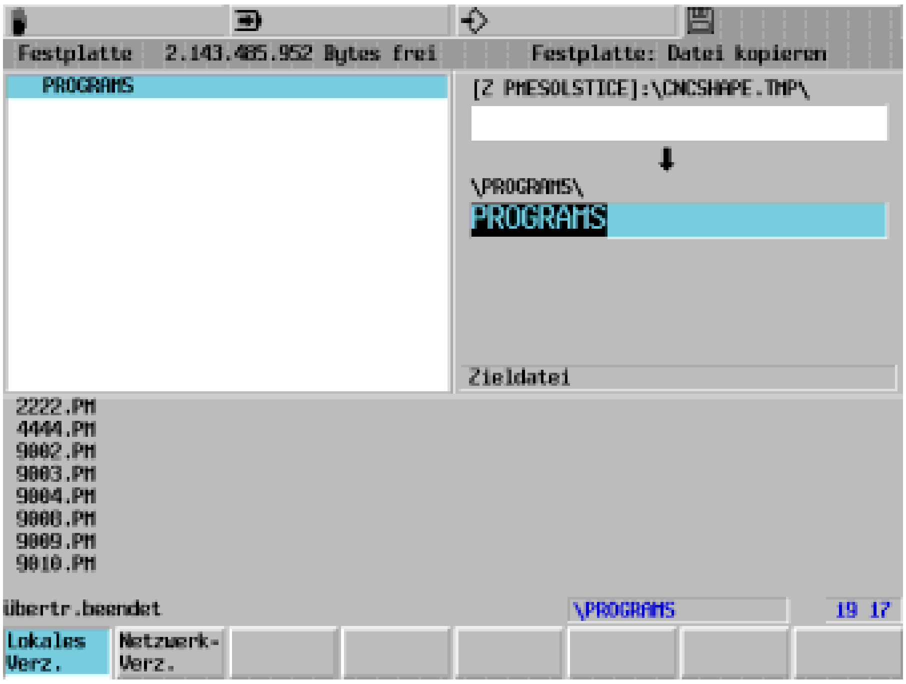
Select directory



Select directory



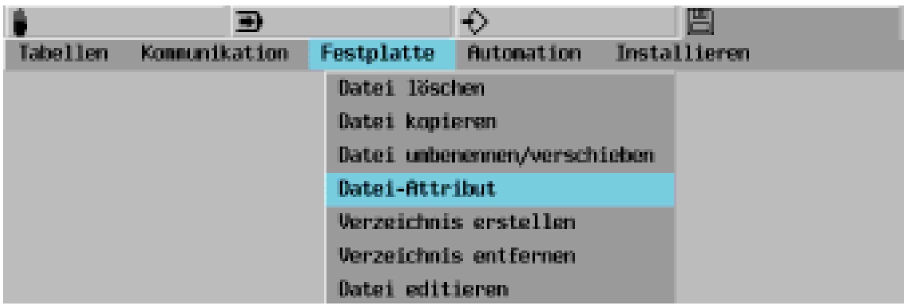
Open directory

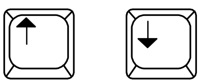


9.8.3 Rename/shift file

Rename/shift file copy analog file operation.

9.8.4 Attribute file (lock/unlock)





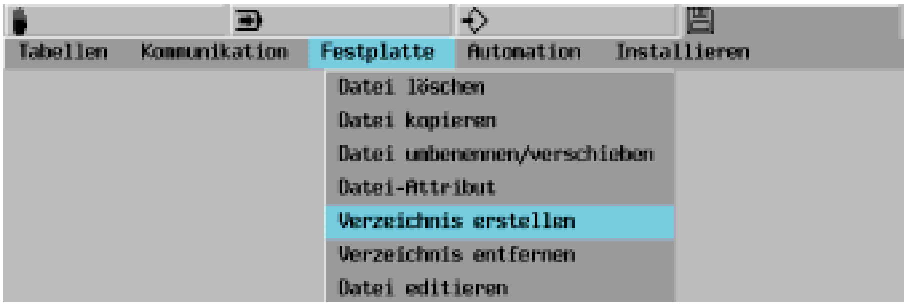
Select program

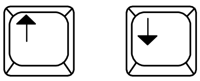
Quellverz wechseln | Sichern/ Freigeben

Sichern/ Freigeben

9.8.5 Make directory

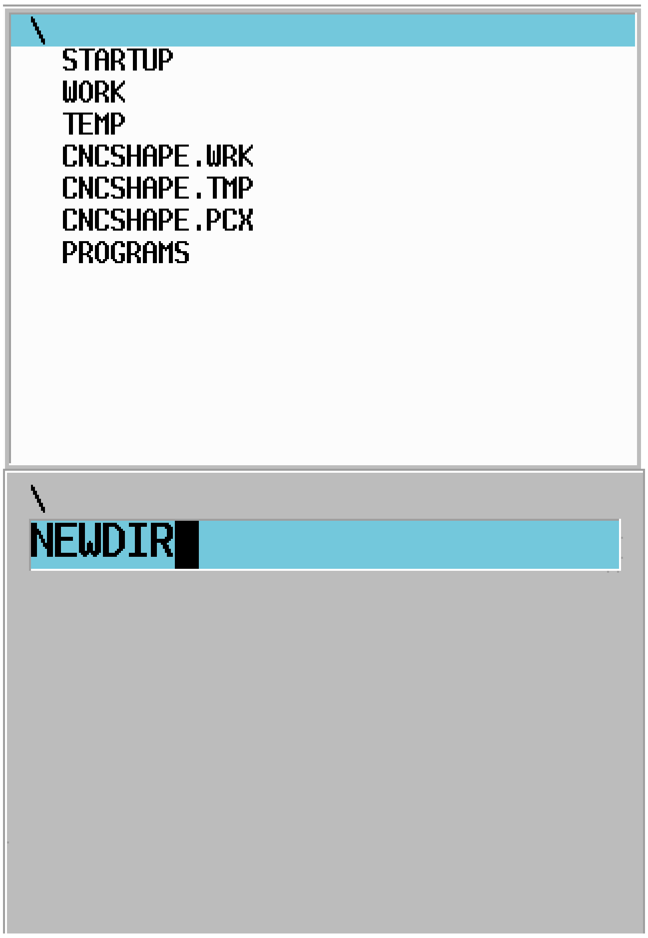
This enables you to create a new directory. The name of the directory consists of a maximum of 11 characters (DOS format 8.3 characters). The directory can have up to 5 levels.

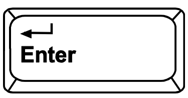


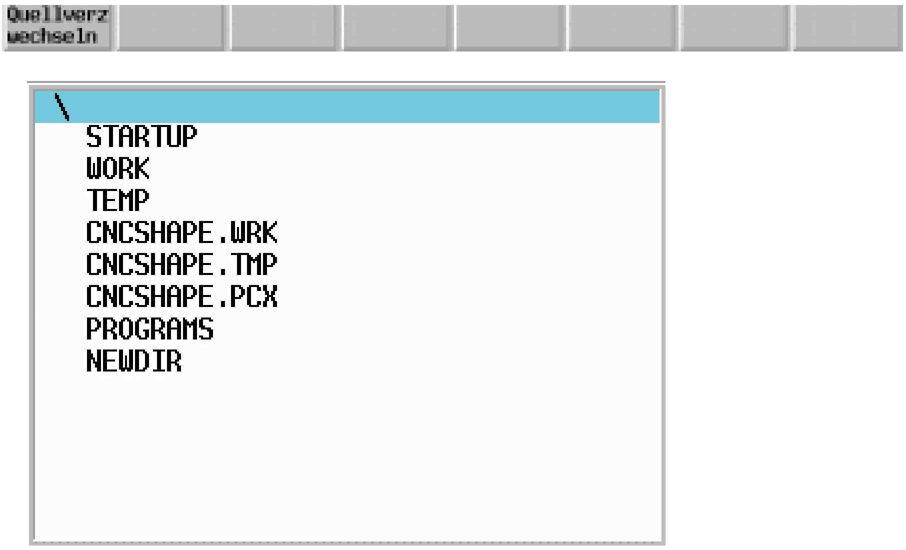


Select directory

Enter directory name (NEWDIR)

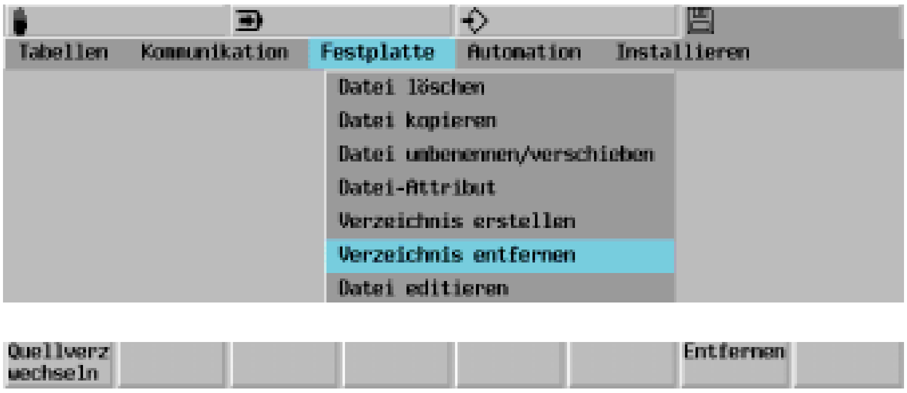


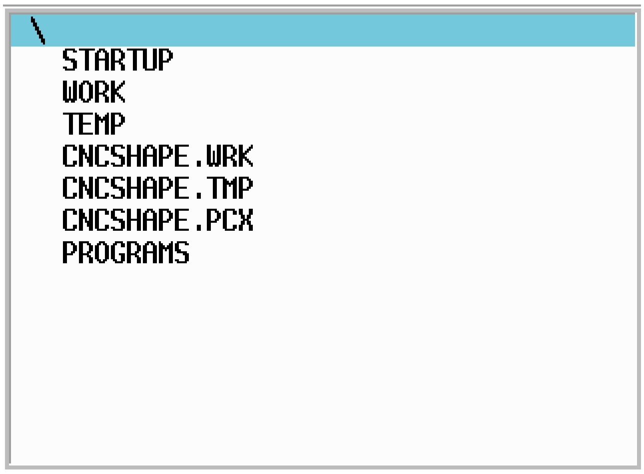


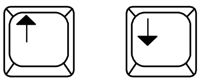

Quellverz
wechseln
\
STARTUP
WORK
TEMP
CNCSHAPE.WRK
CNCSHAPE.TMP
CNCSHAPE.PCX
PROGRAMS
NEWDIR

9.8.6 Remove directory

The directory must be empty. The current directory cannot be removed.

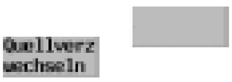






Select directory

Remove directory



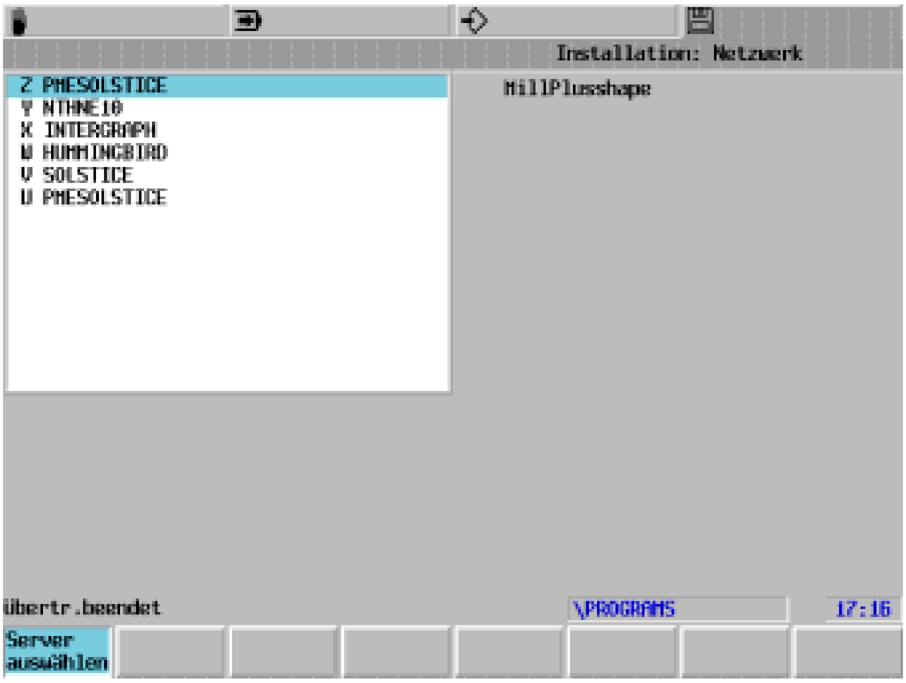
Remove directory No or Yes?

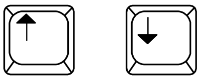
9.9 Ethernet interface

Additional disk drives become available if MillPlus is connected to a network. The Copy File function is the only one that can also be used on network drives.
For details on how to set up the interface, refer to the chapter entitled Miscellaneous.

9.9.1 Select server

The server is the network node that is used to transfer data. Only one server can ever be active at a time.
The configuration file contains a list of possible servers. The server that is selected must be an active server.





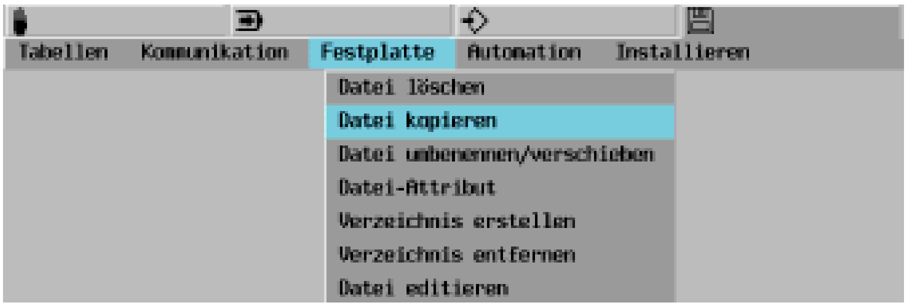
Select server

Make server active

Note

Ethernet provides no way of preventing two clients from accessing the same file on the server at the same time. This may result in corruption of one of the files.

9.9.2 Write to server

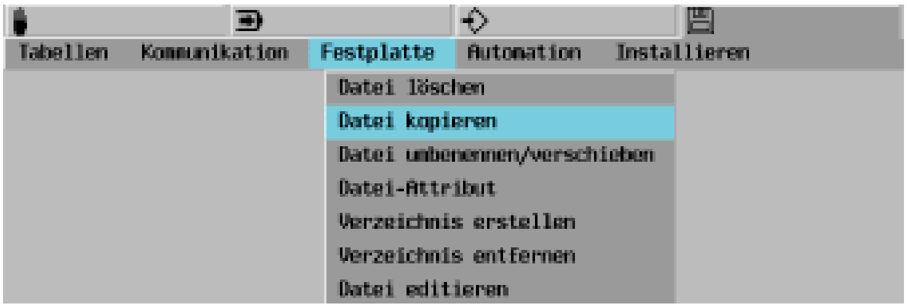


Send the files from the current directory on the CNC hard disk to the specified directory on the server.

-Select source directory on CNC
-Select target directory on server
-Select or enter file name

Write file to server

9.9.3 Read from server



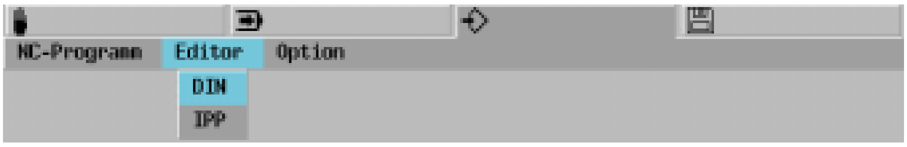
Copy the files from the server to the current directory on the CNC hard disk.

-Select source directory on server
-Select target directory on CNC
-Select or enter file name

Read file from server

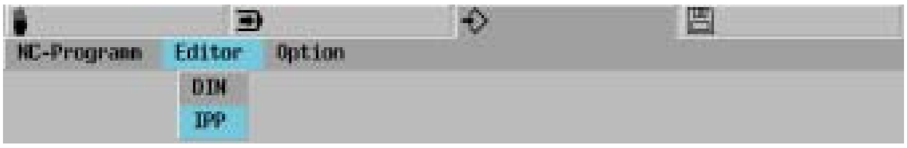
10. Enter / edit program

10.1 DIN/ISO Editor



To edit DIN/ISO programs.

10.2 IPP Editor



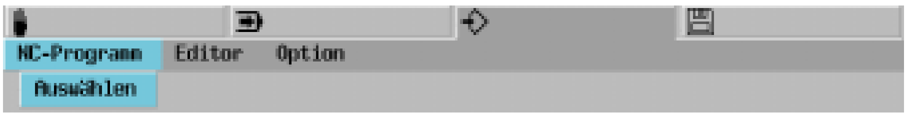
To edit IPP programs.

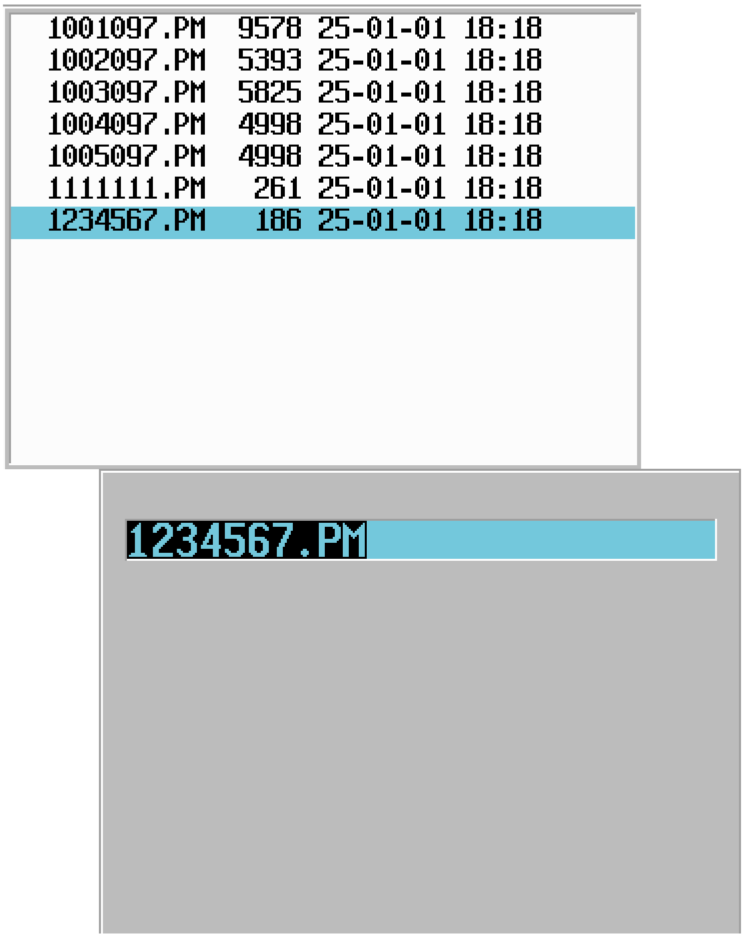
10.3 Input help

The following are available:

Interactive parts programming (IPP)
Interactive contour programming (ICP)
Support for G-Functions

10.4 Enter new program number (main program / macro)





Select program type.

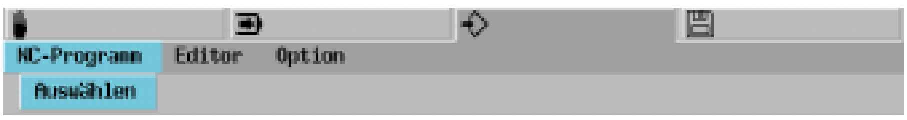
Enter program number (1-999 999 9)
Example: 777777

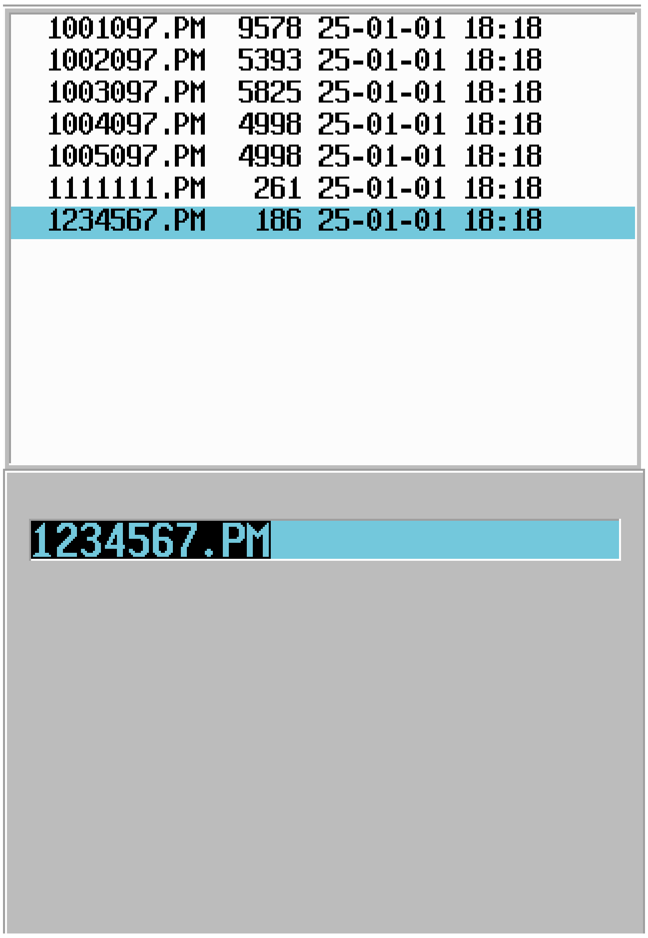
Start the active editor with the new program number.

Note

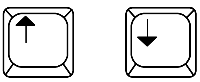
Main programs (invoke with G23) and subprograms (invoke with G22) should be in the directory of the active main program.

10.5 Program selection (main program / macro)

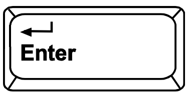




Softkey nur für die Anzeige der Makroliste drücken.



Programm anwählenz. B.1234567.PM. Bei der Eingabe der Programmnummer, braucht man die Erweiterung .PM oder .MM nicht eingeben.



Activate the program for editing.

Storage request after editing and selection of edited NC program via the menu.

Changes in the active main program and in the accessory subprograms are stored automatically.

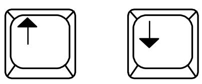
10.6 Save to hard disk

Save program to hard disk.

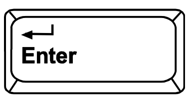
10.7 Enter program block

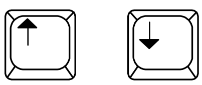
Directly at cursor position using ASCII keyboard

10.8 Insert program block



Select sentence number after which a sentence is to be added.





Edit block and close.

10.9 Text entry

Text in brackets after parameters, maximum length 124 characters.

Example:
G1 X50 Y83 M13 (turn on coolant)

10.10 Mathematical entry

The functions sin(..) cos(..) tan(..) asin(..) acos(..) atan(..) sqrt(..) abs(..) int(..) may only be written in lower case.

Spaces are not permitted in functions.

Maximum size of on one line: 248 Characters.

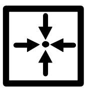
10.11 Position transfer in the program (DIN editor)

Selects the axes which should be transferred

Transfers the current position of the selected axes into the program

to the DIN editor

Transfer position using HR410.
Select the axes which should be transferred.



Transfers the current position of the selected axes into the program at the cursor point. Afterwards an <Enter> is automatically executed.

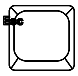
The position can also be transferred while the machine is in motion.

Note

If G0 X100 is written in the line and position X121 Y122 is transferred, the final line entry is G0 X100 X121 Y122. Subsequently the programmer must delete one of the two X addresses.

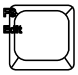
10.12 Delete address

Deletes character to left of cursor.

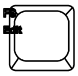


Undelete the most recently deleted addresses in a block.

10.13 Editing function



Activate the EDIT softkeys.

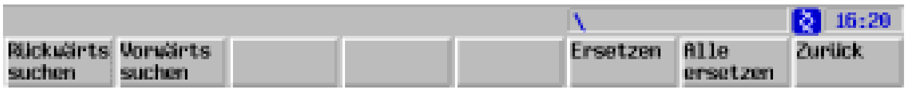


Quit the EDIT function.

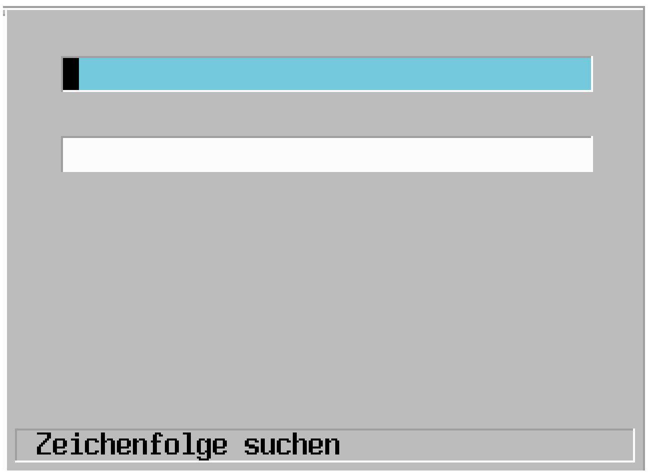
10.13.1 Erase line

This enables you to erase the active line (indicated by the cursor)

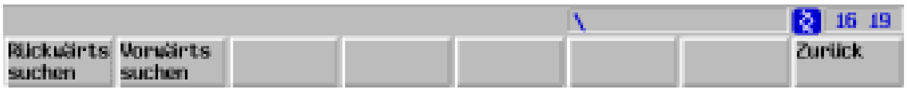
10.13.2 Search & replace



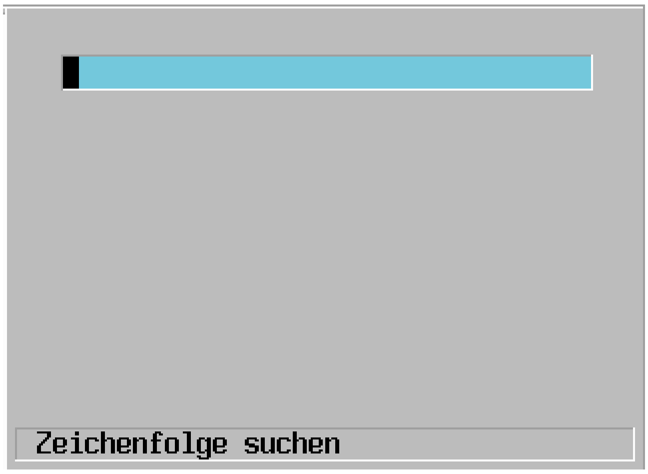
Enter character string



10.13.3 Find



Enter character string



Note

The new numbering begins with the sentence number of the first (marked) sentence.

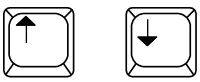
10.13.4 Renumber

Line number in the program blocks are renumbered.

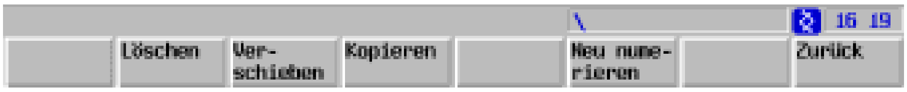
Note

The new numbering begins with the sentence number of the first (marked) sentence.

10.13.5 Block (delete, renumber)



Mark a program record/block.

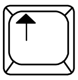
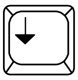


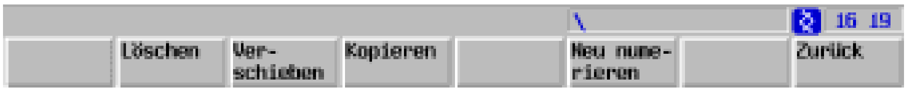
Perform operation

Note

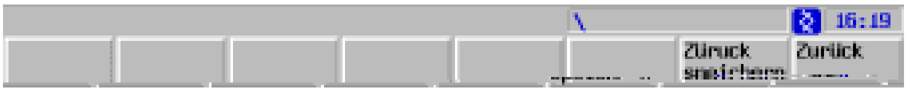
The new numbering starts with the block number of the first highlighted block.

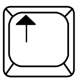
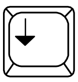
10.13.6 Block (Move, Copy)



Mark a program record/block



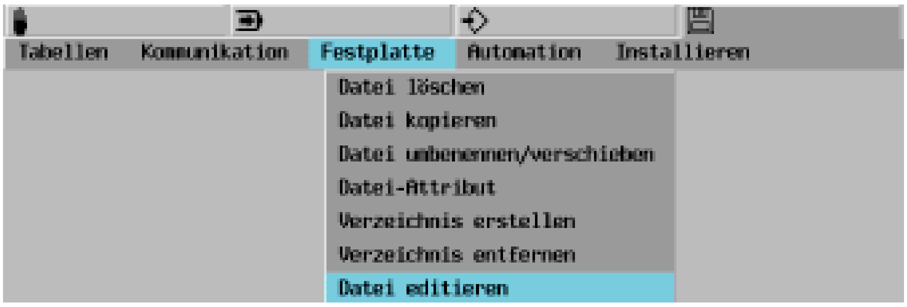
Save program record/block in intermediate storage





Select record number

Save program record/block in program

10.14 File editor



Enter program number, example: 4444.pm

Changes take effect immediately.
The file editor does not check blocks as they are entered and saving. Test the program using the graphic test run function.
The graphic test, support, ICP and technology functions are not supported by the file editor.

Features:

For editing programs larger than 1 Mbyte
Blocks are not checked as they are entered and saved
Editing of active programs not possible
No NC language support while editing

10.14.1 Undo

Up to 100 operations can be undone.

The following operations cannot be undone:
-Select, Delete, Move, Copy Block
-Write Block/Insert File
-Search & Replace

10.14.2 Go to line number

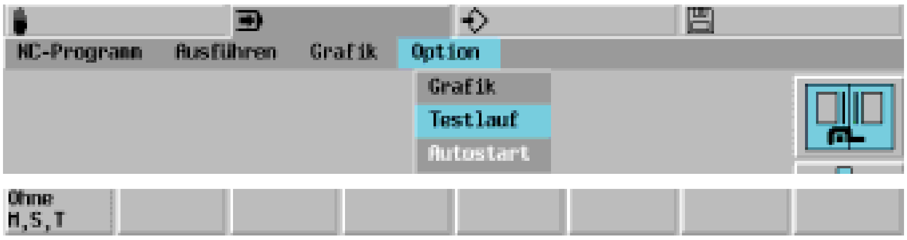
Note:
The line number refers to the line number in the file, not the record number N in a program.

11. Program dry run

11.1 Dry run mode

The test run takes place at an increased feed rate (MC 741).
Activate the program.

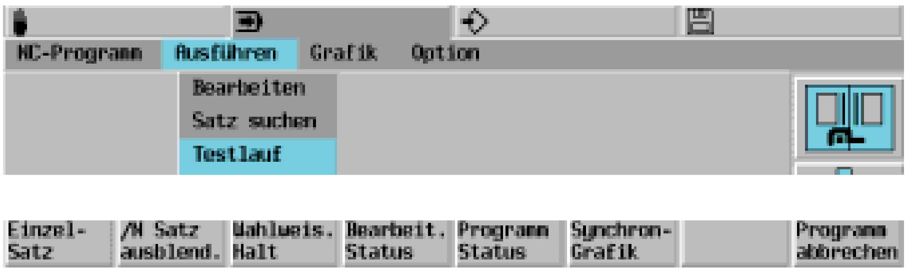
11.1.1 Select dry run option

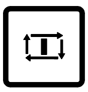


M,S and T not output

Note: Lock axis
MC 100 C3 (1st axis)
MC 105 C3 (2nd axis)
MC 110 C3 (3rd axis)
MC 115 C3 (4th axis)

11.1.2 Perform dry run



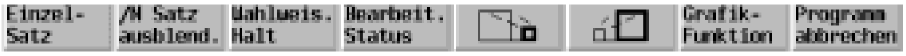


Start dry run

11.2 Graphics dry run

Activate the program.

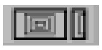
11.2.1 Graphic functions



Select 2/2.5/3D view

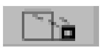
e.g. 3D view

11.2.2 Graphical representation



Graphical representation

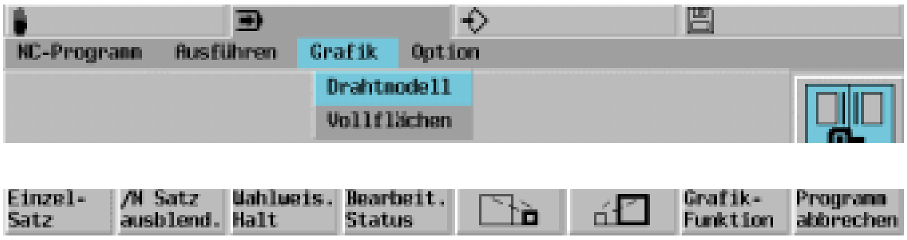
Enlarge drawing step by step

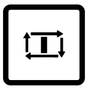


Reduce drawing step by step

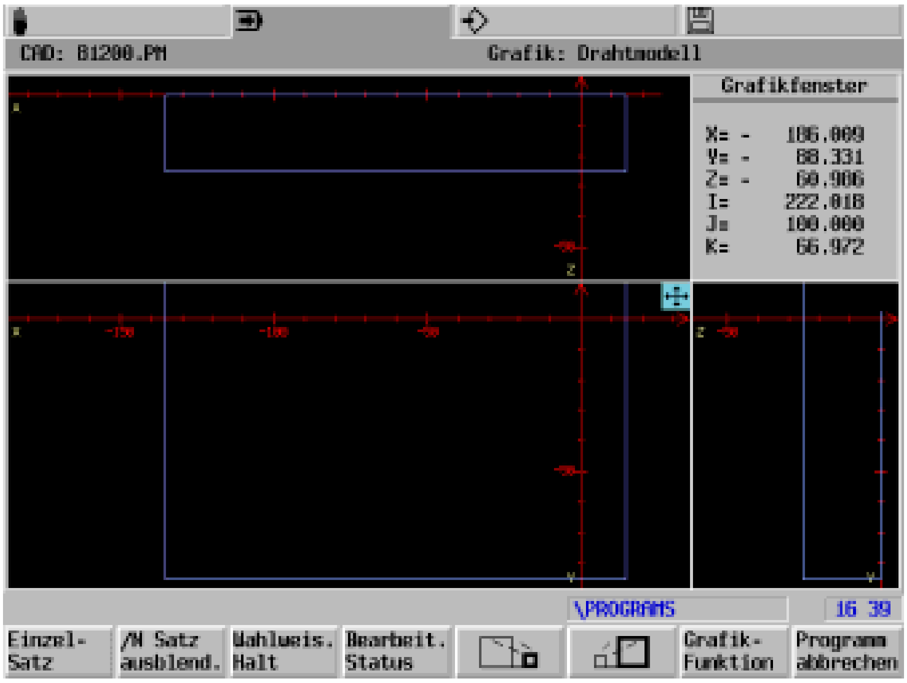
11.2.3 Graphic options

11.2.4 Wire plot





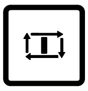
Start wire plot



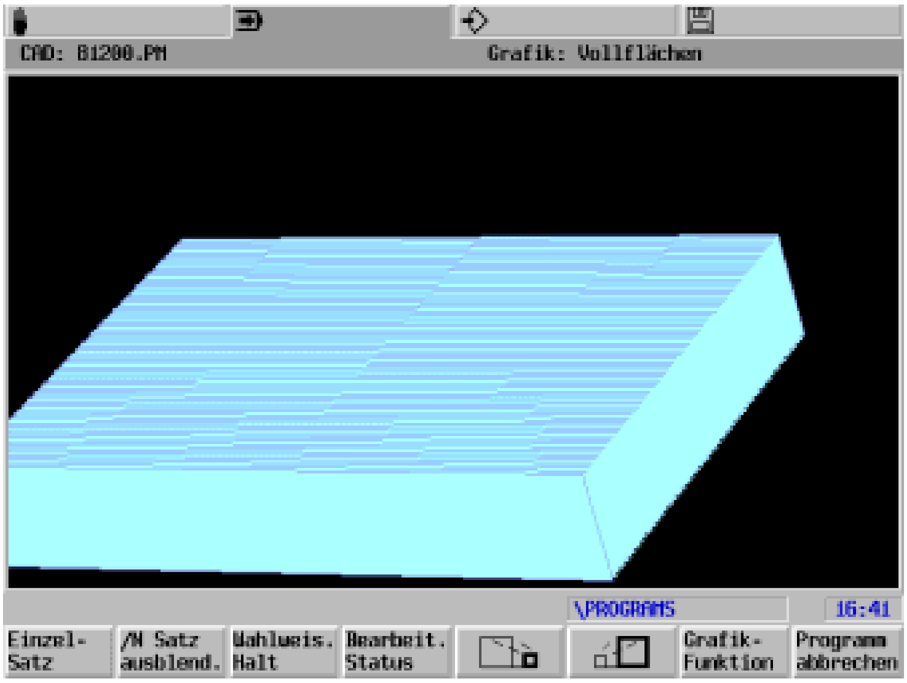
11.2.5 Working with graphics (example)

- Activate the program.
- Select the Graphics option.
- Select Wire or solid graphic.
- Start the program.

11.2.6 Solids



Start graphics solid



11.3 Estimation of run time using graphics operation

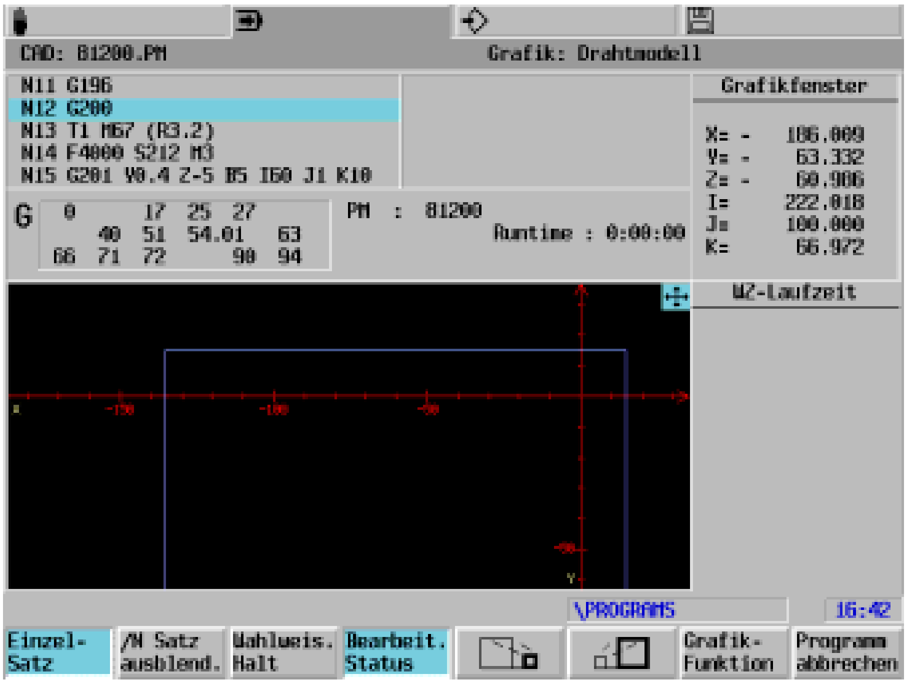
During graphics operation the graphics run time is displayed in the operating status.
The run time is calculated from the tool length and the feed rate (correction = 100%). 10% is added to this calculated time to allow for braking and accelerating in the corners. During programmes at high rates of advance the estimated run time is less than the actual run time, because the machine cannot track the program.

Note

Time taken by the M functions is not taken into account in the estimation.

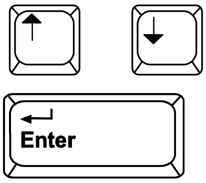
11.3.1 Time for each tool

The estimation of operating time is also calculated for each tool. In the course of this, only the time that elapses with the set rate of advance is taken into account.



12. Activate / execute program

12.1 Activate program



Position the cursor on the desired program or enter program number.

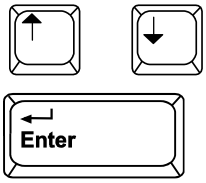
Operating mode "Execution: Machining" is automatically activated.

12.2 Direct activation of an edited program

Editing a program

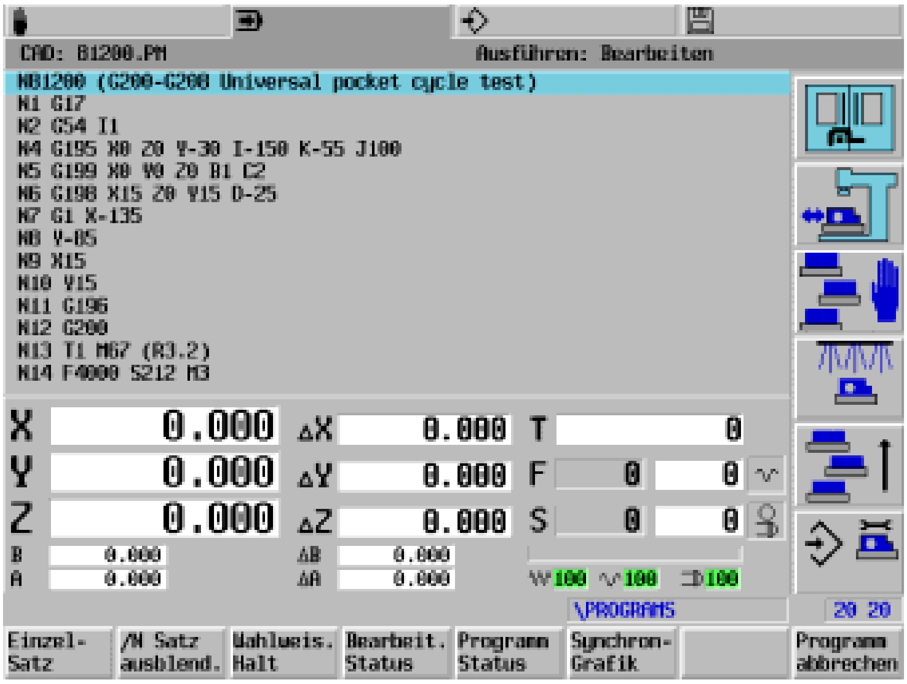
12.3 CAD mode

The "CAD mode" function enables you to process programs that require more memory than is available in the CNC-RAM. The size of BTR memory is defined in MC93. (Example 128kbyte).



Position the cursor on the desired program or enter program number.

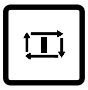
Operating mode "Execution: Machining" is automatically activated.



Note:

The main programs must not contain G23, G14, G29 functions or E0 parameters.
"Satz suchen" backwards is not possible.

12.4 Execute program



12.5 Single block operation

12.6 Delete block

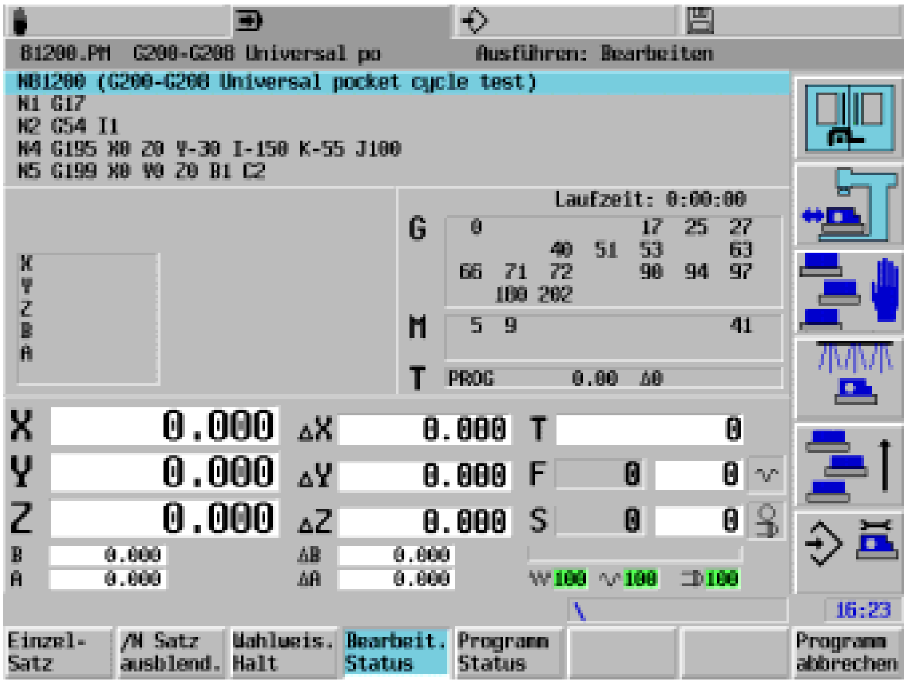
Note:

The program block must start with a '/', e.g.: /N5 G1 X100

12.7 Optional halt

Halt following execution of M1.

12.8 Execution status



The overlap depth is indicated in the operating status behind MM:

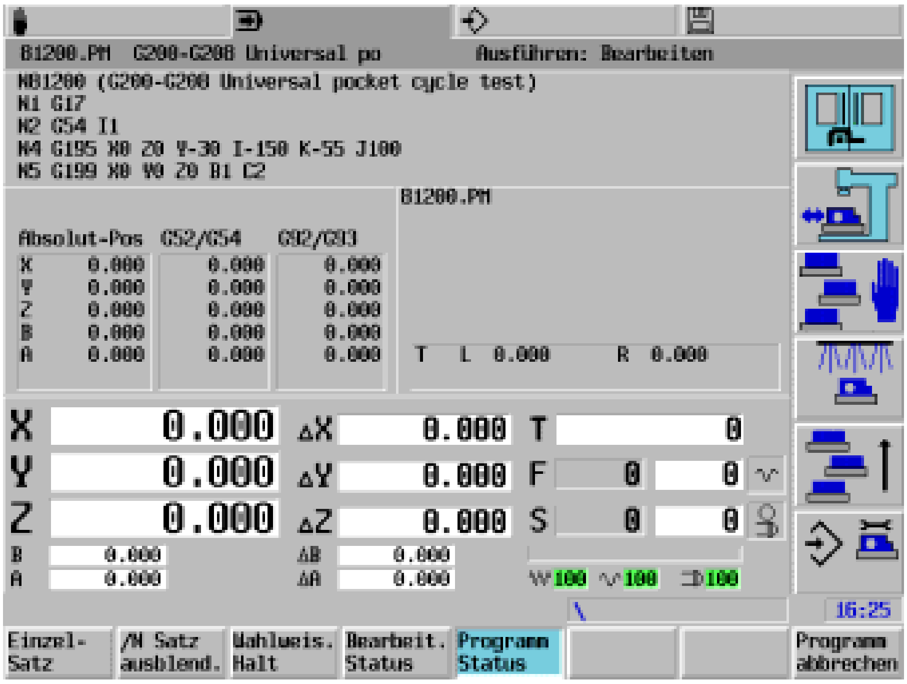
Notes

- During BTR and CAD operation the overlap depth is not calculated by the BTR macro.
- The first overlap or repeat depth is "1" and is not displayed.

Im Bearbeitungsstatus wird die Schachtelungstiefe hinter MM angegeben:

12.9 Program status

The following elements are displayed concurrently:
- current tool length (L+L4=) and tool radius (R+R4=).
- current tool oversize G39 L and R
- the position with reference to the machine null point
- the current G52, G54 (Inn or G54-G59) null point displacement
- the current G92 and/or G93 null point displacement
- the complete "overlap tree" of the main programs, the macros and the repetitions



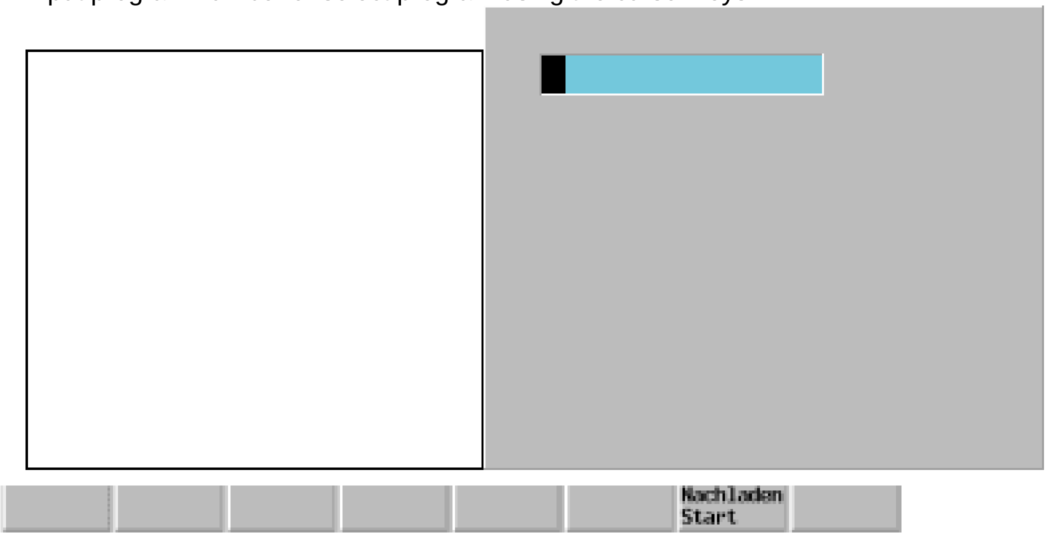
Notes

- The overlap tree can hold a maximum of two main programs, eight secondary programs and four repetitions. It "scrolls" automatically in the window as necessary.
- During repetition only the number "still to run" is displayed in Repetitions.
- The command <Program status> cannot be selected during graphics operation.
- Jumps in the program are not displayed in the overlap tree.

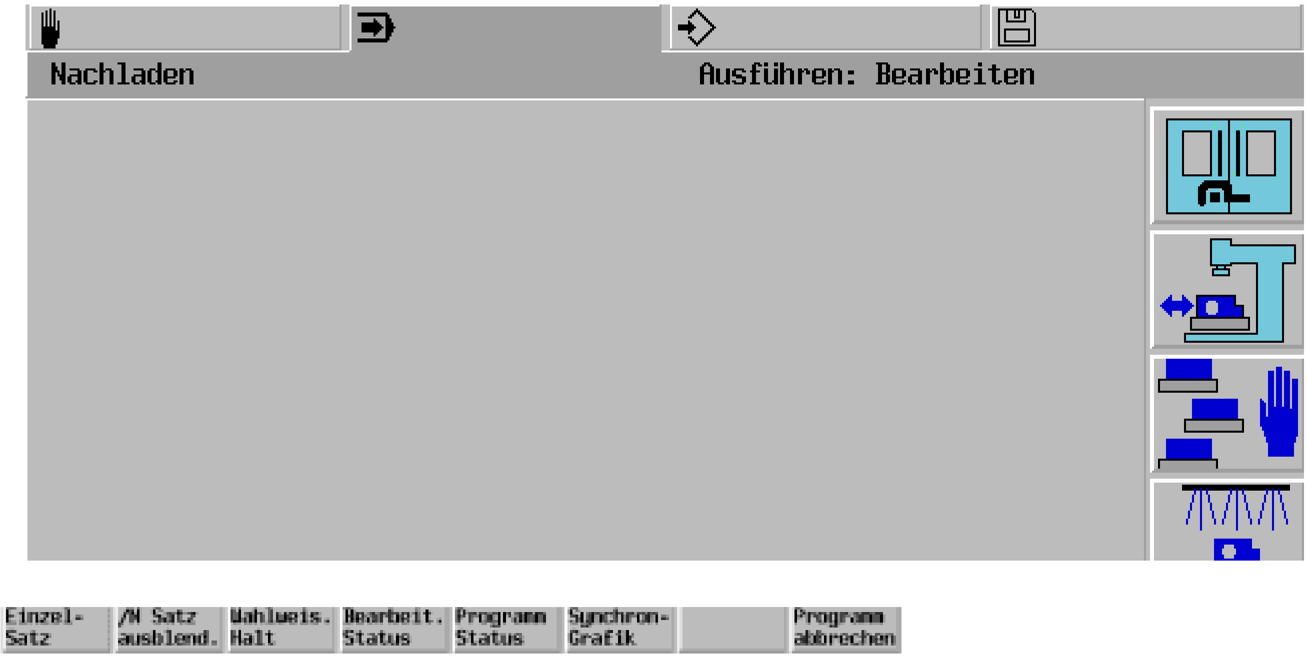
12.10 Reload (BTR)

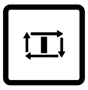
The Reloading function is used to execute programs that need a larger storage volume than the CNC working memory directly from external devices. The size of BTR memory is defined in MC93. Programs from external equipment may be executed by reloading.
Provide data transfer peripheral. (external device with DNC link)

Input program number or select program using the cursor keys.



From external equipment





The program will be executed.

Note:

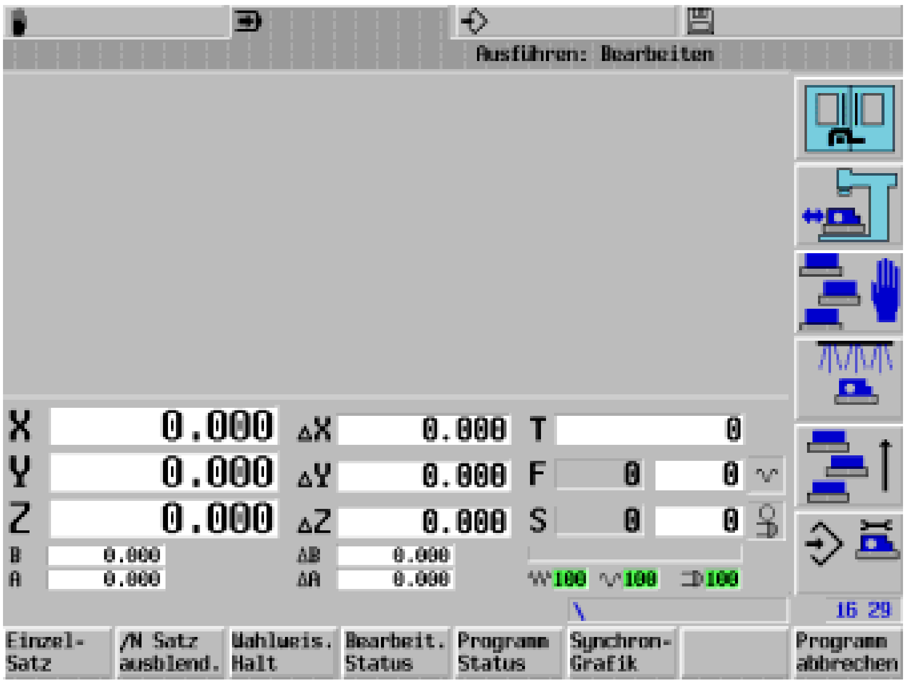
Main programs may not contain any G23,G14,G29 functions or the E0 parameter. A "Search block" is not possible.

12.11 Autostart

The machine should be at operating temperature before machining the first workpiece each morning. The machine is run up to operating temperature by starting a running in program that, for example, lets the spindle rotate for a while. This running in program should be started automatically some time before starting work.

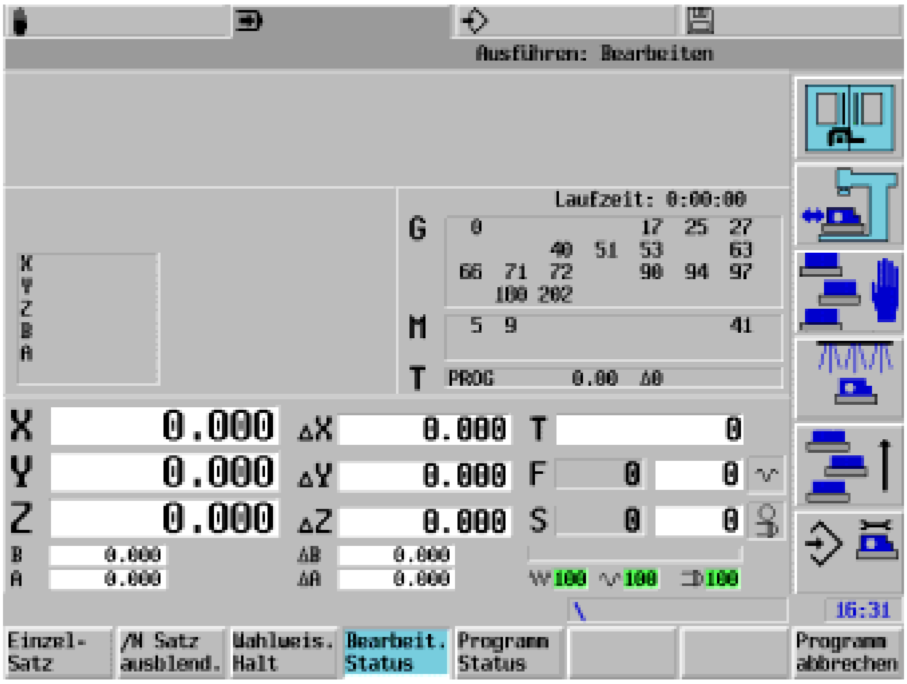
WARNUNG! The operator is responsible for ensuring that the machine is in the correct operating mode when the <Autostart> is issued. It can happen, for example, that the operator is running a program in single block mode at the same time that the Autostart issues a <Start>. In such a case the active block will be 'unexpectedly' executed.

12.11.1 Setting up Autostart

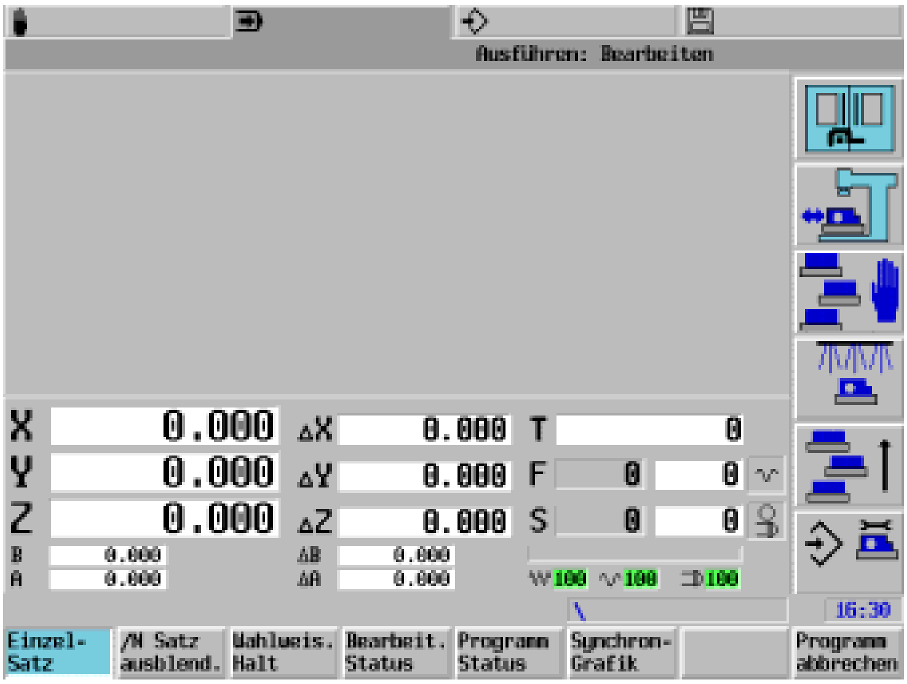


Validates and saves the entered values

Input fields on this page are to be deleted



12.11.2 Activate Autostart



Note:

The CNC and machine tool must be left in the correct operating mode.
If no program is entered, the active program is started.
Autostart active is indicated by a yellow background to the timers.

12.12 Digitizing

Synchronise controller and peripheral device.

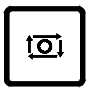
Note

Refer to Renishaw Trace documentation and the chapter Miscellaneous for further details.

13. Interrupt/cancel program, search block

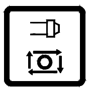
13.1 Interrupt/cancel program execution

Program execution may be aborted at any time during machining and in single block mode.



Feed stop

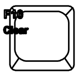
or



Feed and spindle stop

"Interrupt program" enables programmed feed movement, using the axis movement keys.

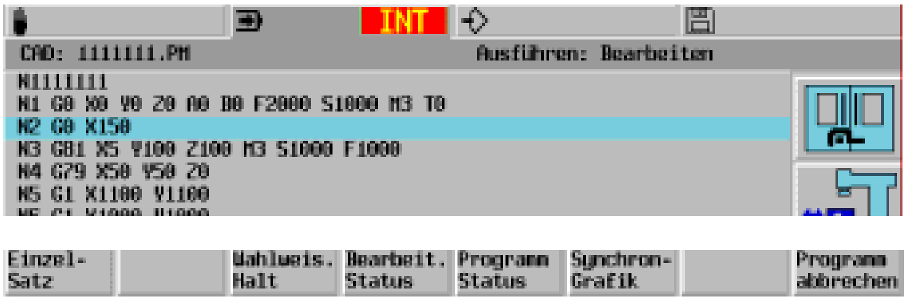
13.2 Erase errors and messages on the screen



Erase errors and messages on the screen. The program is not cancelled.

13.3 Cancel program

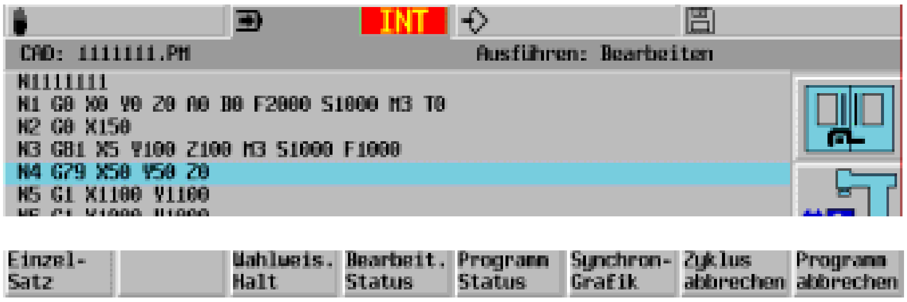
Interrupt program execution

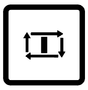


Return to start of program. Only the offset of the current tool, the machining level and the zero offsets remain active. Activated errors and messages will be deleted.

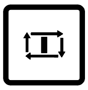
13.4 Interrupt cycle

Interrupt the cycle program run.





Cancel cycle and movement to starting point.

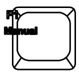


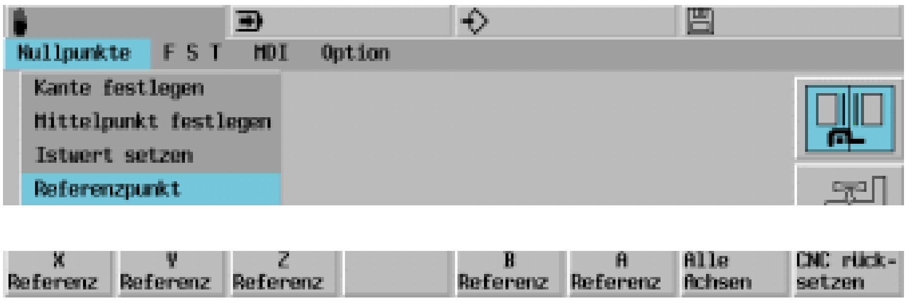
Continue the program from the next block.

13.5 Reset CNC

Reset all functions (predefined values still apply) and clear all modal parameters.

Cancel program.



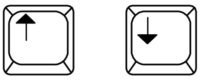


13.6 Search block

Find block (e.g. program resumed after program interruption)

Entry of block number

or



select block

Return to program

Note

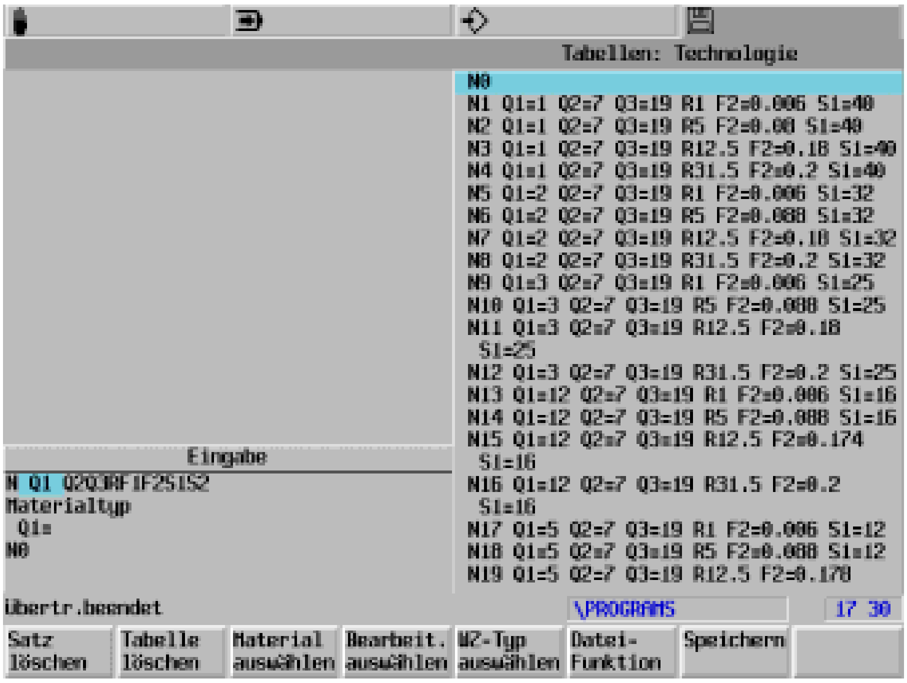
Search for block in repetition part (G14) or subprogram (G22):
- search for program block G14 or G22
- execute G14 or G22 block (single block)
- search for block in repetition part or subprogram

Search in macros:
It is only possible to search for blocks, not for characters.

14. Technology

Establishing the cutting values in a practice-oriented fashion is a most comprehensive task because of the various tools, materials to be cut, coatings, cutting geometries, range of applications, workpiece materials etc.
The feed and speed values suggested by the cutting value calculator may therefore not suit all conditions. Where appropriate, the user should optimise these values.
The cutting values recommended by the tool manufacturer may be useful.

14.1 Technology table



Q1= Material code, which is taken from the file for the material texts.

Q2= Machining process code, which is taken from the file for the machining texts

Q3= Tool type code, which is taken from the file for the tool type texts.

R Tool radius (in mm). If R = 0 is entered, you will be asked to enter the workpiece radius in case the feed rate or spindle speed has to be calculated in a unit of measurement other than that specified in the technology table (the programmed data is specified in rpm, for example, while in the technology table it is given in m/min).

F1 Feed rate in mm/rev. The feed rate for the combination of material, machining process, tool type and tool radius given in the other parameters must be taken from special tables or calculated.

F2 Feed rate per tooth in mm/rev. Refers to tool types with more than one cutting surface. The feed rate for the combination of material, machining process, tool type and tool radius given in the other parameters must be taken from special tables or calculated.

S1 Cutting rate in m/min. This value should be taken from the appropriate documents of the tool manufacturer or empirical values should be used.

S2 Spindle speed in rpm. This value should be taken from the appropriate documents of the tool manufacturer or empirical values should be used.

14.1.1 Tools with more than one radius

In situations where identical tools can have different radii, it is not necessary to enter values in the table for each tool. If the combination of material, machining process and tool type stays the same, only two values need to be entered in the table, one for the smallest tool raadius and one for the largest. The system then uses these two values to interpolate the feed rate and speed and puts forward suggestions for F1 and S1.

14.1.2 Table values for tapping

In some cases, interpolation of the values in the table is not desirable or is not possible, e.g. when tapping. In such situations the feed rate (F1) must be identical to the thread pitch. Interpolation is not possible in such cases.

14.1.3 Relationship between F1 and F2

Both F1 and F2 are used to specify the feed rate. Generally, F1 is used to define the feed rates used in tapping or for drilling using a milling machine. Milling machines usually have more cutting surfaces (teeth). F2 is normally used to specify the feed rate for milling work.

F1 = F2 x number of cutting surfaces

14.1.4 Relationship between S1 and S2

S1 is specified in meters/minute. S2 is displayed in rpm.

S1 = (S2 x 2 x π x R) / 1000

R is the tool radius.

Note

A value is assigned either to parameter F1 or F2 but not to both. The same applies to parameters S1 and S2.

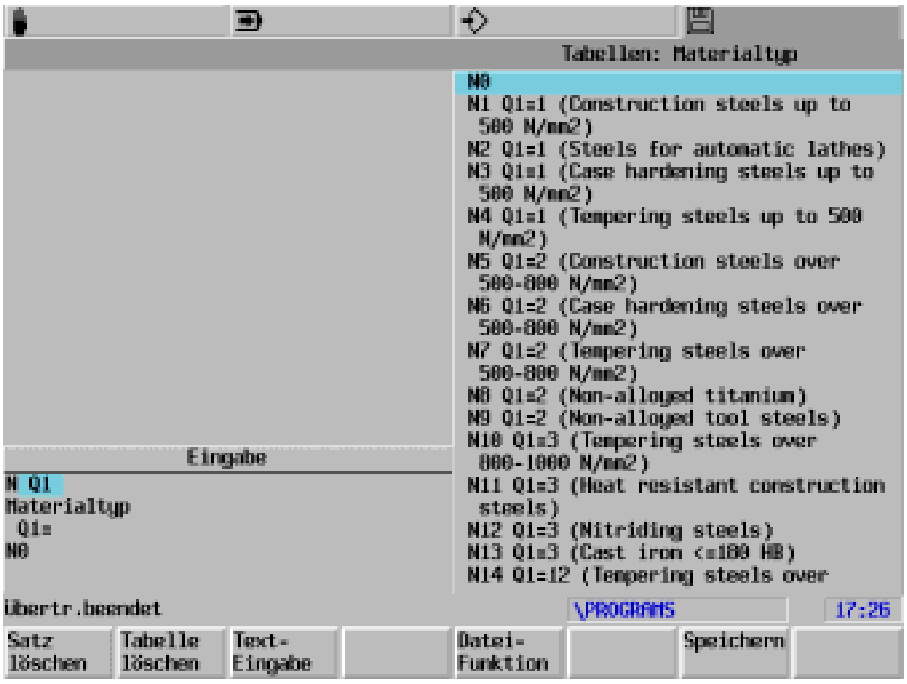
14.2 Storing the technology tables

Storing technology tables on hard disk.

Storing technology tables in CNC_RAM.

14.3 Material type

Defining the materials to be machined.



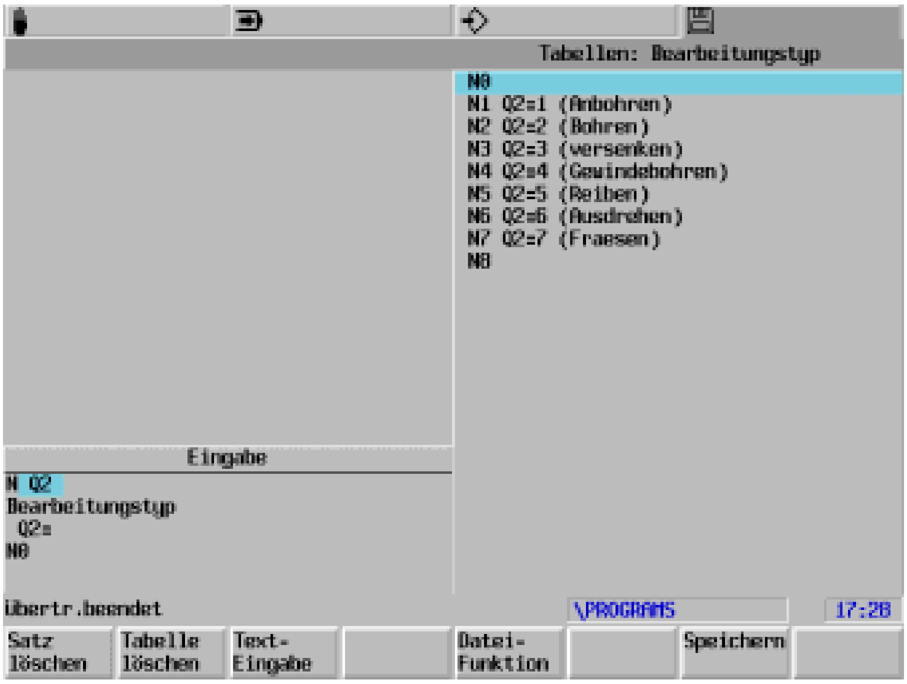
Q1= Material code

Materials having the same machining properties may be assigned the same material codes.

The texts on the material must be in brackets.

14.4 Machining type

Defining the machining process.

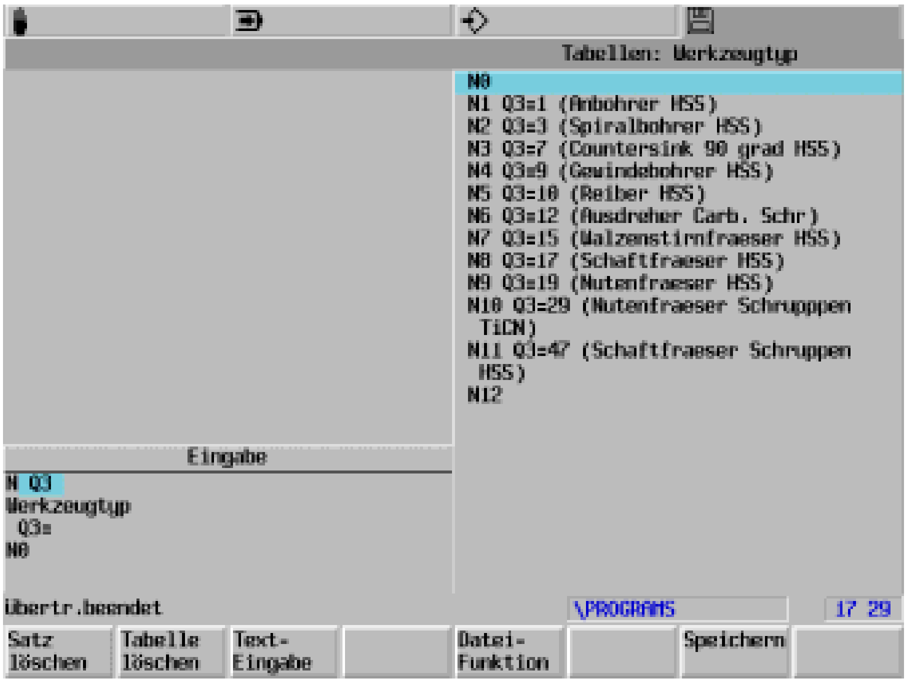


Q2= Machining operation

The texts on the material must be in brackets.

14.5 Tool type

Defining the tools.



Q3= Tool type

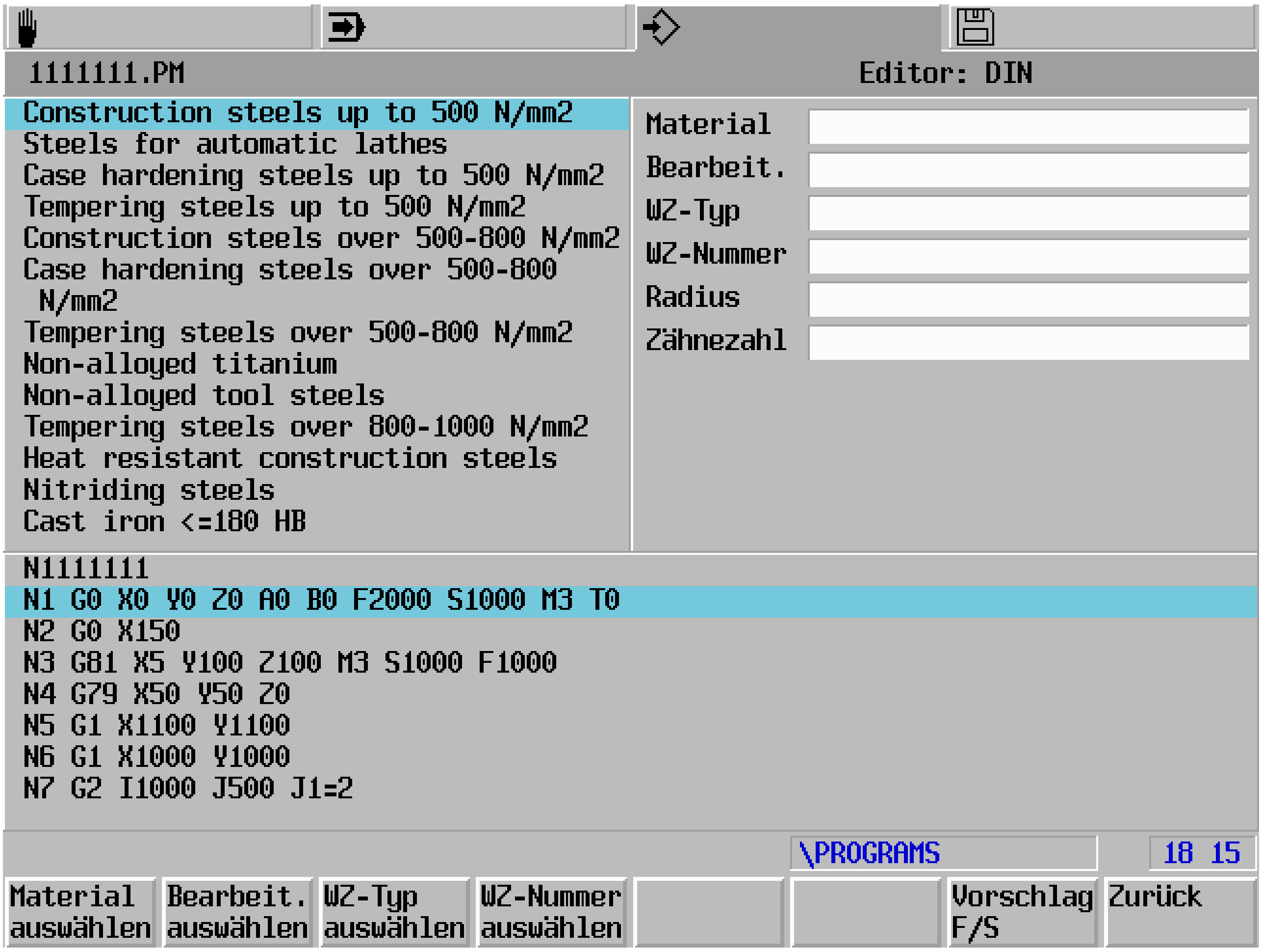
The texts on the material must be in brackets.

14.6 Using the technology

Select program process level and program

The proposed feed rate and spindle speed can be generated using the following key sequence:

Select the desired material.



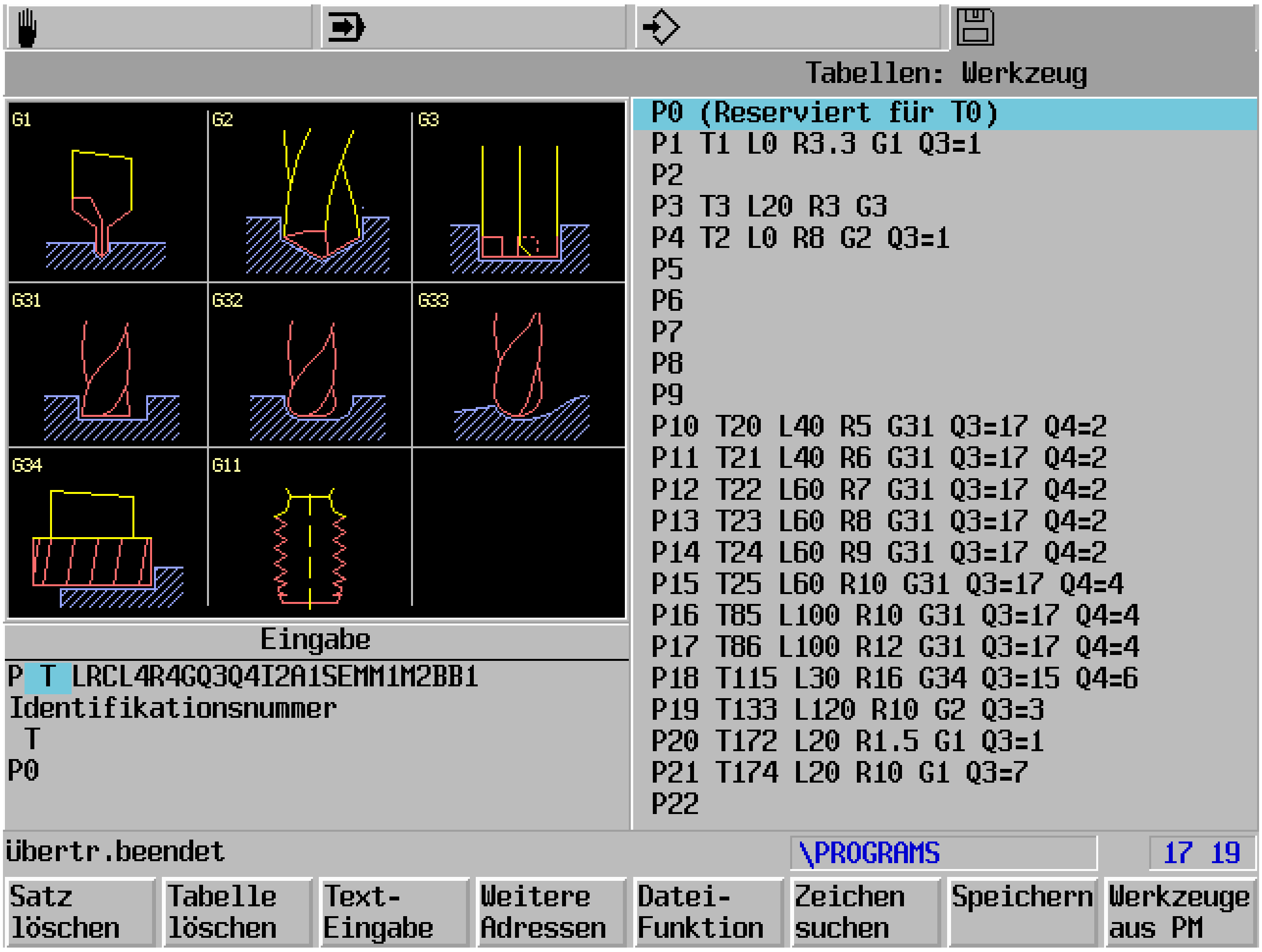
Select the desired machining operation.

Select the type of tool.

Select the desired tool identification number.

The propsed F, S and T values are transferred to the selected program block.

15. Tool



Tool used in current program

Plaintext input in table. Enter text in brackets.

File function.

15.1 Tool addresses

P Magazine pocket. Location of the tool in the tool magazine (if present). Location P0 is reserved for the new tool and cannot be used to store tool parameters. Pocket 1 is indicated by P1, pocket 2 by P2, etc. The actual number of tool pockets in the magazine is saved as a machine constant.

T Identification number, e.g. T 12345678.00
L Length
R Radius
C Corner radius
L4= Length allowance
R4= Radius allowance
L and/or R are adjusted when measuring. L4= and/or R4= are set to zero.
L and R are not adjusted when checking. Only L4= and/or R4= will be modified.
G Graphics. Define the tool shape in graphics mode.
Q3 Type. The numbers to identify the tool type are entered in this parameter.
Measuring probe Q3=9999: no spindle rotation possible and rapid motion (MC) restricted.
Q4 Number of cutting tips

I2= Cutting direction
3 clockwise M3
4 anti-clockwise M4

A1 Heel angle (0.1-15 degrees)
S Size (0=normal, 1=oversize). The maximum tool dimensions and diameter above which a tool is classified as oversize are specified in the supplied machine tool manual. The control keeps a magazine pocket in front of and behind an oversize tool free.
E Status. The normal setting is E0 (tool enabled, not measured). When the specified tool life is exceeded, E-1 is set automatically. When the tool has been enabled or measured, E1 is set.
E-2,-3,-4 tool disabled (new as of V321).
The machine tool builder may define other negative status values. Refer to your machine tool manual.
M Tool life (mins.)
M1 Current tool life (mins.)
M2 Tool life monitoring (0 = off, 1 = on)
B Break tolerance (0 = MC value) (maximum 255)
B1 Break monitoring (0 = off, 1 = on)

Next address selector

P Magazine pocket. Location of the tool in the tool magazine (if present). Location P0 is reserved for the new tool and cannot be used to store tool parameters. Pocket 1 is indicated by P1, pocket 2 by P2, etc. The actual number of tool pockets in the magazine is saved as a machine constant.

T Identification number, e.g. T 12345678.00
L Length
R Radius

L1 First additional length
R1 First additional radius
C1 First additional corner radius
L2 Second additional length
R2 Second additional radius
C2 Second additional corner radius
Q5 Break monitoring cycle (0-9999)

S	Size (0=normal, 1=oversize). The maximum tool dimensions and diameter above which a tool is classified as oversize are specified in the supplied machine tool manual. The control keeps a magazine pocket in front of and behind an oversize tool free.
L5=	Wear tolerance length (mm)
R5=	Wear tolerance radius (mm) A fault is signalled if the deviation is greater than the values specified here.
L6=	Offset length (mm) Displacement (>=0) of measuring position compared to tool tip.
R6=	Offset radius (mm) Displacement (>=0) of measuring position compared to centre of tool.

15.2 Tool identification

The tool identification number may contain up to eight digits for the tool number plus 2 decades (00) for identifying the tool (original tool or replacement tool). The decade entry may be omitted for the original tool. If a replacement tool is entered for a tool, e.g. T1, this is indicated by the information in the decades (e.g. T1.01, T1.02 etc, i.e. these tools are replacements for T1).

15.3 Calling tool data

The T address and M-function are used to call a tool in the machining program.

Example of calling a tool:	
Tool number T.. [Format 8.2] (255 tools max.)	N.. T1 M..
Original tool (T1-T99999999)	N.. T1
Replacement tool (Tx.01-Tx.99)	N.. T1.01
Activation:	
Automatic tool change	N.. T.. M6
Manual tool change	N.. T.. M66
Activate tool data	N.. T.. M67
First additional tool offset	N.. T.. T2=1 M6/M66/M67
Second additional tool offset	N.. T.. T2=2 M6/M66/M67
Tool life T3=..[0-9999,9min]	N.. T.. T3=x M6/M66
Cutting force control T1=..[1..99]	N.. T.. T1=x M6/M66
Deactivate (T1=0 or T1= not programmed)	N.. T1=0

Modal parameters T, T1=, T2=

Tool preselection in the machining program:
The next tool to be used is preselected by programming the tool number T without a tool change command.

15.4 Input of the tool memory

Options during input of the tool memory. The options are changed by means of MC774:

0 The input addresses are loaded or overwrite the existing addresses.
1 First, the tool memory is cleared. Subsequently, the new addresses are loaded.
2 The existing tools are not changed and, during input, are missed without a fault report.
3 Tool without P overwrites the tool currently existing.

The input addresses are loaded or overwrite the existing addresses.

MC774 = 0	Bestehendes TM	Ein zu lesen TM	Ergebnis
Normal	P1 T1 L1 P2 T2 L2	P3 T3 R3	P1 T1 L1 P2 T2 L2 P3 T3 R3
Ohne T	P1 T1 L1 P2 T2 L2	P3 R3	Fehler O/D 61
Ohne P	P1 T1 L1 P2 T2 L2	T3 R3	P1 T1 L1 P2 T2 L2 P25 T3 R3 (außerhalb Magazin)
T besteht schon	P1 T1 L1 P2 T2 L2	P3 T1 R1	Fehler O/D 60
Kein P T besteht schon	P1 T1 L1 P2 T2 L2	T1 R1	Fehler O/D 62

First, the tool memory is cleared. Subsequently the new addresses are loaded.

MC774 = 1	Bestehendes TM	Ein zu lesen TM	Ergebnis
Normal	P1 T1 L1 P2 T2 L2	P3 T3 R3	P3 T3 R3
Ohne T	P1 T1 L1 P2 T2 L2	P3 R3	Fehler O/D 61
Ohne P	P1 T1 L1 P2 T2 L2	T3 R3	P25 T3 R3 (außerhalb Magazin)
T besteht schon	P1 T1 L1 P2 T2 L2	P3 T1 R1	P3 T1 R1
Kein P T besteht schon	P1 T1 L1 P2 T2 L2	T1 R1	P25 T3 R3 (außerhalb Magazin)

The existing tools are not changed and, during input, are missed without a fault report.

MC774 = 2	Bestehendes TM	Ein zu lesen TM	Ergebnis
Normal	P1 T1 L1 P2 T2 L2	P3 T3 R3	P1 T1 L1 P2 T2 L2 P3 T3 R3
Ohne T	P1 T1 L1 P2 T2 L2	P3 R3	Fehler O/D 61
Ohne P	P1 T1 L1 P2 T2 L2	T3 R3	P1 T1 L1 P2 T2 L2 P25 T3 R3 (außerhalb Magazin)
T besteht schon	P1 T1 L1 P2 T2 L2	P3 T1 R1	Fehler O/D 60
Kein P T besteht schon	P1 T1 L1 P2 T2 L2	T1 R1	überspringen

Tool without P overwrites the currently existing tool.

MC774 = 3	Bestehendes TM	Ein zu lesen TM	Ergebnis
Normal	P1 T1 L1 P2 T2 L2	P3 T3 R3	P1 T1 L1 P2 T2 L2 P3 T3 R3
Ohne T	P1 T1 L1 P2 T2 L2	P3 R3	Fehler O/D 61
Ohne P	P1 T1 L1 P2 T2 L2	T3 R3	P1 T1 L1 P2 T2 L2 P25 T3 R3 (außerhalb Magazin)
T besteht schon	P1 T1 L1 P2 T2 L2	P3 T1 R1	Fehler O/D 60
Kein P T besteht schon	P1 T1 L1 P2 T2 L2	T1 R1	P1 T1 R1 P2 T2 L2

15.5 Tool parking time monitoring

If the parking time of a tool (M) or the required parking time (T3=..) of a tool is reached, during the next tool change the replacement tool is loaded automatically.

Addresses in the tool memory:
M tool parking time in minutes
M1 residual tool parking time (only an indication)
M2 tool parking time monitoring (0 = OFF 1 = ON).

The parking time M1 remaining =... can be queried with the function G149 and changed in the tool memory using the function G150.

15.6 Tool breakage monitoring

Machines can be fitted out with a tool breakage monitoring. This function can only be programmed with the help of macros.

The following addresses are used by the tool memory:
B breakage tolerance in mm
R6= radius position for breakage control

In the event of the breakage tolerance being exceeded, tool status E=4 is set and a fault is output in addition.
Also, if at the start of the cycle the tool status is E=1, the breakage control is implemented.
Default value for tolerance is input in MC33.
The breakage monitoring is activated by means of MC32.

The tool breakage monitoring is a machine dependent function. Please consult your user handbook.

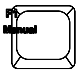
Note

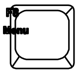
If an original tool is blocked, a replacement tool is automatically loaded in its place (if available).
See G604

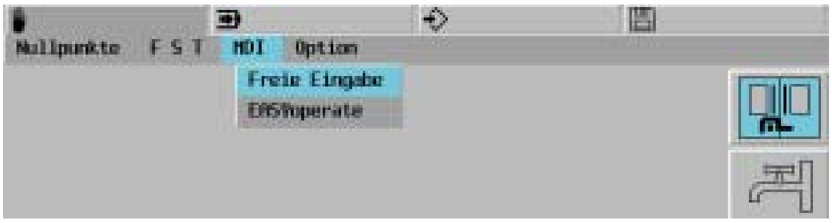
15.7 Manual change of tool (Example)

Change of tool is a machine dependent function. Please consult your user handbook.

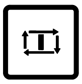
Call-up tool change:







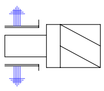
T... M66
Report: int T..



The working area door is unlocked.

Open the working area door.

Please look up the notes on general security



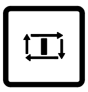
Press "Select tool holder"

Take hold of the tool and use the rotating key or the foot lever "Release tool holder" to halt the machine. The tool holder is released.

Take the tool out of the tool holder.
Insert a new tool.

Release the rotating key or the foot lever and support the tool holder while inserting the tool.

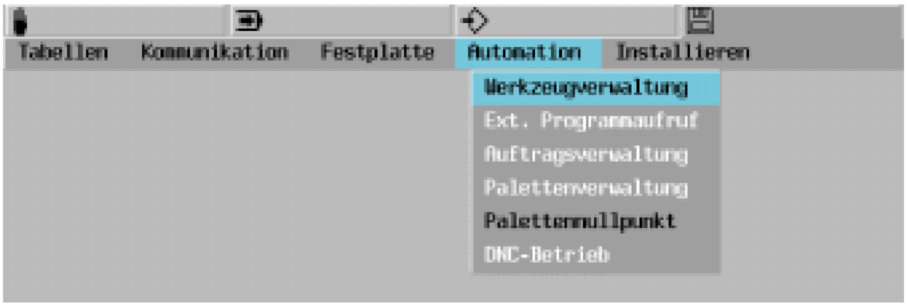
Close the working area doors.

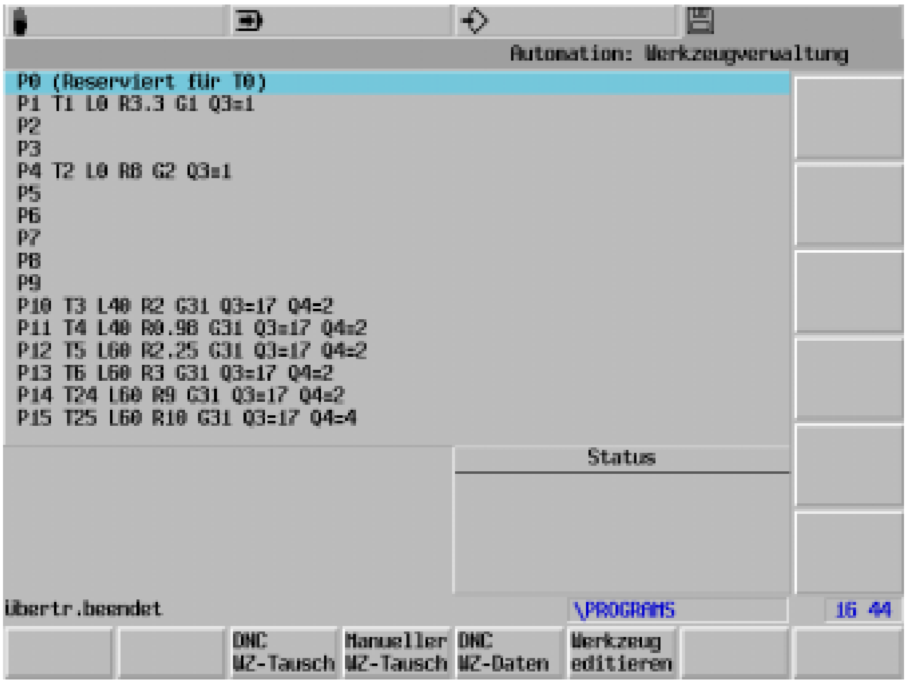


The working area doors are locked.

15.8 Tool management

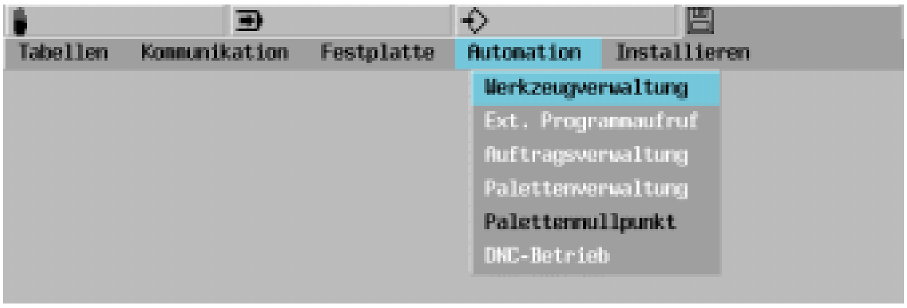
Die Werkzeugverwaltung erlaubt die Eingabe bzw. Entnahme der Werkzeuge aus dem Werkzeugmagazin bei gleichzeitiger Aktualisierung der Werkzeugdaten im Werkzeugspeicher.

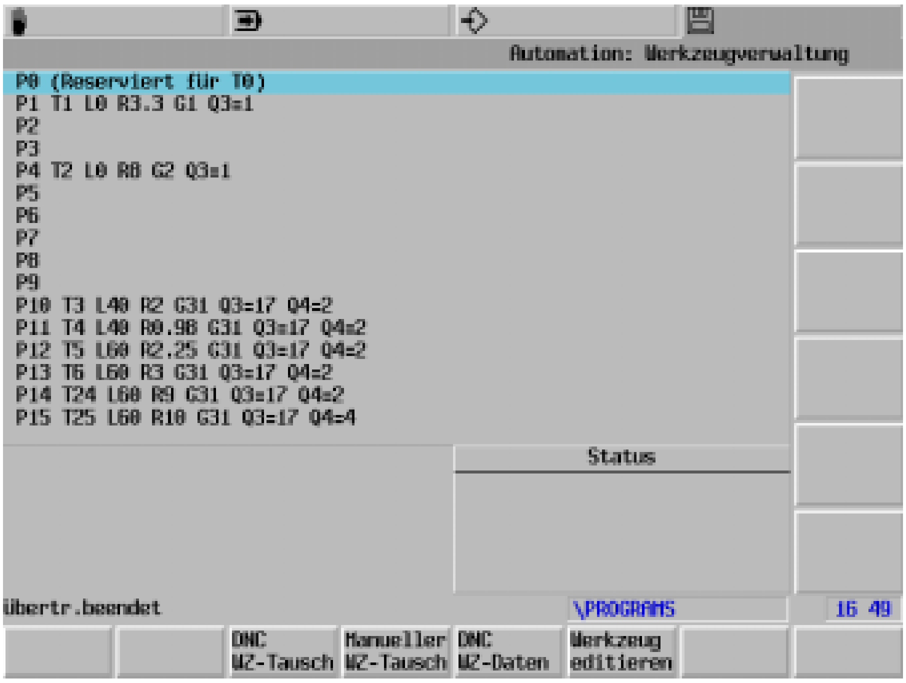


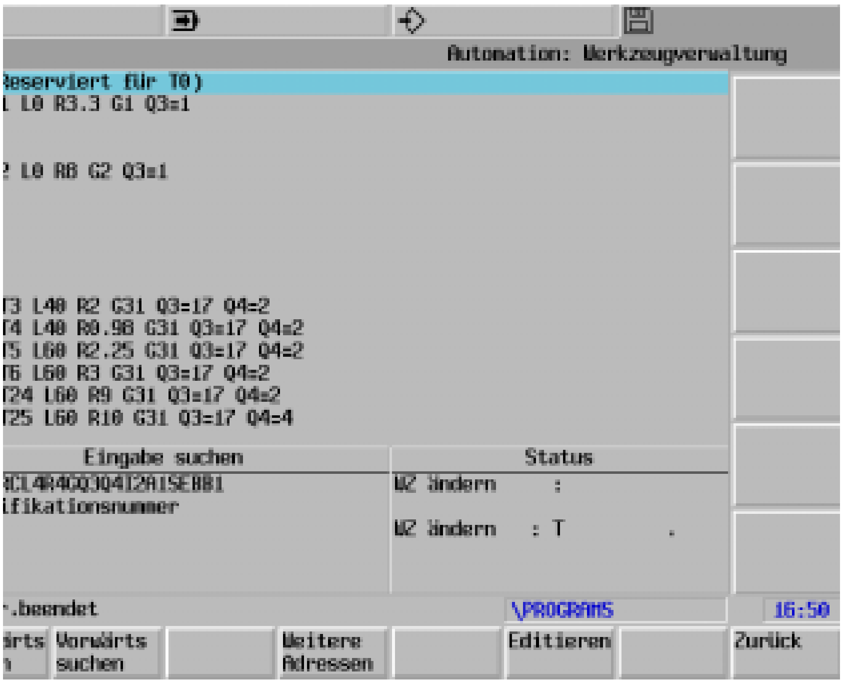


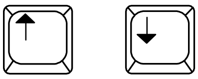
15.8.1 Werkzeugkorrektur

Während der Bearbeitung sein alle Werkzeugdaten bis auf das Spindelwerkzeug editierbar.



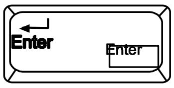


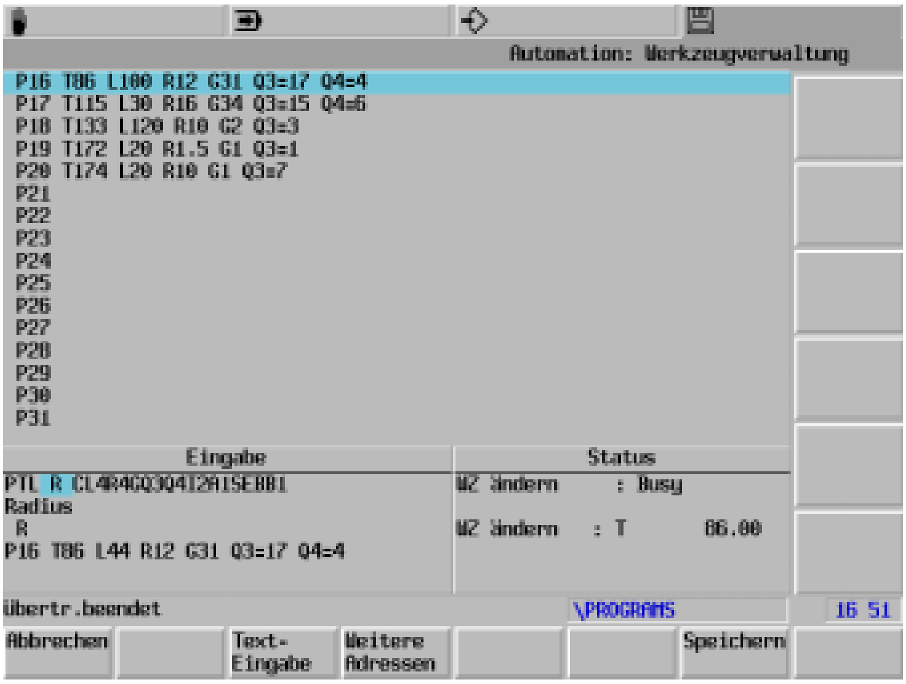


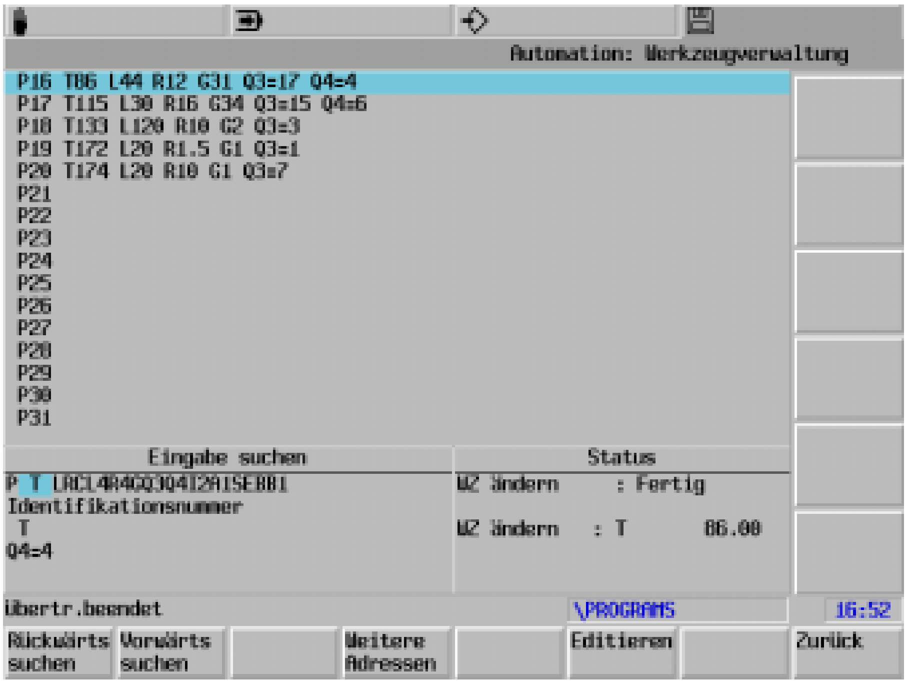


Satz anwählen

Eingabe L44






Automation: Werkzeugverwaltung
P16 T86 L44 R12 G31 Q3=17 Q4=4
P17 T115 L30 R16 G34 Q3=15 Q4=6
P18 T133 L120 R10 G2 Q3=3
P19 T172 L20 R1.5 G1 Q3=1
P20 T174 L20 R10 G1 Q3=7
P21
P22
P23
P24
P25
P26
P27
P28
P29
P30
P31
Eingabe suchen
P T LRCL4R4GQ3Q4I2A1SEBB1
Identifikationsnummer
T
Q4=4
Status
WZ ändern : Fertig
WZ ändern : T 86.00
übertr.beendet
\PROGRAMS
16:52
Rückwärts suchen
Vorwärts suchen
Weitere Adressen
Editieren
Zurück

15.8.2 To take a tool out of the tool magazine (example)

Manueller WZ-Tausch

	Werkzeug tauschen	Werkzeug einfügen	Werkzeug entfernen		Autom. M-Platz		Zurück

Werkzeug entfernen

Select the tool or enter the tool number.

	Gewähltes WZ.entf.					Zeichen suchen	Zurück

Gewähltes WZ.entf.

Abbrechen			Starten			Zeichen suchen	

Starten — The tool magazine is in position.

```
              Status
WZ entfernen  : Busy
Positioniert auf    : P 10

WZ entfernen: T       3.00
```

Abbrechen				Werkzeug entfernen		Zeichen suchen	

Werkzeug entfernen — Confirmation that the tool has been removed.

```
              Status
WZ entfernen  : Fertig
Positioniert auf    : P 10

WZ entfernen: T       3.00
```

15.9 To update enhanced tool measurement

The machine and MillPlus must be prepared by the machine manufacturer for the TT120/TT130 measuring key system or the laser measurement system. Please consult your user handbook.

The menu and the corresponding machine settings are updated using the following machine settings:

MC261 >0: measurement cycle functions
MC254 >0: tool measurement
MC840 =1: present measuring key
MC854 =1: tool measuring equipment type (0=none, 1=laser, 2=TT120/TT130)
MC859 =1: signal type 2 measuring key

MC356 measurement: radial axis: 1=X, 2=Y, 3=Z
MC357 measurement: tool axis: 1=X, 2=Y, 3=Z
MC358 measurement: 3 axis 0=off, 1=on
MC359 radial measurement: -1=neg, 0=aut, 1=pos

MC370 measurement: maximum tool radius
MC371 measurement: maximum tool length
MC372 free space under the laser beam

MC350 position 1 negative axis
MC351 position 1 positive axis
MC352 position 2 negative axis
MC353 position 2 positive axis
MC354 position 3 negative axis
MC355 position 3 positive axis

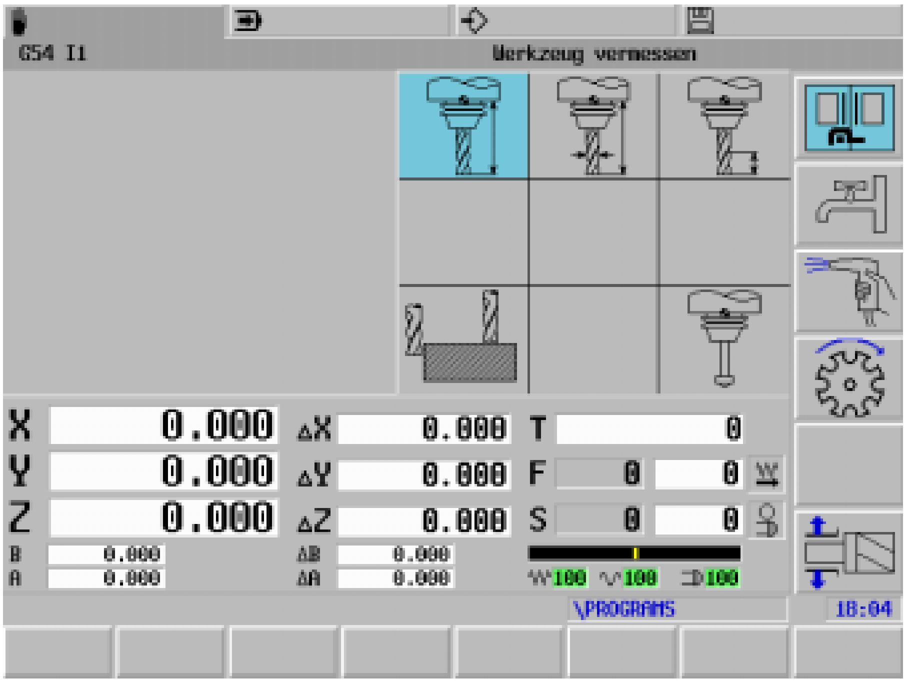
Following calibration, the exact values are stored in MC350 to MC355.

MC392 Maximum measuring error with tool rotating [µm]
MC394 Inching motion with tool not rotating [mm/min]
MC395 Gap between bottom edge of tool and bottom edge of stylus [µm]
MC396 Diameter of TT120/TT130 stylus [µm]
MC397 Pre-positioning safety zone [µm]
MC398 Rapid motion in inching cycle [mm/min]
MC399 Maximum speed of rotation [m/min]

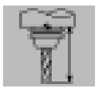
15.10 Tool measurement using the laser measurement system

You can measure tools automatically using the laser measurement system and the MillPlus tool measurement cycle. The correction values for length and radius are placed in the tool memory.

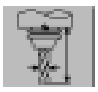
After selecting "Measure tool" the following menu screen appears (MC254=1):



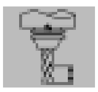
The following cycles are available:



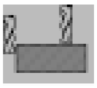
Measurement of the tool length of concentric tools



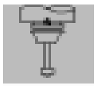
Measurement of tool length and radius of eccentric tools



Individual cutting control

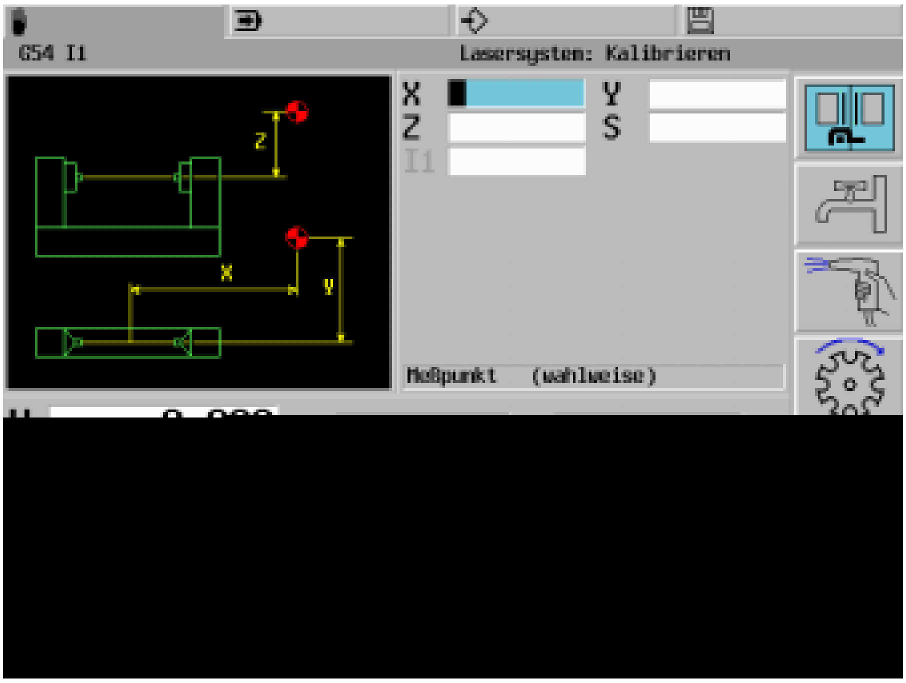


Measure tool manually (see the chapter on setting the axis value)



Calibration of the laser measurement system

15.10.1 Calibration of the laser measurement system



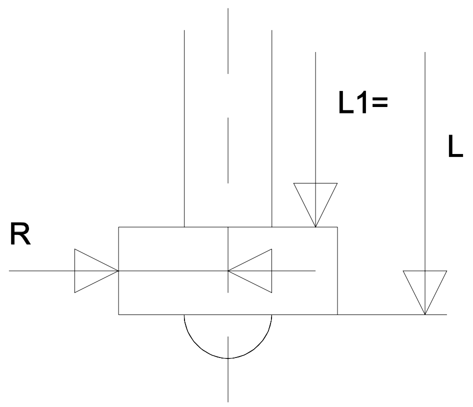
Addresses:
X1 = measuring equipment - X position
Y1 = measuring equipment - Y position
Z1 = measuring equipment - Z position
S = revolutions per minute (recommended value for S = 3000)

During calibration the exact position of the measuring equipment is measured and stored in MC350-MC355. The stored values refer to the machine reference point.

X1,Y1,Z1 is the global position (+/- 5 mm exactly) of the measuring equipment referred to the machine null point.
If X1,Y1 or Z1 have not been entered, the calibrated settings from the machine constants are used.

- Rotary axes are not taken into account or positioned.
- Null point displacements are not allowed to be active if X1,Y1 or Z1 have been entered.
- G7 free operating planes are not allowed to be active.
- A calibrator tool must be chosen. T0 is not allowed.

The dimensions of the calibrator spike are entered in the tool memory.
L = length of the calibrator spike (lower surface of the cylindrical part)
R = radius
L1= first additional length (upper surface of the cylindrical part)
The length L1= is not entered if a cylinder pin is employed. In this case, only the upper surface of the laser beam is calibrated.

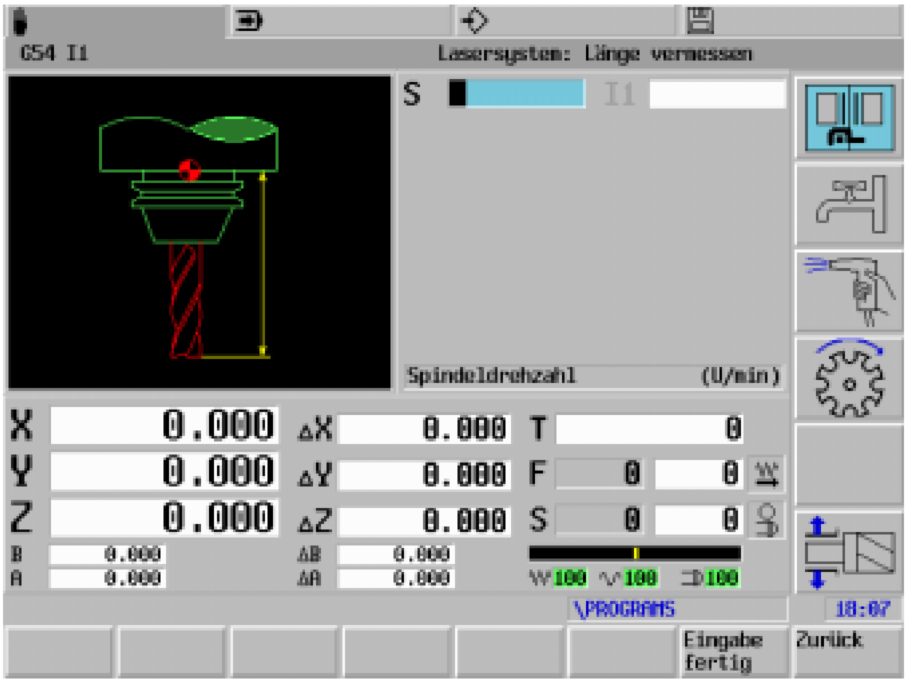


Calibrator tool definition in the tool memory

Note:

See G600

15.10.2 Measurement of the length of concentric tools



Addresses:
S = Speed in revolutions per minute (recommended value for S = 3000)

The following addresses from the tool memory are used:
L5= length tolerance

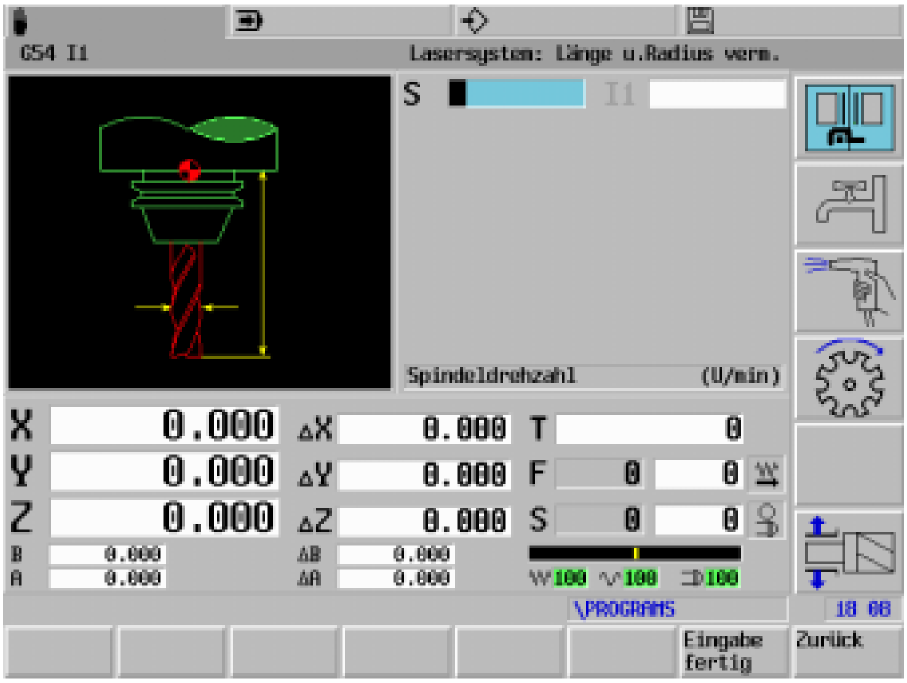
"Measure" (E=0 or no value): During the initial measurement the tool length is overwritten, the oversize L4 =0 and the tool status is set to E =1.
"Check" (E=1): The measured variance is added to the tool table at L4.

- If the tolerance is exceeded, the tool status is set to E1 and, in addition, a fault is displayed in Easy Operate.
- If the tool status is at E1 during start-up, then a fault is displayed in Easy Operate.
- The advance measurement is calculated from the speed in rpm.
- The measurement is effected while the spindle is rotating.

Note:

See also G601
The measuring cycle is suitable for measuring the length of the following tools:
twist drills, centre drills, tapping drills, countersinks, reamers, grinding points, etc.

15.10.3 Measurement of the length and radius of eccentric tools



Addresses:
S = speed in revolutions per minute (recommended value S=3000)

The following addresses from the tool memory are used:
L5= wear tolerance - length
R5= wear tolerance - radius
L6= length offset
R6= radius offset
Q4= cutting number

Checking (E=1):
The measured variance is added to the tool table at L4 and R4.
Measurement (E=0 or no value):
During the initial measurement,the tool length and radius are overwritten, the oversize L4 and R4=0 and the tool status is set to E=1.

- If the tolerance is exceeded, the tool status is set to E1 and, in addition, a fault is displayed in Easy Operate.
- If the tool status is at E1 during start-up then a fault is displayed in Easy Operate and the cycle is stepped forward to automatic operation.
- The advance measurement is calculated from the speed in rpm.
- The measurement is effected while the spindle is rotating.

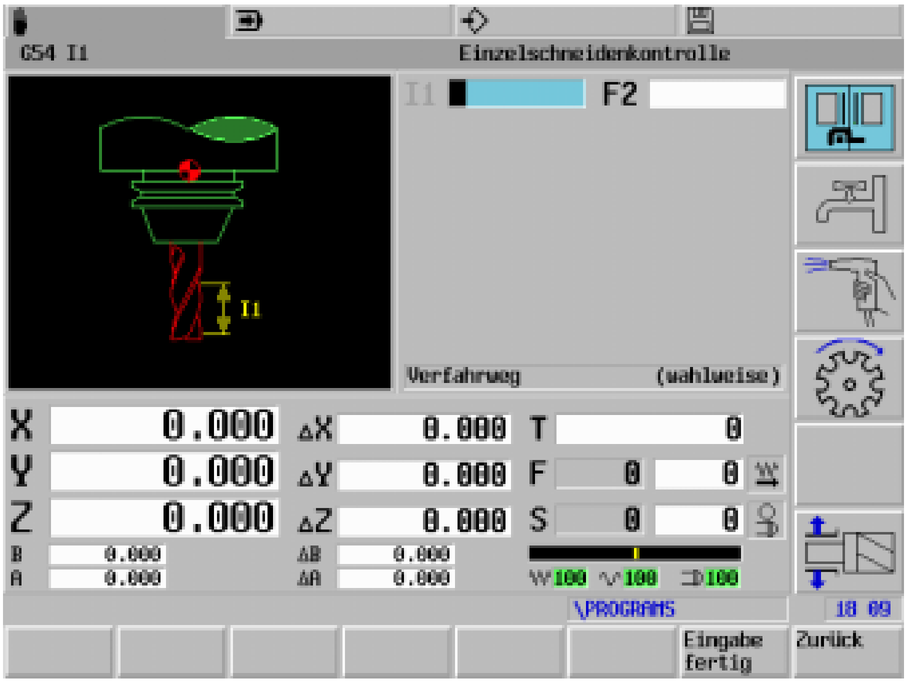
Measurement of the radius:
- If L6>0 or has not been entered (in the tool table) a radius measurement is carried out.

Circular working check:
- If Q4>0 (Number of cogs from the tool table) then, following the measurement of the radius a circular working check is carried out.
- The circular working check is carried out using a calculated speed in rpm.
- The speed override is inactive.
- The maximum tolerance is determined using L5.

Note:

See G602

15.10.4 Individual cutting control



Addresses:
I1 = Travel
F2 = Scan advance

The following addresses from the tool memory are used:
R5= radius wear tolerance
L6= length offset
Q4= cutting number

- If the tolerance is exceeded, the tool status is set to E-1 and, in addition, a fault is displayed in Easy Operate.
- If, during the start-up of the cycle, the tool status is E=1, then a fault is displayed in Easy Operate and the cycle is advanced to automatic operation.

- If I1=0 only the circular working check is carried out.
- The cutting control is implemented with a calculated speed in rpm.
- The speed override is inactive.
- The maximum error is determined using R5.

Note:

See G603

15.11 Laser measurement cycles in the program

15.11.1 Beispiel

```
N12345
N1 G54 I1
N100 T1 M6 ... (mill D50)
...  \
...   milling operation
...  /
N191 G602 S3000 (measurement of length and radius wear)
N200  T2 M6 ... (drill D4)
...  \
...   drill operation
...  /
N291 G604 S3000 (breakage monitoring)
N300 M30
```

Tool memory of program start-up.
Tools are measured beforehand using the measurement cycles.
The mill is blocked (E-1) if the parking time has elapsed or the wear limit is exceeded.
The drill is blocked (E-1) if the parking time has elapsed. During a breakage the drill is blocked (E-4) and a program stoppage with faults is executed.

50mm diameter milling with replacement tool:
P.. T1.01 L102.023 R24.978 L4=0 R4=0 E1 M15 M2=1
P.. T1.02 L102.167 R24.986 L4=0 R4=0 E1 M15 M2=1

4mm diameter drill with replacement tool:
P.. T2.01 L85.467 L4=0 E1 B1 M15 M2=1
P.. T2.02 L85.246 L4=0 E1 B1 M15 M2=1

15.12 Tool fault reports

If a tool fault (breakage, wear or concentric operation) is identified, then the E status is changed in the tool table.

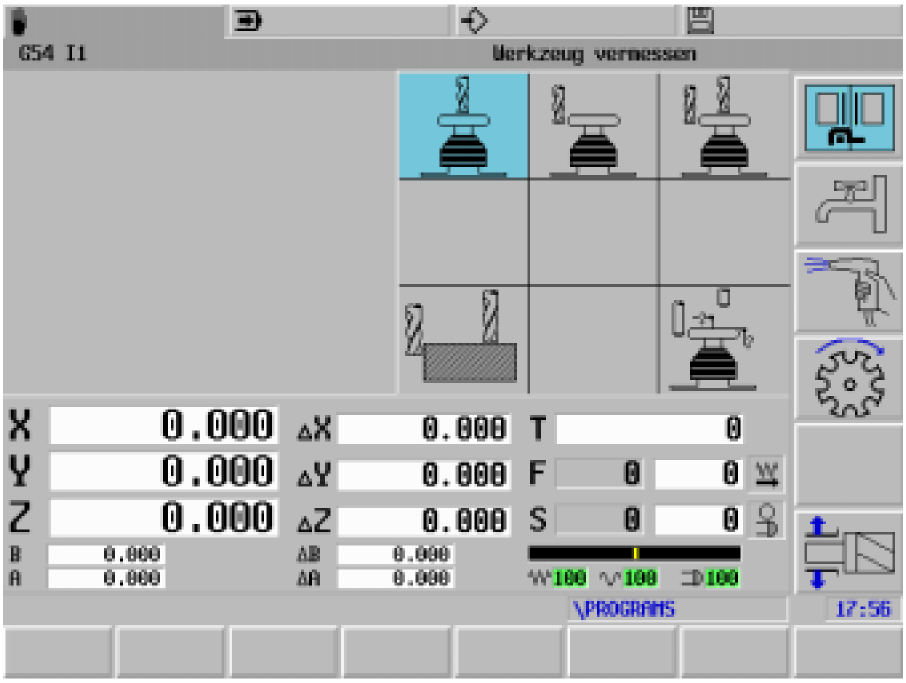
E= -1 Tool is outside tolerance.
E= -4 Tool is broken.

Particulars are recorded by the cycles affected.

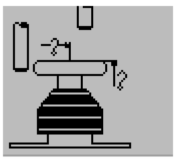
15.13 Tool measurement using the TT120/TT130

Using the TT130 and the MillPlus tool measurement cycles you can measure tools automatically. The correction values for length and radius are placed in the tool memory.

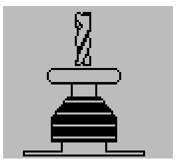
After selecting "Measure tool", the following menu screen appears (MC854=2):



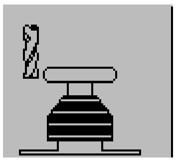
The following cycles are available:



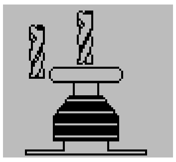
Calibration of the TT120/TT130



Tool length measurement



Tool radius measurement



Tool length and radius measurement

Tool length and radius
Before you measure tools for the first time, enter the approximate radius (R10), the approximate length (L100), the number of cuts (Q4=4) and the cutting direction (I2=0) of the tool to be used in the tool table.

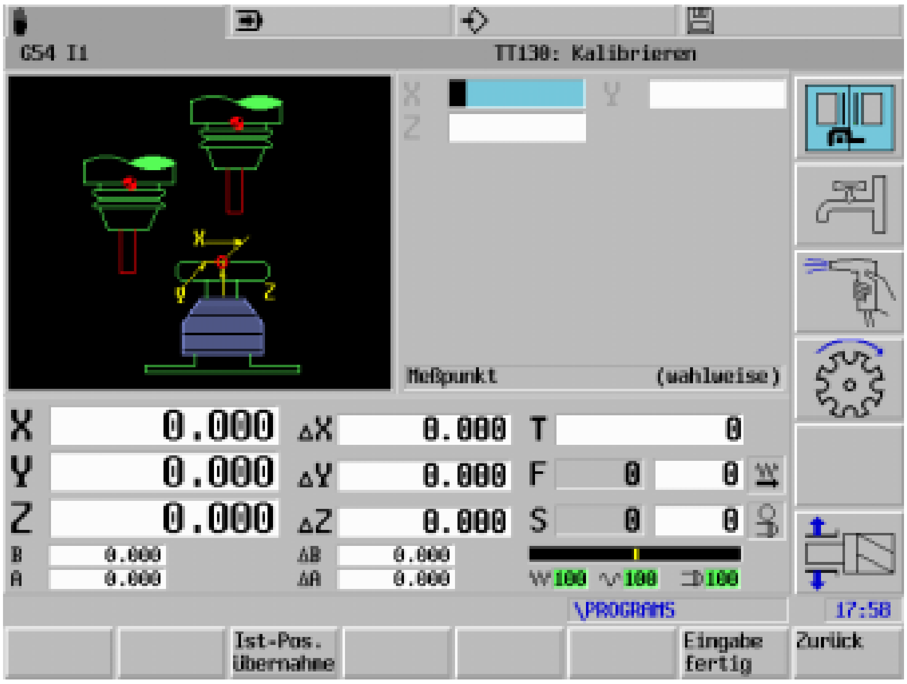
Measurement results
During the initial measurement, MillPlus overwrites the tool radius (R10 with R10.012) and the tool length (L100 with L99.456) in the tool memory and sets the oversizes R4 and L4 = 0.

Checking the tool
In the event that you check a tool, the measured tool data is compared with the tool data extracted from the tool memory. MillPlus calculates the mathematically correct variances and enters these as oversizes (R4=0.015 and L4=0.06) in the tool memory.

Radial axis approach direction
The approach direction depends on the position of the masuring key system. It is automatically keyed from the direction in which the greatest travel range is available.

15.14 Calibration of the TT120/TT130

The way in which the calibration cycle functions depends on MC854.



Before you calibrate, you must enter the exact radius and the exact length of the calibration tool in the tool table.

The location of the TT120/TT130 must be set down in the working area of the machine in MC350 MC352 MC354. If you change any of MC350 MC352 MC354, you must recalibrate.

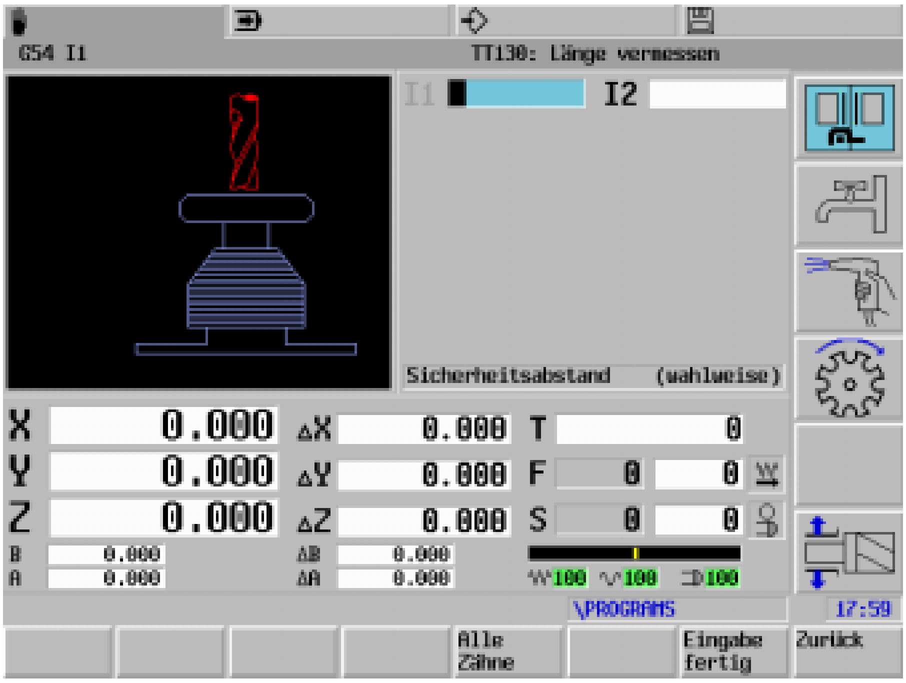
You calibrate the TT120/TT130 using the measurement cycle dialog entry. The calibration process starts up automatically. MillPlus also determines the mean setting of the calibrator tool automatically. To achieve this, MillPlus rotates the spindle through 180° at the halfway point of the calibration cycle. As the calibrator tool you employ an exactly cylindrical component, for example a cylinder pin. MillPlus stores the calibration values with the machine settings and takes account of them in subsequent tool measurements.

Position:
Enter the position in the X, Y and Z axes, such that a crash with workpieces or tool holders is excluded. If the position height input is so small that the tool tip would lie below the upper surface of the plate, MillPlus automatically positions the calibrator tool over the plate.

Note:

See G606

15.14.1 Tool length measurement



You can determine the tool length in three different ways:

If the tool diameter is greater than the diameter of the measuring area of the TT120/TT130, then you measure with the tool rotating.

If the tool diameter is smaller than the diameter of the measuring area of the TT120/TT130 or if you are determining the length of drills or radius form millers, then you measure with the tool stationary.

All cogs are measured using the soft key "All teeth". The measurement is effected with the spindle stationary. The largest cog lengths are stored in the tool table.

Measurement start-up:"Measurement with the tool rotating".
In order to determine the longest cuts the tool is set at the midpoint of the measuring key system and rotating on the measuring area to be traversed. You program the setting in the tool table under the tool setting; radius (R6=).

Measurement start-up:"Measurement with the tool stationary" (e.g. for drills).
The tool to be measured is moved to the dead centre position over the measuring area. Finally it is moved on to the measuring area with the spindle stationary. To make this measurement, you enter the tool setting: radius (R6=0) in the tool table.

Measurement start-up:"Individual cut measurement"
MillPlus pre-positions the tool to be measured to the side of the probe. In this way, the tool face is positioned below the top of the probe, as prescribed in MC395. In the tool table, under tool setting; length (L6=) you can prescribe an additional setting. MillPlus approaches radially with the tool rotating in order to determine the start angle for individual cut measurement. Finally, the length of all cuts is measured by changing the orientation of the spindle. You select the soft key "All teeth" to make this measurement.

Checking the tool:

During the initial measurement, MillPlus overwrites the tool length L in the tool memory and sets the oversize L4=0. In the event that you are checking a tool, the actual length measured is compared with tool length L extracted from the tool table. MillPlus calculates the mathematically correct variance and enters this as the oversize L4 in the tool table. If this oversize is greater than the permissible wear or breakage tolerance for the tool length, then a fault report is made.

Safe height:

Enter a position in the spindle axis, by means of parameters from the entry dialog (I1 = safety distance), such that a crash with pieces of work or their supporting holders is excluded. The safe height refers to the reference point for the active piece of work. If the safe height entered is so small that the tool tip would lie below the top surface of the plate, MillPlus does not automatically place the tool over the plate (security zone from MC397)

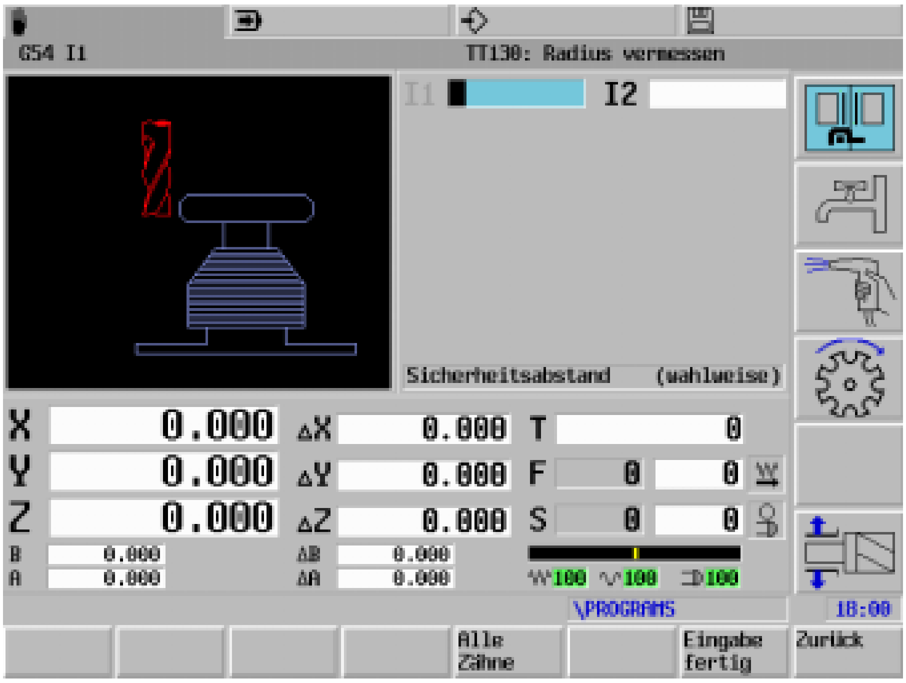
Cut measurement:

switch on or off individual cut measurement (Parameter I2=)

Note:

See G607

15.14.2 Measurement of tool radius



You can determine the tool radius in two ways:
Measurement with the tool rotating.
Measurement with the tool rotating, followed by individual cut measurement

Measurement start-up
MillPlus pre-positions the tool to be measured to the side of the probe. In this way, the mill face is placed below the top of the probe, as prescribed in MC 395. MillPlus approaches with the rotating tool. In the event that an individual cut measurement should be implemented as well, the radii of old cuts are measured by means of the spindle orientation.

Checking the tool:
During the initial measurement MillPlus overwrites the tool radius R in the tool memory and sets the oversize R4=0. In the event that you are checking a tool, the measured radius is compared with the tool radius R extracted from the tool table. MillPlus calculates the mathematically correct variance and enters it as oversize R4 in the tool table. If this oversize is greater than the permissible wear or breakage tolerance for the tool radius, then a fault report is generated.

Safe height:
Enter a position in the spindle axis, by means of parameters from the entry dialog (I1 = safety distance), such that a crash with pieces of work or their supporting holders is excluded. The safe height refers to the active workpiece reference point. If the safe height entered is so small that the tool tip would lie below the top surface of the plate, MillPlus does not automatically place the tool over the plate (security zone from MC397)

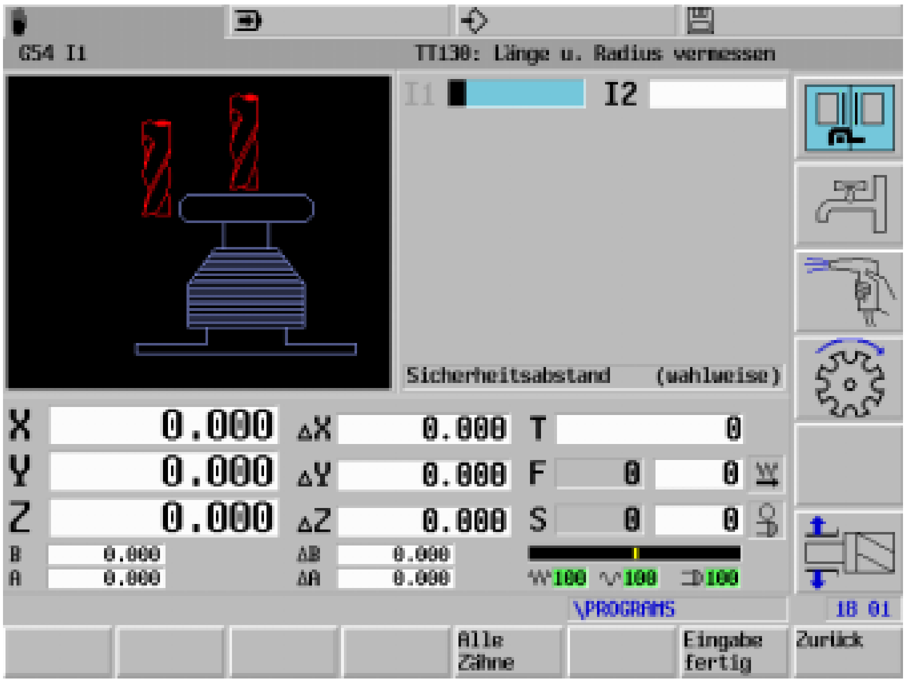
Cut measurement:

switch on or off individual cut measurement (Parameter I2=)

Note:

See G608

15.14.3 Measure the tool completely



You can measure the tool in two ways:
Measurement with the tool rotating
Measurement with the tool rotating, followed by individual cut measurement

Measurement start-up
MillPlus measures the tool in accordance with a pre-programmed start-up routine. The tool radius is measured first, followed by measurement of the tool length.

Checking the tool:
During the initial measurement MillPlus overwrites the tool radius R and the tool length L in the tool memory and sets the oversizes R4 and L4 = 0. In the event that you are checking a tool,the measured tool data is compared with the tool data in the tool table. MillPlus calculates the mathematically correct variances and enters these as oversizes R4 and L4 in the tool table. If any of the oversizes is greater than the permissible wear or breakage tolerances, then a fault report is generated.

Safe height:
Enter a position in the spindle axis, by means of parameters from the entry dialog (I1 = safety distance), such that a crash with pieces of work or their supporting holders is excluded. The safe height refers to the active workpiece reference point. If the safe height entered is so small that the tool tip would lie below the top surface of the plate, MillPlus does not automatically place the tool over the plate (security zone from MC397)

Cut measurement:

switch on or off individual cut measurement (Parameter I2=)

Note:

See G608

15.15 Update machine settings

MillPlus employs the approach rate from MC394 for measurement with the spindle stationary.

MillPlus automatically calculates the spindle speed in rpm and the approach rate during measurement with the tool rotating. The spindle speed in rpm is calculated in the following way:

$$n = \frac{MC399}{r \cdot 0.0063}$$

where:

n	= turning speed in revolutions/min
MC399	= maximum permissible turning speed in metres per minute [m/min]
R	= active tool radius [mm]

The approach rate is calculated from:
V = measurement tolerance • n

where:

V	= approach rate [mm/min]
measurement tolerance	= measurement tolerance [mm], dependent on MC391
N	= speed in revolution per minute [1/min]

where:
You enter in MC391 the calculation for the approach rate:

MC391=0:
the measurement tolerance remains constant - independent of the tool radius. However, with very large tools the approach rate is reduced to zero. The lower you choose the maximum turning speed (MC399) and the permissible tolerance (MC392), this effect is produced in order to make it perceptible earlier.

MC391=1:
The measurement tolerance is altered with increasing tool radius. This guarantees an approach rate which is still appropriate for large radius tools. MillPlus alters the measurement tolerance in accordance with the following table:

Tool radius	Measurement tolerance
up to 30 mm	MC392
30 to 60 mm	2 • MC392
60 to 90 mm	3 • MC392
90 to l20 mm	4 • MC392

MC391=2:
The approach rate remains constant; however, the measurement error increases linearly as the tool radius becomes larger:

$$\text{measurement tolerance} = \frac{r \cdot MC392}{5mm}$$

where:

r	= tool radius [mm]
MC392	= maximum permissible measurement error

Synopsis of machine settings:

The TT120/TT130 function can be activated by means of MC854. Following a reboot of the CNC the following machine settings are then available.

MC NUMMER	FUNKTION	EINGABE
MC391	Berechnung des Antastvorschubs.	0 Berechnung des Antastvorschubs mit konstanter Toleranz. 1 Berechnung des Antastvorschubs mit variabeler Toleranz. 2 Berechnung Antastvorschub
MC392	Maximal zulässiger Meßfehler bei Werkzeug-Vermessung mit rotierendem Werkzeug	2 1000 m
MC394	Antastvorschub bei Werkzeug-Vermessung mit nicht rotierendem Werkzeug	10 3000 mm/min
MC395	Abstand Werkzeug-Unterkante zu stylus Oberkante bei Werkzeug Radius-Vermessung.	1 100000 m
MC396	Dürchmesser bzw. Kantenlänge des Stylus des TT120/TT130.	1 - 100000 m
MC397	Sicherheidszone um den Stylus des TT120/TT130 für vorpositionierung.	1 10000 m
MC398	Eilgang im Antast-Zyklus für TT120/TT130.	10 10000 mm/min
MC399	Maximal zulässige Umlaufgeschwindigkeit an der Werkzeugschneide.	1 120 m/min
MC854	Werkzeug-Vermessungs-Type	0=kein,1=Laser,2=TT120/TT130
MC350 MC352 MC354	Koordinaten des TT120/TT130 Stylus-Mittelpunkts bezogen auf den Machinen-Referenzpunkt.	-max - +max m

15.16 TT120/TT130 measurement cycles for automatic operation

15.16.1 Example

N66666
N1 G54 I1
N100 T1 M6 ... (mill D50)
... \
... milling operation
... /
N191 G609 (measurement of length, radius wear)
N200 T2 M6 ... (drill D4)
... \
... drill operation
... /
N291 G607 (measurement of length, breakage monitoring)
N300 M30

Tool memory at program start-up.
Tools are measured beforehand using the measurement cycles.
The mill is blocked (E-1) if the parking time has elapsed or if the wear limit has been exceeded.
The drill is blocked (E-1) if the parking time has elapsed. During a breakage the drill is blocked (E-4) and a program stoppage with fault executed.

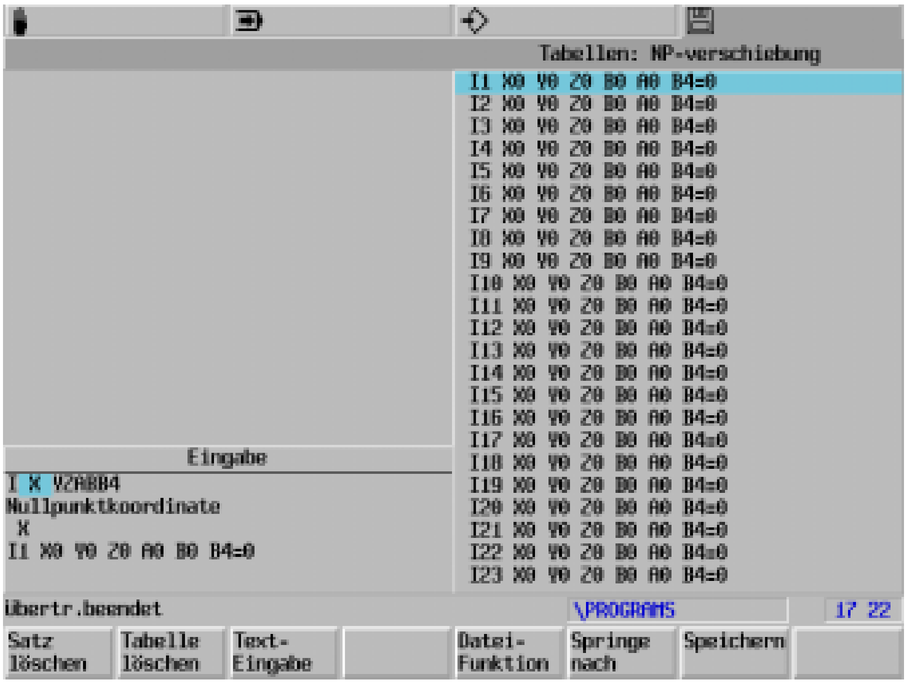
50mm diameter milling with replacement tool:
P.. T1.01 L102.023 R24.978 L4=0 R4=0 E1 M15 M2=1
P.. T1.02 L102.167 R24.986 L4=0 R4=0 E1 M15 M2=1

4mm diameter drill with replacement tool:
P.. T2.01 L85.467 L4=0 E1 B1 M15 M2=1 R6=0
P.. T2.02 L85.246 L4=0 E1 B1 M15 M2=1 R6=0

16. Tables

16.1 Zero offset

Display and entry



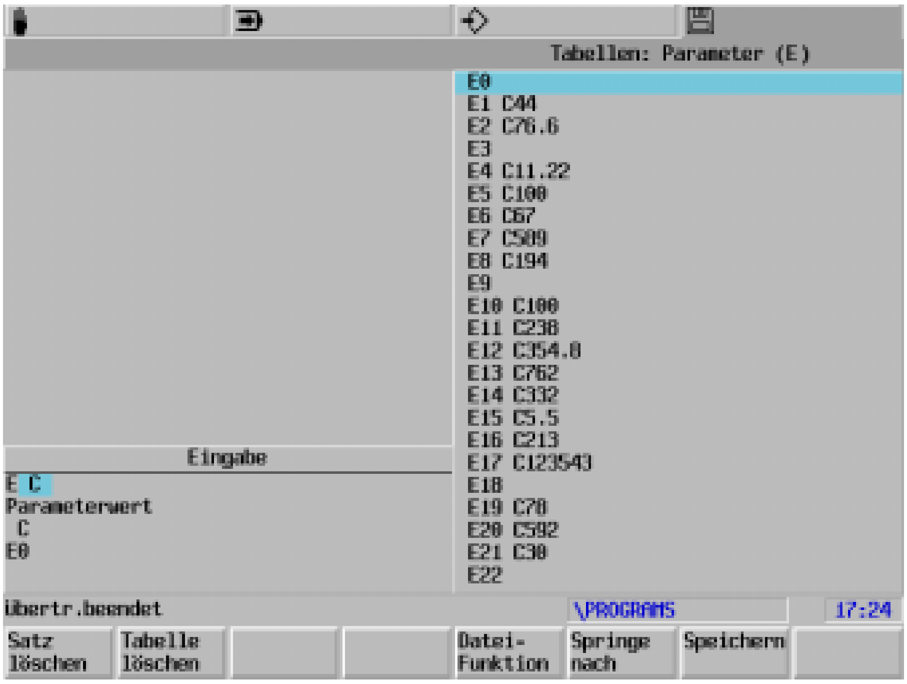
Note

mc84>0
Zero offset G54 I1-I99
Storage name ZE.ZE

mc84=0
Zero offset G51-G59
Storage name ZO.ZO

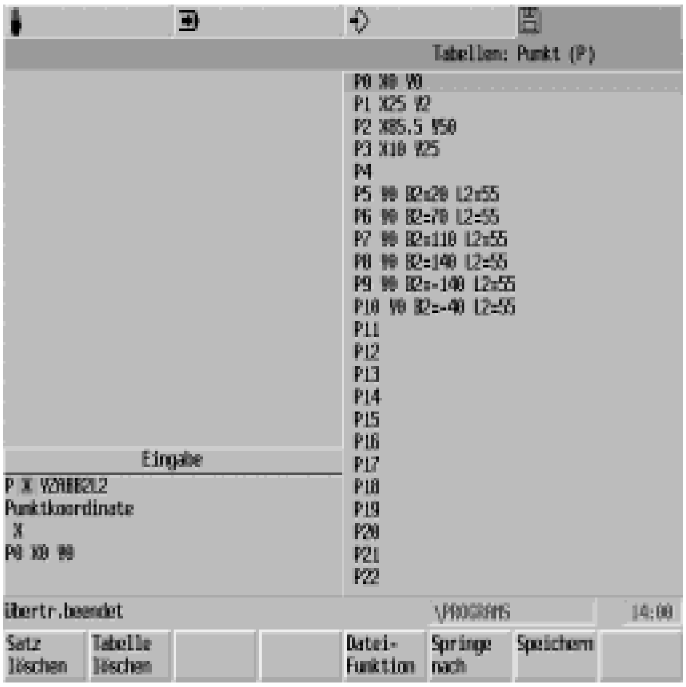
16.2 Parameter(E)

Display and entry of the E parameters



16.3 Point (P)

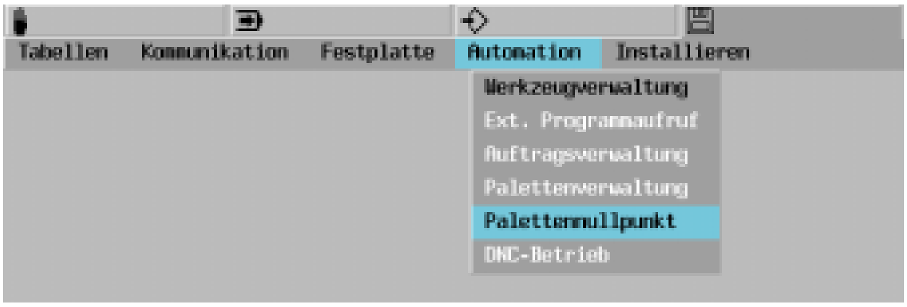
Display and entry of the point definitions



16.4 Pallet zero point

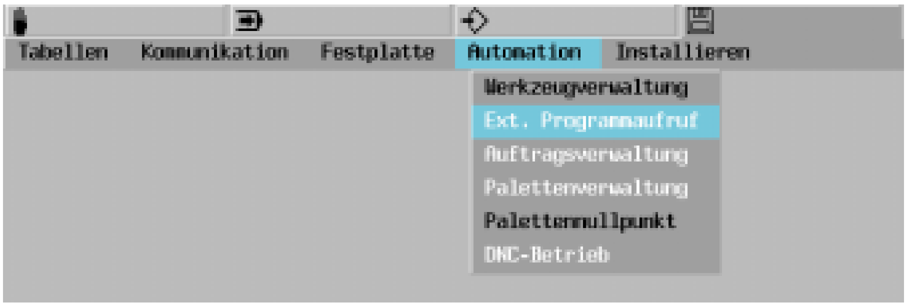
Only with activated ZE.ZE-memory: (see zero offset).

Storing the pallet zero point.



Note:
See Technical Handbook for more information.

17. Automation

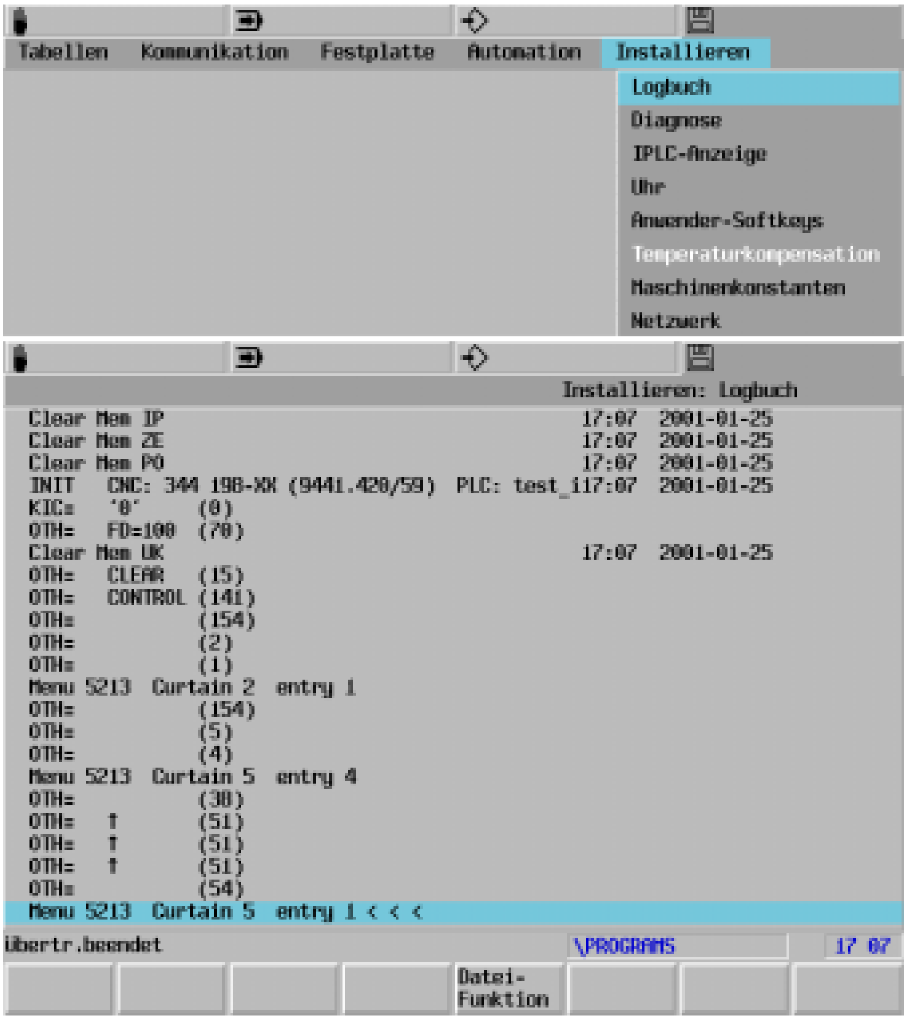


Refer to the documentation provided by the machine builder for information regarding the Ext. program call, job administration, palette management and DNC mode functions.

18. Installation

18.1 Logbook

The most recent inputs from the keyboard are stored in the logbook.

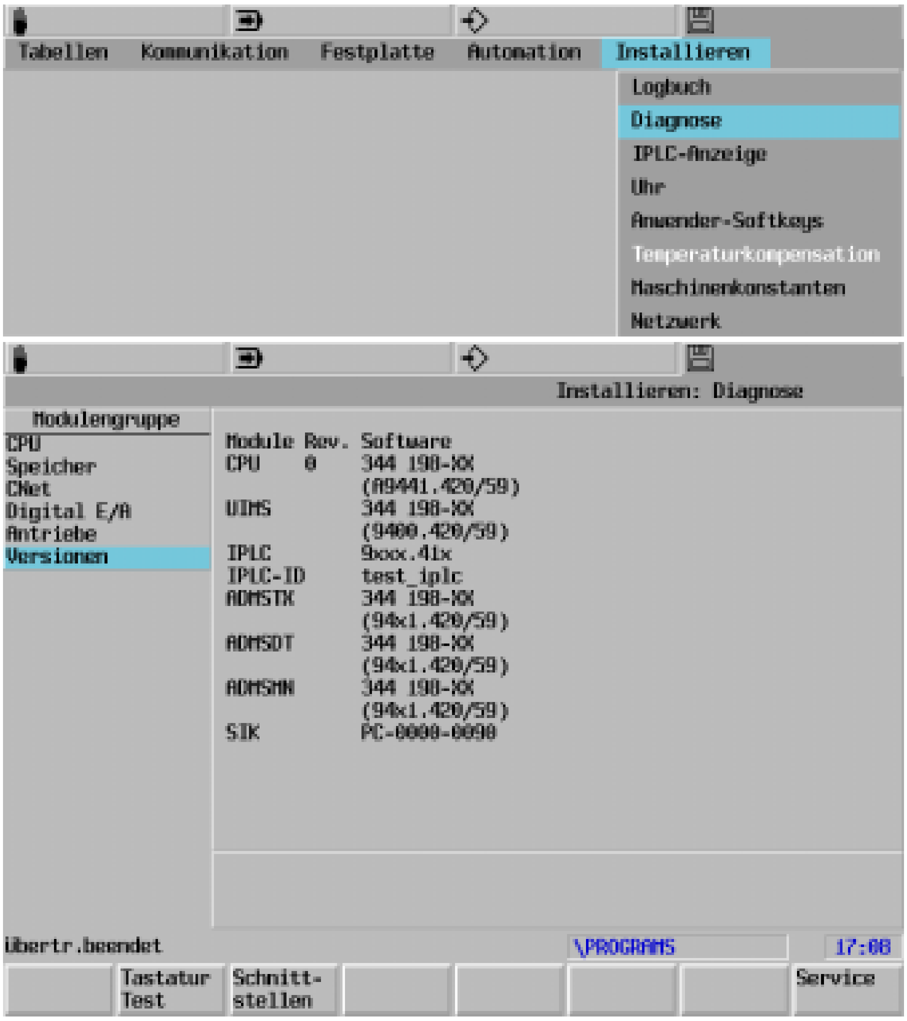


18.1.1 Error log

Display of the last error messages (only in Manual and Automatic operating modes).

18.2 Diagnostics

System information may be displayed in Diagnosics.



18.2.1 Remote diagnosis

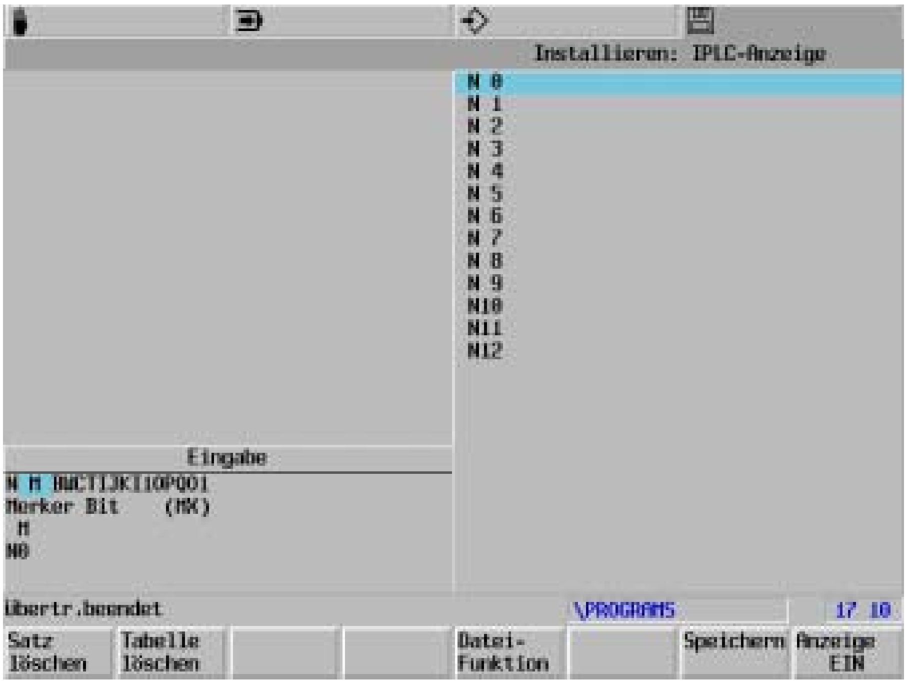
Setting the CNC up for remote diagnosis. The display changes to black/white.

18.3 Clock

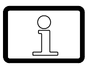
Entering and storing the real time.

18.4 IPLC monitor

This function only to be used by maintenance/customer service personnel.



18.4.1 I/O layout



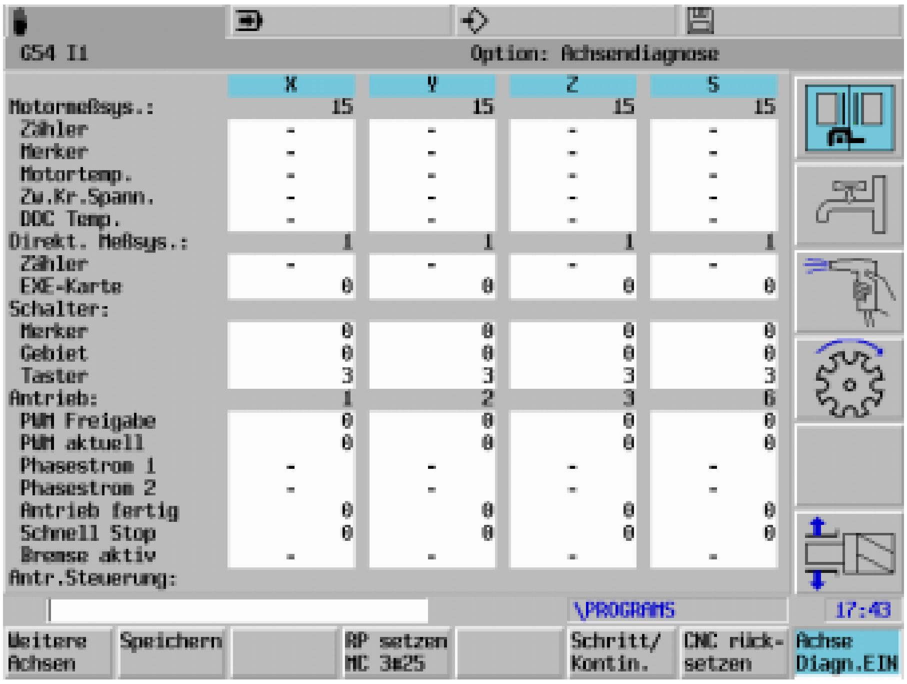
Status indication of I / O layout (only in Manual and Automatic operating modes)

18.5 Temperature compensation

This function only to be used by maintenance/customer service personnel.

18.6 Axes diagnose

This function only to be used by maintenance/customer service personnel.



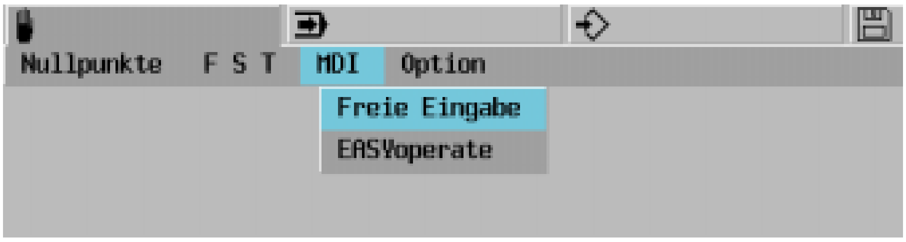
Note

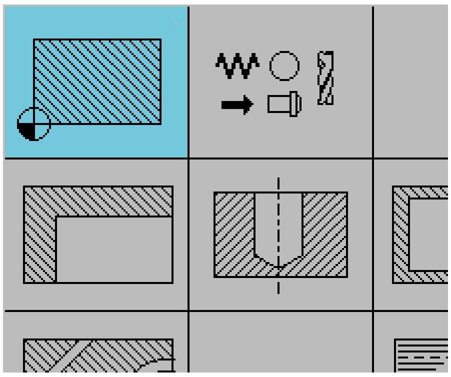
Displayed only when diagnosis switch on.

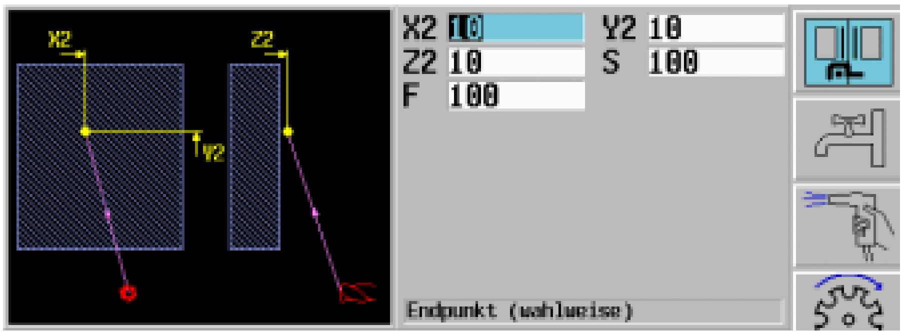
19. Easy Operate

Easy Operate is used for the programming on the machine of simple machining operations.
Before machining can be started, F,S,T are to be activated and the spindle must be switched on (not with Graphic mode).

19.1 Contours







Parameters
X2,Y2,Z2 Endpunkt (wahlweise)

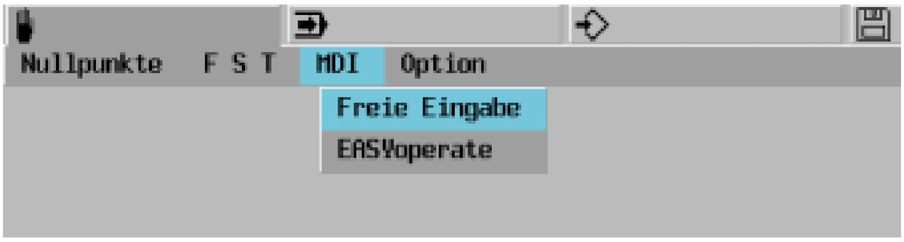
S Spindeldrehzahl (U/Min)
F Vorschub

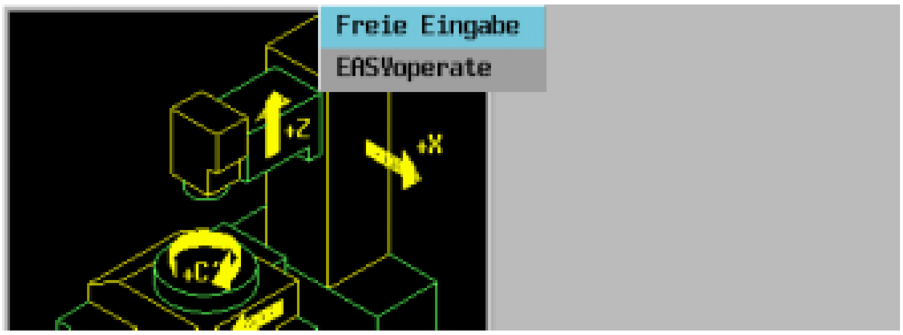
Note:

Whether or not radius corrections is performed depends on the present tool position;
if the present tool position is inside/outside the contour being programmed, radius correction is performed inside/outside the equidistant path.

* = S and F must be activated in advance, by means of F,S,T or MDI

19.2 Facing / edging





Parameters

X2,Y2 Endpunkt (wahlweise)
C1 Zustellbreite (wahlweise)
C2 Zustelltiefe (wahlweise)
Z2 Tiefe
S Spindeldrehzahl (U/Min) *
F Vorschub *

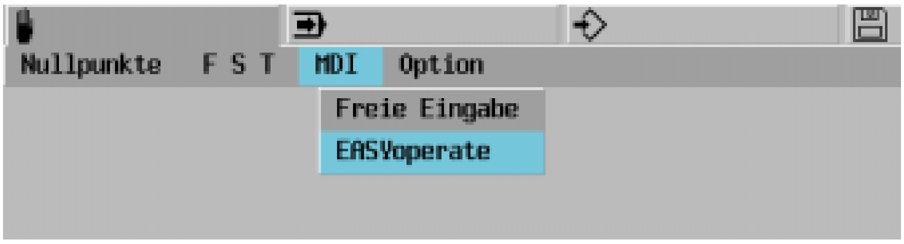
* = S und F müssen vorher aktiviert werden, mittels F,S,T oder MDI

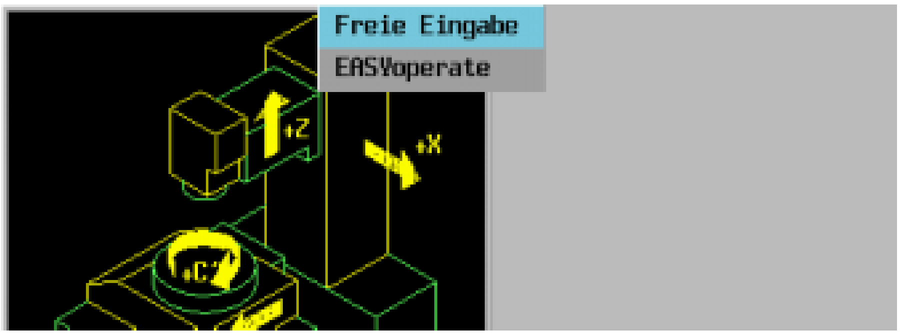
Notes:

- Facing
If C1 is not programmed the infeed width is 75% * Tooldiameter.
If Tooldiameter = 0, minimum value = 0.1.

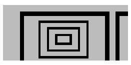
- Edging
The infeed width is 75% * Tooldiameter.
If Tooldiameter = 0, minimum value = 0.1.
The finishing allowance is 10% * toolradius.

19.3 Pockets

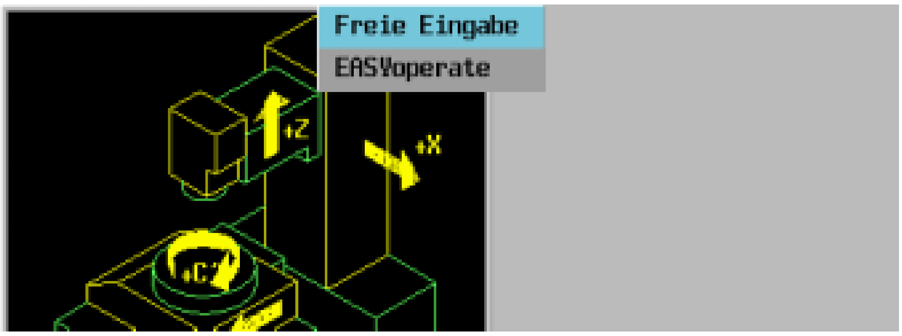




optional



,

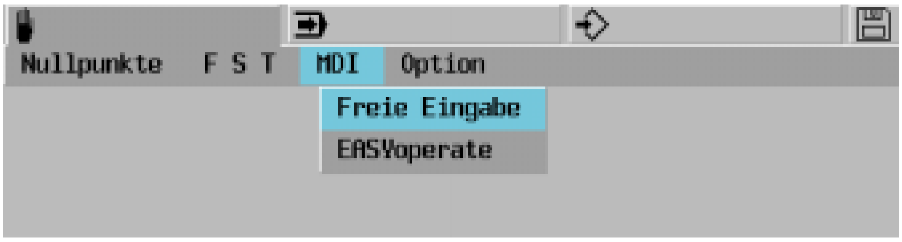


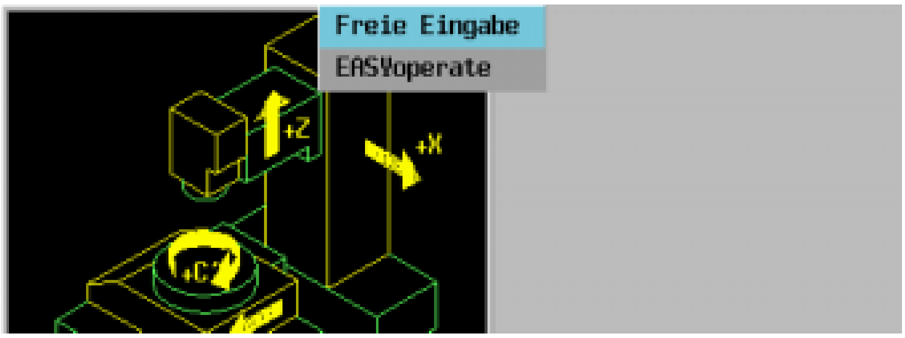
S and F must be activated in advance, by means of F,S,T or MDI

Note:

If C1 is not programmed, the infeed width is set to the value of machine constant MC720.
If the centre point (X1,Y1) has not been programmed, the tool position (X,Y) becomes the centre point of the pocket.

19.4 Drilling / tapping





Parameters

X1,Y1	Punktkoordinate (wahlweise)
C2	Reduzierwert Zustelltiefe (wahlweise)
Z2	Tiefe
S	Spindeldrehzahl (U/Min) *
F	Vorschub *

* = S and F must be activated in advance, by means of F,S,T or MDI

Notes:

- General
If the (Tap)Hole Position (X1,Y1) is not programmed, the Toolposition (X,Y) is the position of the tap(hole).

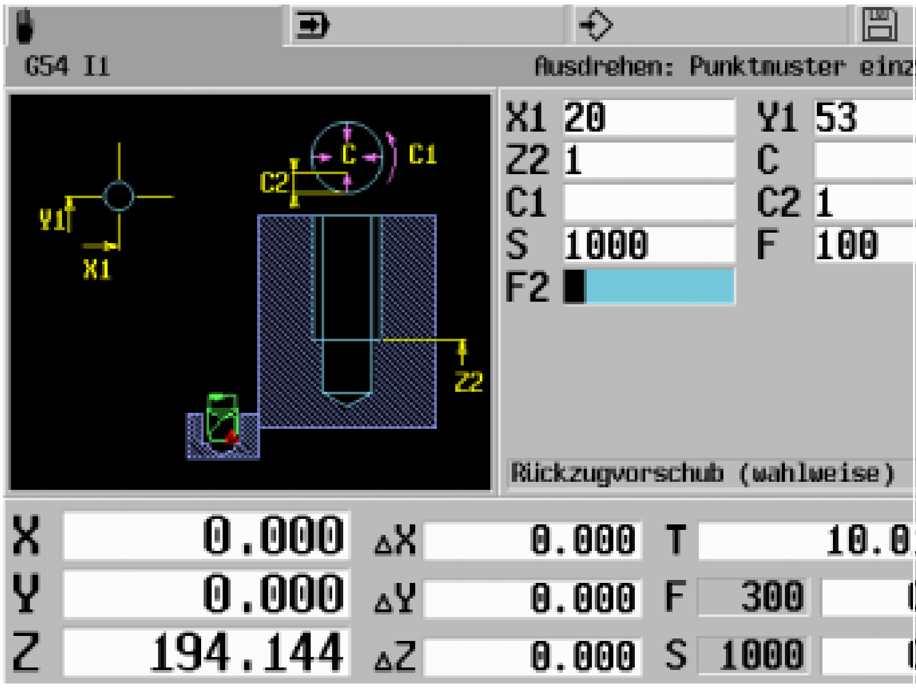
- Circle Pattern
If the Angle between the first/last Hole (A2) is not programmed, the holes will be equally divided over 360 degrees.

- Taphole
If F1 is not programmed, the feed of the tool is F.

- Rectangular Pattern
If the Angle between the first/last Hole (A2) is not programmed, the angle is 90 degrees.

19.5 Boring

19.5.1 Boring: single

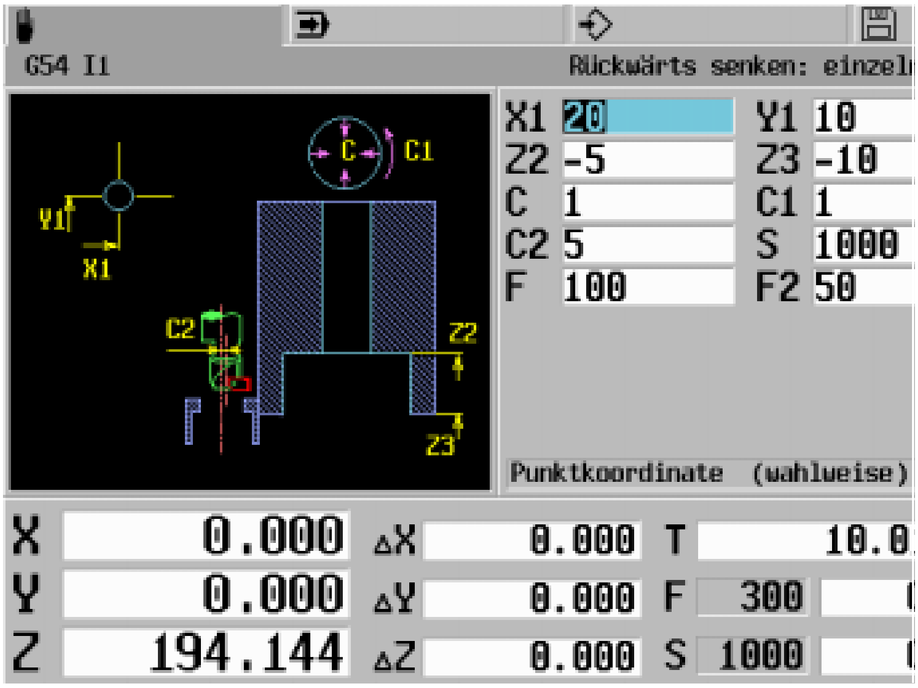


Boring point:

X1	point coordinates (according to choice)
Y1	point coordinates (according to choice)
Z2	depth
C	return traverse distance 1=X 2=Y (according to choice)
C1	1=0 2=90 3=180 4=270 (according to choice)
C2	safety distance (according to choice)
S	spindle speed (revolutions/min)
F	advance
F2	programmable power traverse (according to choice)

19.6 Reverse sinking

19.6.1 Reverse sinking point



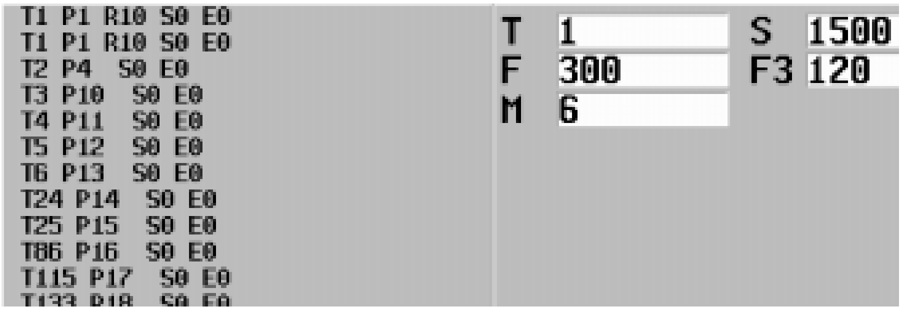
Reverse sinking point:
X1 point coordinates (according to choice)
Y1 point coordinates (according to choice)
Z2 depth
Z3 depth of sinking
C Reverse traverse distance 1=X 2=Y (according to choice)
C1 1=0 2=90 3=180 4=270 (according to choice)
C2 {Excenter} dimension
S Spindle speed (revolutions/min)
F Advance
F2 Programmable power traverse (according to choice)

19.7 Example of Easy Operate: Mill workpiece face

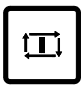
Enter in tool memory
T1 L50 R10
T2 L60 R3

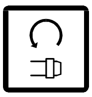
Step 1

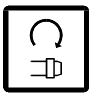
Enter T1, S1500, F300, F2=120, M6



Initiate tool change



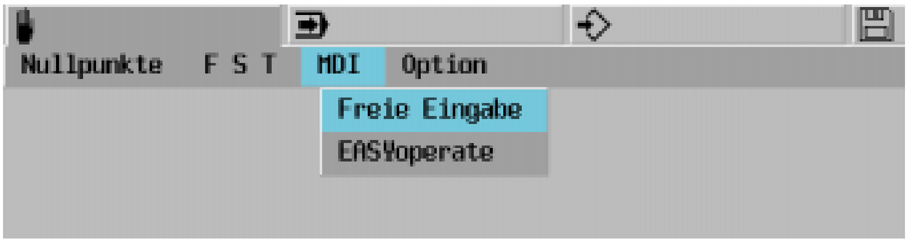


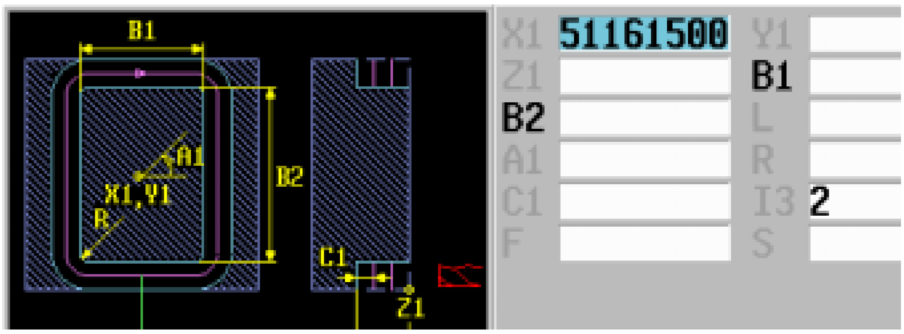


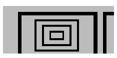
Switch on spindle (M3 or M4)

Step 2

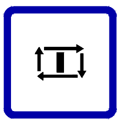
Move to starting point: X-15, Y-15, Z0



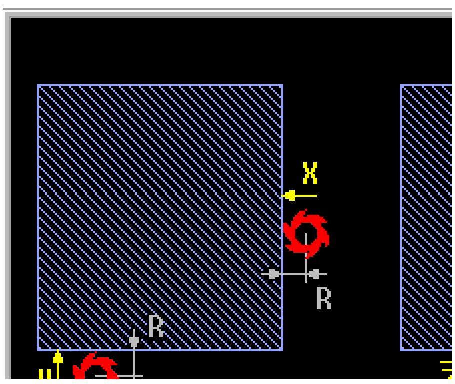




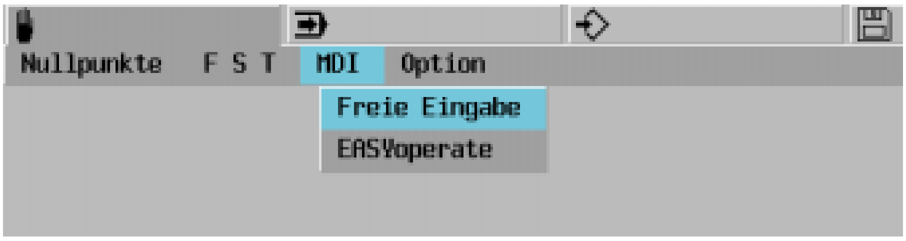
optional

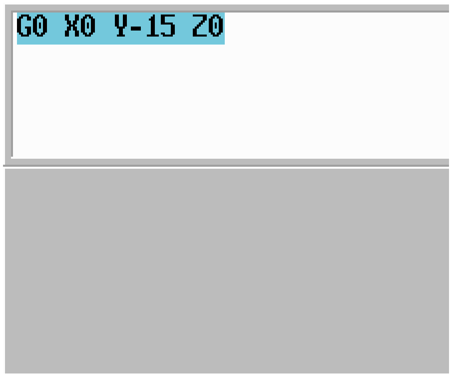


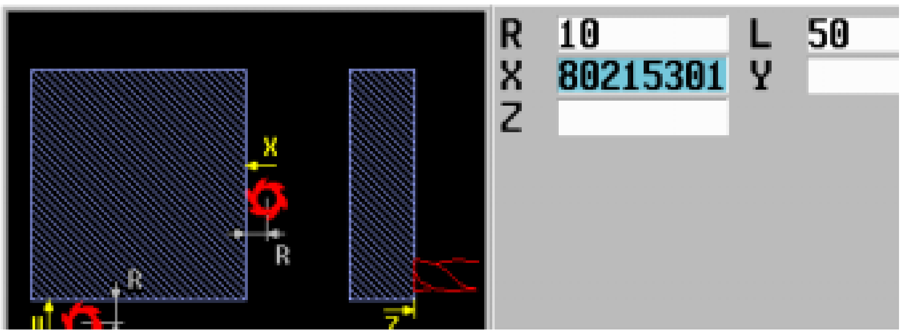
easyfxp.sp

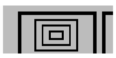


Step 3

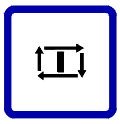


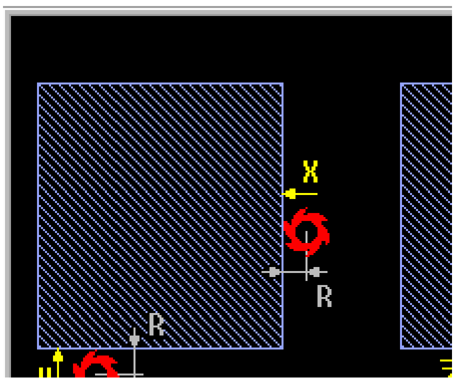






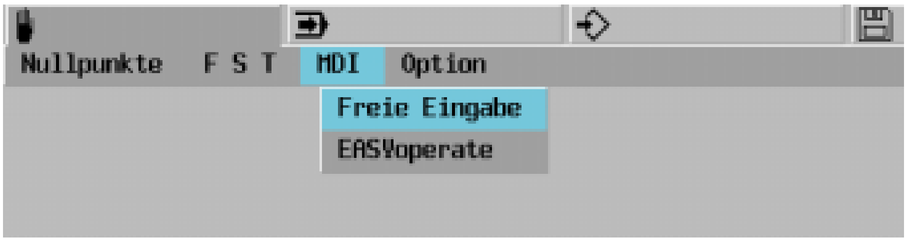
optional

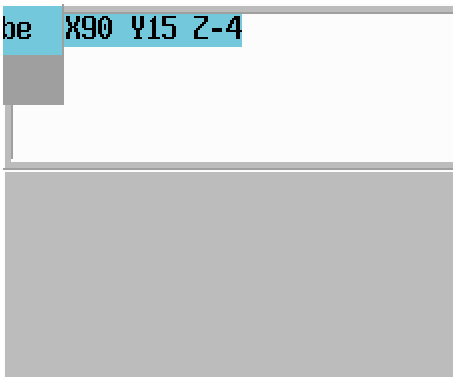


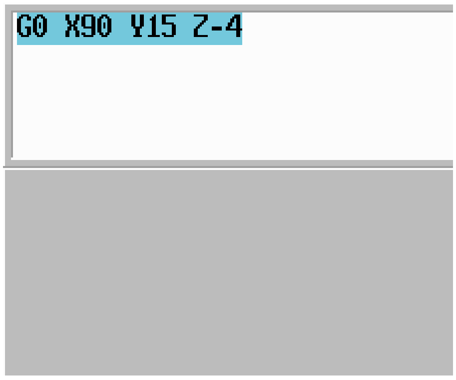


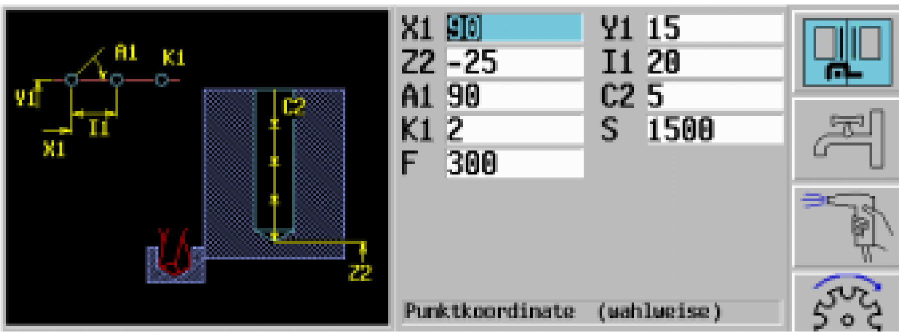
Step 4 Tool change.
See Step 1 for input T2, S1500, F300, F2=120, M6.
Start spindle.
Move to correct Z position.

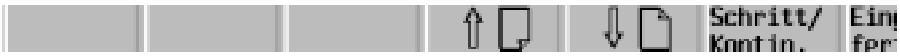
Step 5
Move to first drill position: X90, Y15, Z-3
Enter C2=5, Z2=-25





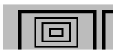




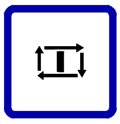


Select parameter X1

Select parameter Y1



optional



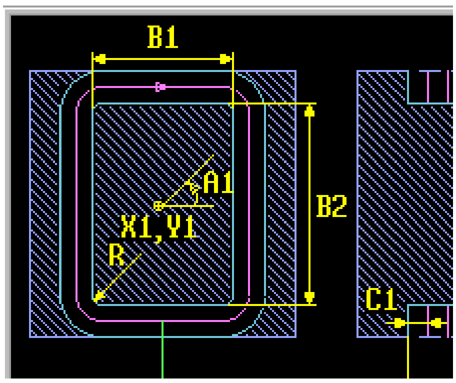
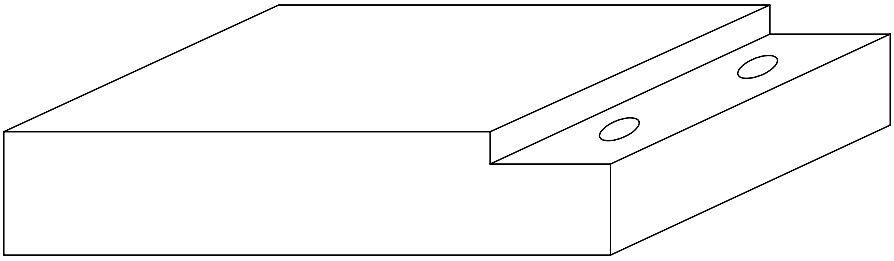


Figure: machined workpiece



20. Interactive contour programming (ICP)

20.1 Allgemeines

ICP can be used with new or existing main programs or macros.

ICP can be used with DIN/ISO and IPP.

The programmer begins at a certain point of the contour and processes the entire workpiece in either a clockwise or anticlockwise direction, each contour being described as a linear or circular movement. Following this initial selection other options are offered until the movement is defined. You are then requested to enter positional data.

Using ICP each contour is drawn as soon as its position is known, to be precise once the Store key is pressed. However, this does not always have to be the case. If a contour cannot be classified immediately it is joined to the following contour until sufficient positional data is available to calculate its exact position.

20.2 ICP graphic symbol menu

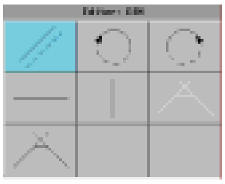
ICP has a dynamic menu structure. Options are enabled or blocked depending on the previous option selected.

Mittelpunkt

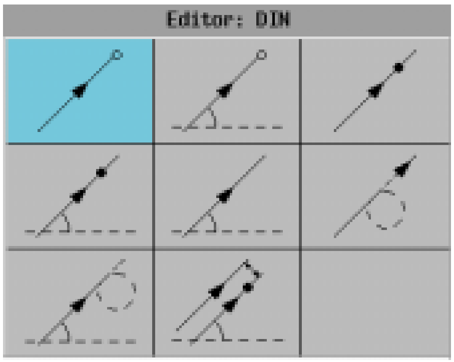
Endpunkt

Hilfspunkt

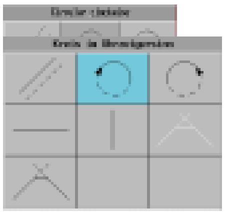
Main menu level

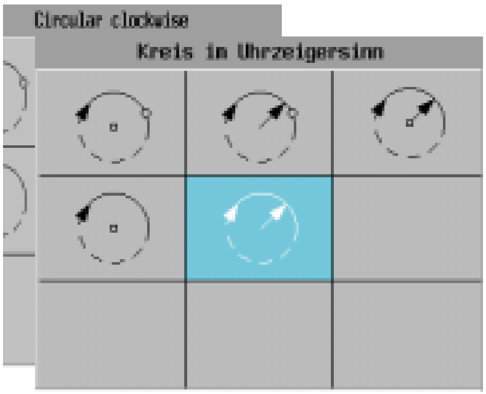


Menu for linear movement

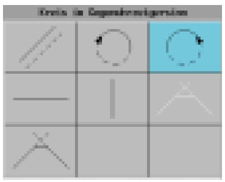


Menu for circular movement in a clockwise direction

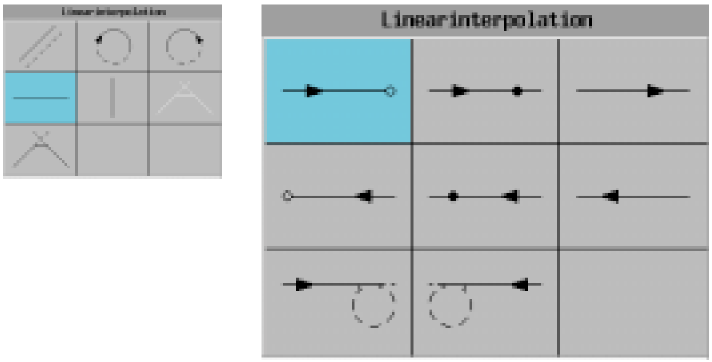




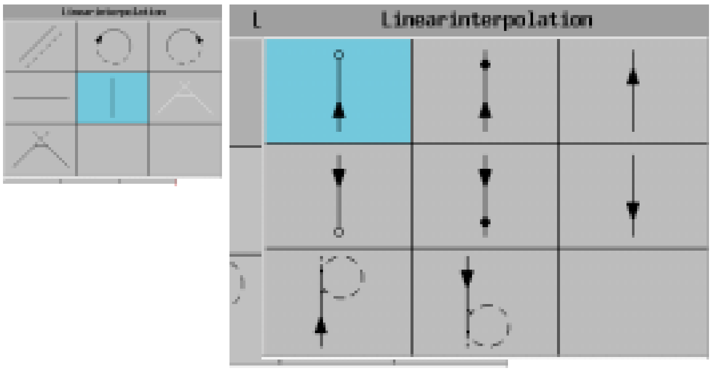
Menu for circular movement in an anti-clockwise direction



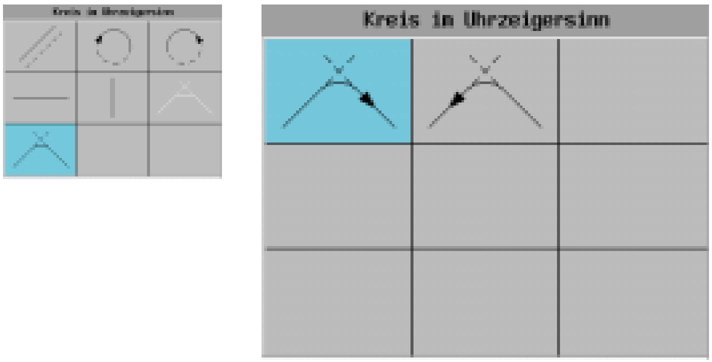
Menu for linear movement horizontally



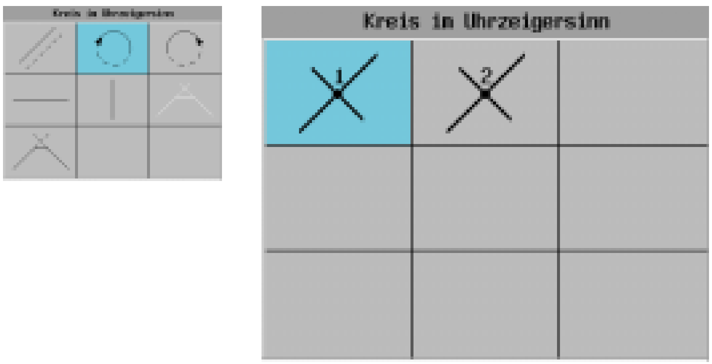
Menu for linear movement vertically



Menu for rounding

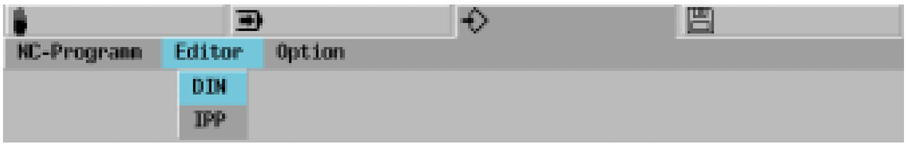


Menu for point of intersection

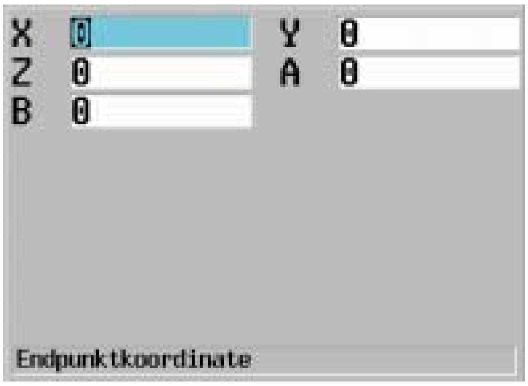


20.3 New ICP programs

20.3.1 Einstieg in den ICP-Modus



New programs may be completely empty apart from the header line. In this case the programmer is prompted to enter a starting point.



Enter a value for all parameters specified, even if this value is 0.

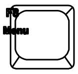
Note

No allowance is made in ICP for a pole position predefined with G9. G9 must be deselected before ICP.

20.3.2 Exiting ICP

ICP beenden durch Betätigen des Softkeys.

or



It is possible to exit the ICP INPUT mode at any time during data input, although exiting ICP during contour programming may result in an error message when ICP is re-entered.
The program line or lines affected must then be found and deleted.

20.4 Editing existing programs

When using an existing program, the cursor is positioned at that point in the program at which ICP is to commence.

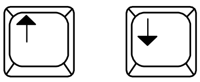
Using the cursor keys, scroll upwards and downwards through the program. The relevant contour section is shown white in the graphic window.

ICP scans the program section before the cursor position for a G64 function without G63 (the cursor is in an ICP section in the program). If the cursor is located outside a G64-G63 area, these ICP G-functions are situated in successive program lines.

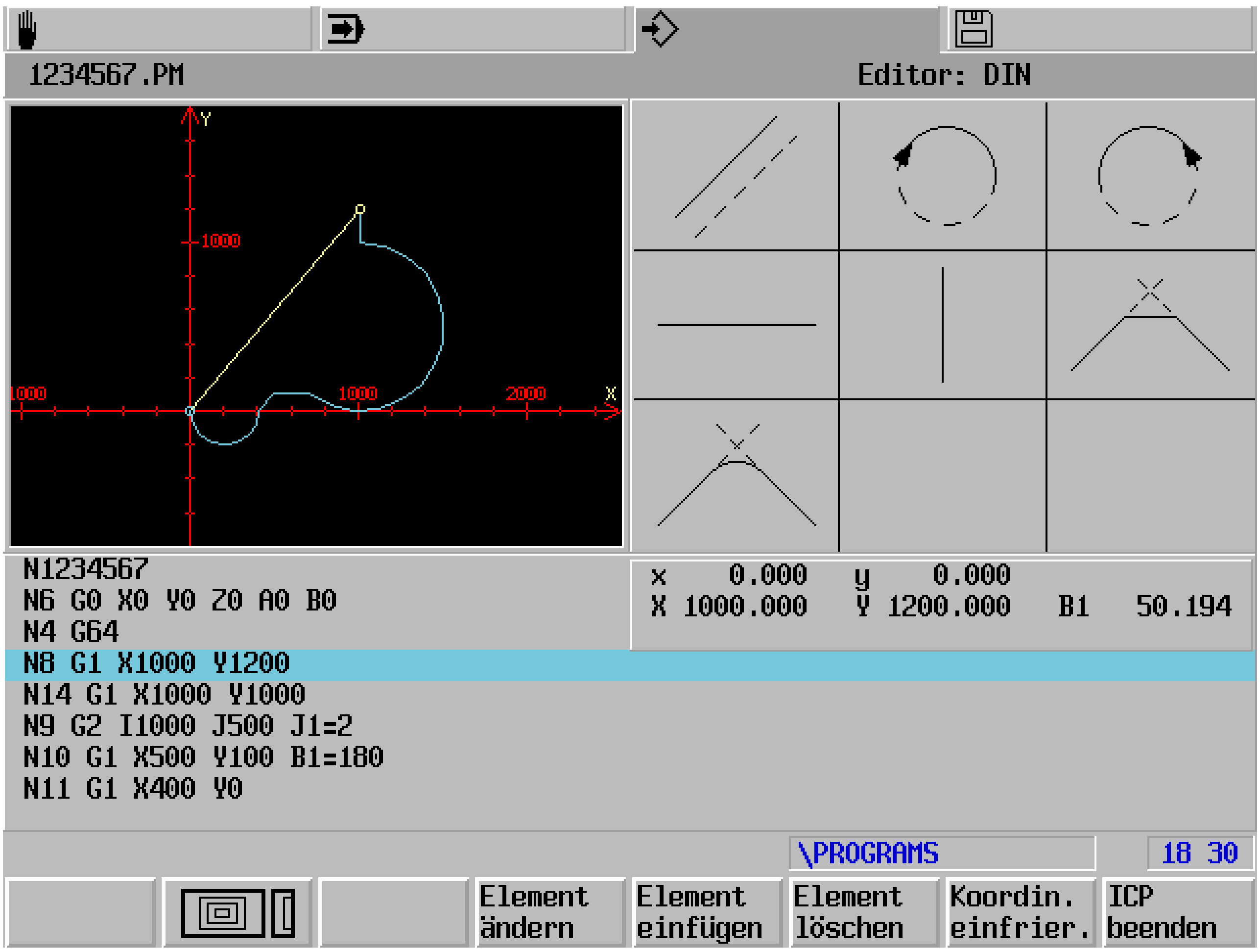
The program is checked in advance as to whether at least a feed movement has been programmed for the addresses of the main plane. If not, the user is requested to enter a traversing movement.

20.4.1 Edit element

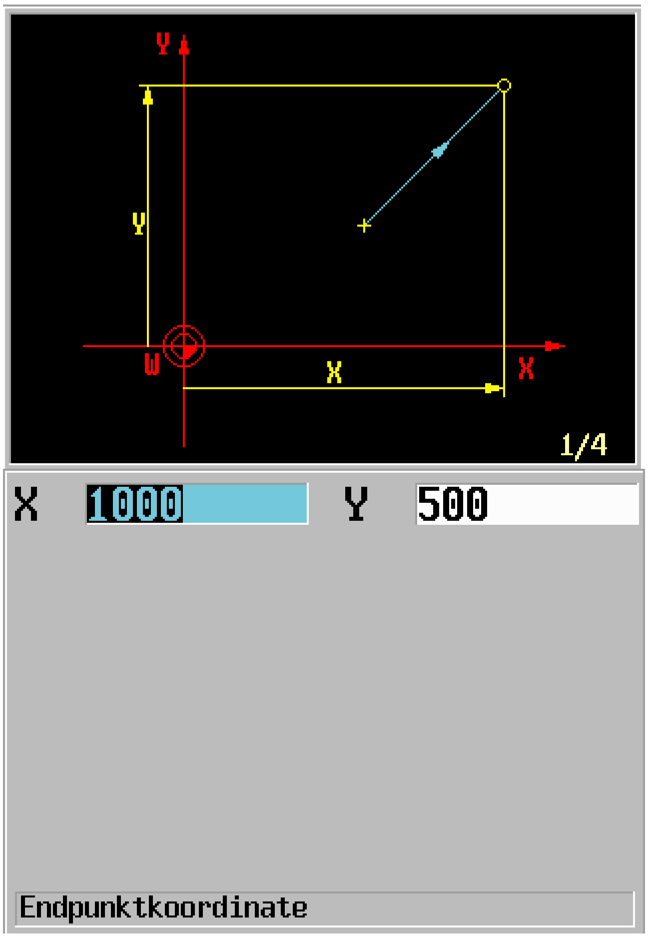
Select ICP.



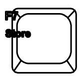
Select program block, e.g. N8.



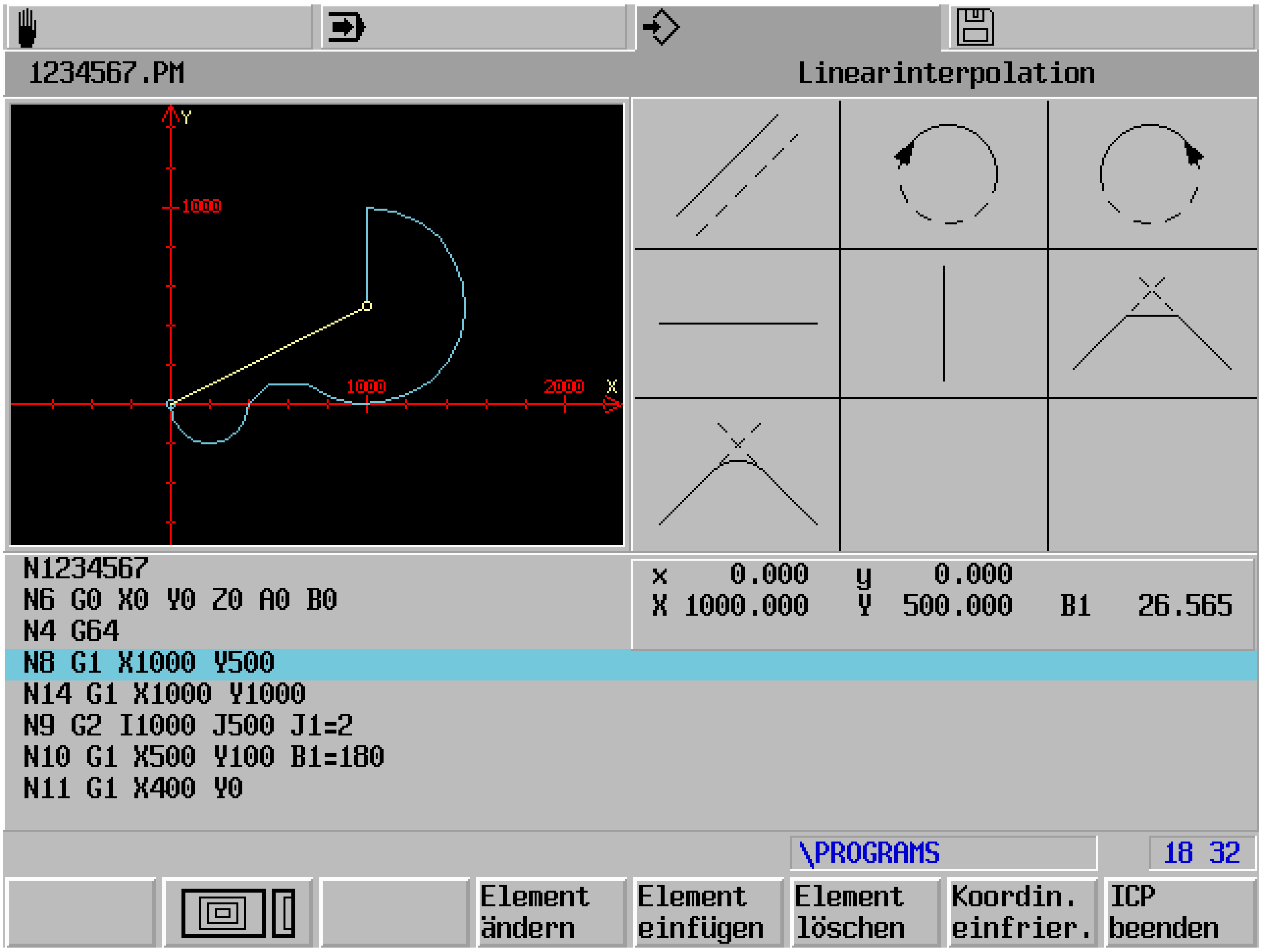
The contour element can be defined differently,
e.g. an address value can now be changed.
Enter addresses.



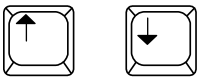
or



The element is stored and the contour recalculated and displayed.



Have all the changes in change mode been implemented?
No?



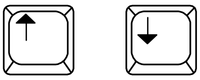
Next element.

Yes?

Note

For specific elements (rounding circles) there are additional solution variants. These variants can only

20.4.2 Insert element



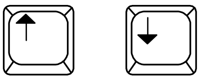
Insert contour element / select block

Note:

For specific elements there are several input options:

Selection of the options

20.4.3 Delete element

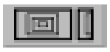


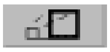
Select the contour element / block to be deleted

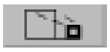
Note

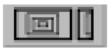
By deleting, changing or inserting elements, it is possible to create non-continuous contours. The modified element or subsequent elements are shown as dotted lines.

20.4.4 Graphial display of the contour



 Zooming out

 Zooming in

 Original size

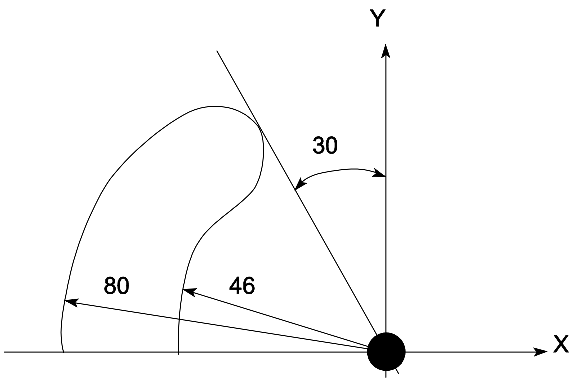
20.5 ICP programming notes

20.5.1 Auxiliary elements in ICP

Lines and circles may be defined by auxiliary elements, for instance tangents or circles. Missing coordinates or angles can be calculated with the auxiliary elements. These calculated values are always displayed for every element.

These calculated values are saved by the "Freeze coordin." softkey. The auxiliary elements may then be deleted and the required circle or straight line can be re-entered.

Example:



N100 G0 X-80 Y0	Starting point
N101 G64	Select ICP
N102 G2 I0 J0	Circle with centre
N103 G2 R17	Rounding (clockwise)
N104 G1 X0 Y0 B1=-60	Auxiliary line with end point and angle, select intersection 2

- Place cursor on block N103.

- Display:	x -57.211	y 55.918		Starting point (lower case)
	X -30.332	Y 52.536		End point (capital)
	I -45.054	J 44.036	R17	Centre and radius

- Save these coordinates by pressing F7 "Freeze coordin.".

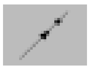
- Delete auxiliary line N104 and circle N103.

- Re-enter program blocks N103 (circle with centre) and N104:

N103 G2 I-45.054 J44.036	Circle (clockwise) with centre
N104 G3 X-46 Y0 R46	Circle (counterclockwise) with end point and radius
N105 G63	

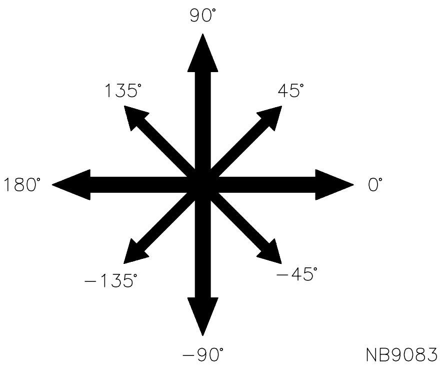
20.5.2 Help points

The "Help point" programming option in ICP offers a simple solution to the problem of defining axis end points in complex contours. The option is used when the axis end point is unknown. As soon as the axis end point is determined by the next movement or those following on from it, it is classified.



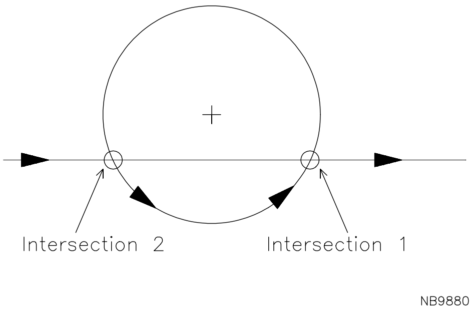
20.5.3 Required angle parameters

Some of the linear interpolation movements call for an angle parameter (specified relative to the horizontal).



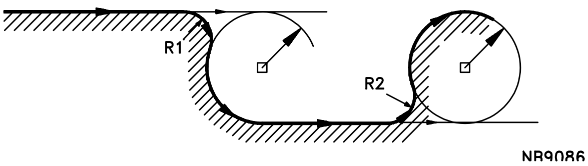
20.5.4 Line intersects circle

ICP draws the line which passes through the circle and the points of intersection (1 and 2) are marked. The programmer is requested to select the correct point of intersection.



20.5.5 Rounding

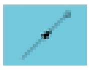
The movement preceding the rounding may be designed in any manner, including with an endpoint. The rounding is specified purely as a radius. Its position and its start and end point are calculated by ICP as soon as sufficient data is available to classify it.



20.6 ICP programming example

First create a new program N111111 with the start point X0, Y0, Z0.

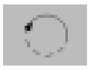
L1

X0
Y=12.7

Enter, Store

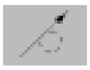
C1

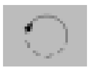
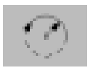
I=12.7
J=12.7

Enter, Store

L2

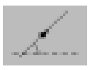
C2

I = 76.2
J = 63.5
R = 7.94

Enter, Store

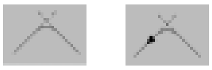
L3

B1 = -135

Enter, Store

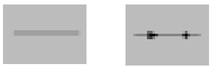
C3



R = 10

Enter, Store

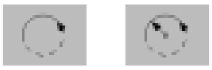
L4



X = 120
Y = 19.05

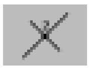
Enter, Store

C4

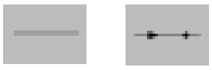


I = 96.2
J = 25
R = 12

Enter, Store

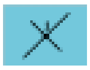


L5

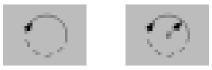


X = 120
Y = 19.05

Enter, Store

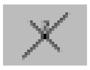


C5



I = 114.3
J = 6.35
R = 12.7

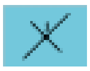
Enter, Store



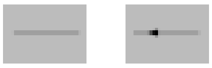
L6

X = 120.65
Y = 0
B1 = -135

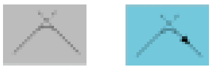
Enter, Store



L7



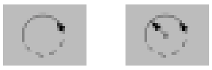
C6



R = 1

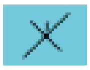
Enter, Store

C7

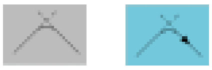


I = 38.1
J = 0
R = 10

Enter, Store



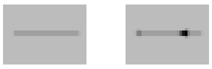
C8



R = 1

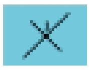
Enter, Store

L8



X = 0
Y = 0

Enter, Store



20.6.1 ICP-generated program

N111111 (ICP-generated program)
N1 G0 X0 Y0 Z0
N2 G64
N4 G1 X0 Y12.7
N5 G2 I12.7 J12.7 R1=0
N6 G1 R1=0

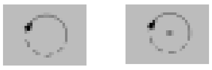
```
N7 G2 I76.2 J63.5 R7.94 R1=0
N8 G1 B1=-135
N9 G3 R10
N10 G1 X120 Y19.05 B1=0 I1=0 J1=2
N11 G3 I96.2 J25 R12 J1=1
N12 G1 X120 Y19.05 B1=0 I1=0 J1=2
N13 G2 I114.3 J6.35 R12.7 J1=1
N14 G1 X120.65 Y0 B1=-135
N15 G1 B1=180 J1=1
N16 G2 R1
N17 G3 I38.1 J0 R10 J1=1
N18 G2 R1
N19 G1 X0 Y0 B1=180
N3 G63
```

20.6.2 Alternative ICP programming methods

The previous example showed only one possible method of programming the individual movements. The same result can be achieved in several ways. The various options for programming a line 1 and circle 1 are shown below:

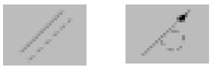
X = 0
Y = 12.7

N4 G1 X0 Y12.7
N5 G2 I12.7 J12.7 R1=0



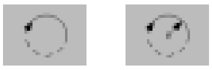
I = 12.7
J = 12.7

1. Line as tangent

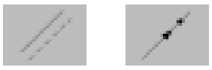


I = 12.7
J = 12.7
R = 12.7

N4 G1 R1=0
N5 G2 I12.7 J12.7 R12.7 R1=0

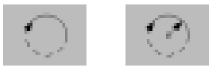


2. Line with help point

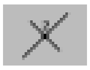


X = 0
Y = 10

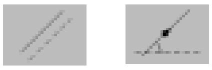
N4 G1 X0 Y10 I1=0 J1=2
N5 G2 I12.7 J12.7 R12.7 R1=0



I = 12.7
J = 12.7
R = 12.7

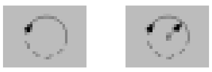


3. Line with angle

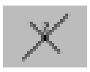


B1 = 90

N4 G1 B1=90 J1=2
N5 G2 I12.7 J12.7 R12.7 R1=0



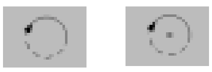
I = 12.7
J = 12.7
R = 12.7



4. Line, vertical

Y12.7

N4 G1 Y12.7 B1=90
N5 G2 I12.7 J12.7



I = 12.7
J = 12.7

21. Interactive part programming (IPP) / GRAPHIPROG

21.1 General

21.1.1 Introduction to Interactive Parts Programming (IPP)

When using interactive parts programming you must select from a number of features and machining strategies to create a program. For the most part no knowledge of DIN programming is assumed.

IPP technology proposals are derived from the information in the technology database. The information stored therein is based on your own experience in the workplace. Please refer to the chapter on technology.

Each feature begins with a block, which contains the feature description and an identification. You can switch from IPP to DIN programming at any time.

The machining sequence can be simulated at any time during the creation of a program.

21.1.2 Preparation for IPP programming

- The technology tables must contain suitable data.
- The IPP start macro must contain the right data (see 21.8)

Note

- Always make sure that the retract movement of the tool axis in parameter E714 is large enough to avoid a collision between tool and workpiece or fixture.

- The tool table must display a list of the tools used most frequently.

- If there is no suitable tool in the tool table, IPP will generate a new tool in this table.
  All tools created with the aid of IPP should be entered in the tool table.
  During simulation M6 is changed to M67.

21.1.3 IPP programming sequence

The procedure for programming a new program in IPP is described below:

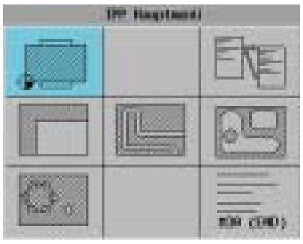
1. First define a blank.

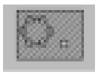
2. You also have the option of defining the type of workpiece clamping device to be used.

3. Program the workpiece with the aid of IPP features.

4. Select the M30 feature to conclude the program.

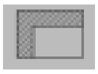
21.2 IPP graphic main menu symbols



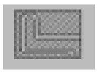


Drilling operations

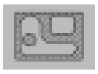
End of program



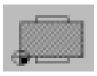
Facing and edge milling



Contour input, thread milling



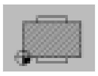
Pocket with or without islands

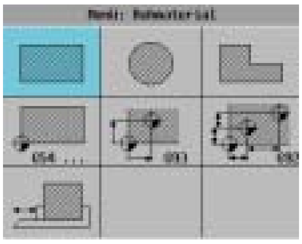


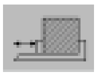
Invoke macro or main program

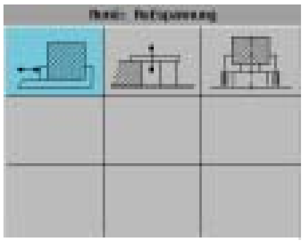
Set-up (material, zero points and clamping)

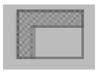
21.3 IPP graphic symbol menu

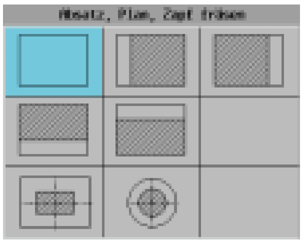


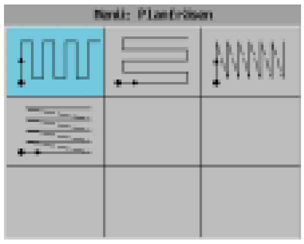


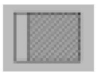


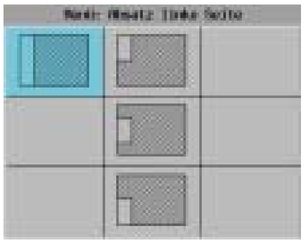


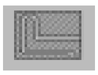


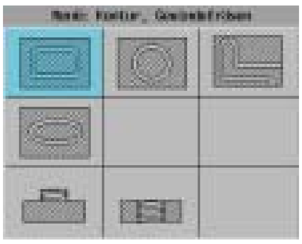


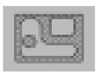


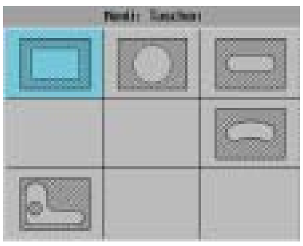


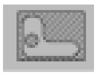


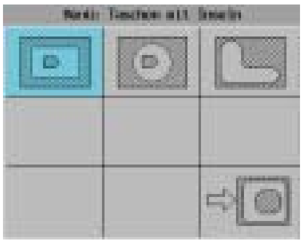


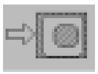


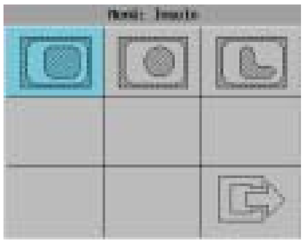


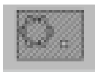


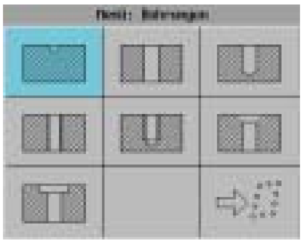


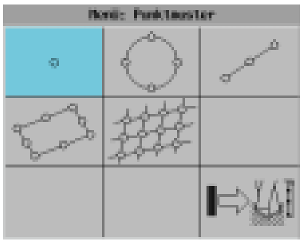


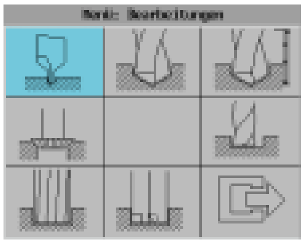








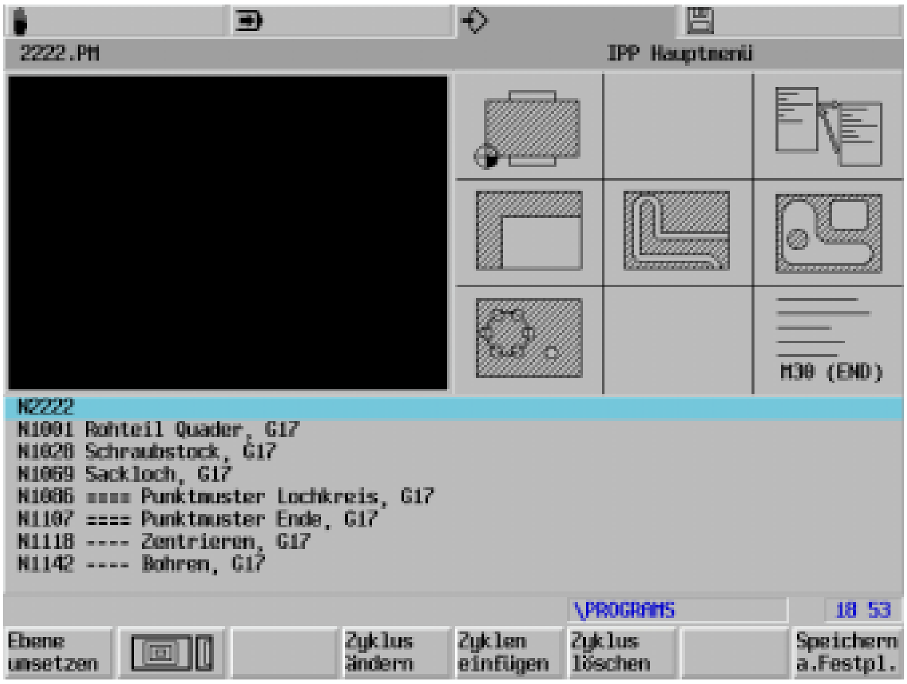




21.4 New IPP-program

21.4.1 Entering IPP mode

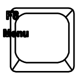
Select program



Note

If it is impossible to access IPP, it should be checked whether the reference point has been approached in all axes or whether G19, G91, G182, G201, G64 or G199 is active.

21.4.2 Exiting IPP

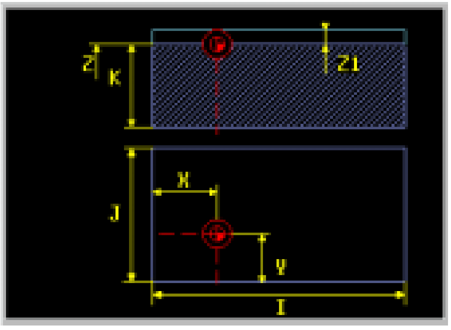


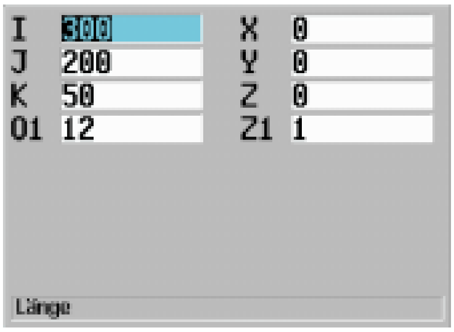
Exit IPP.

Note

Exiting IPP during programming will result in an incomplete program.

21.4.3 Entering program data



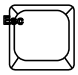


Once a machining operation has been defined by means of Feature, the data entry window appears with the addresses required for complete definition.
A value must be entered for each address. A value is already suggested for many addresses.

Storing the input values and displaying the next data entry.

Storing the input values and exiting the data entry.

Note

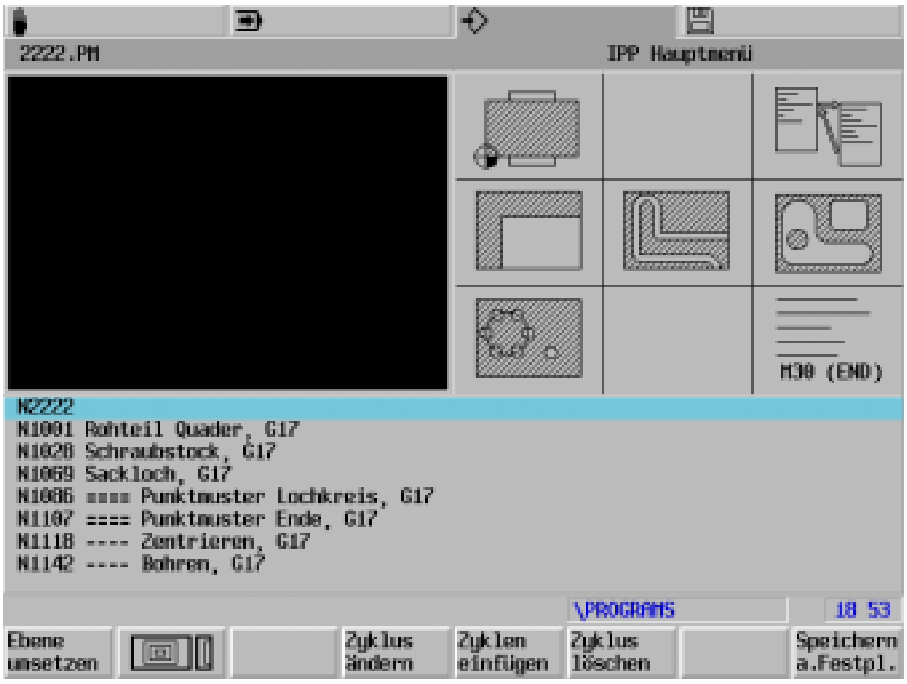


Return without saving data.

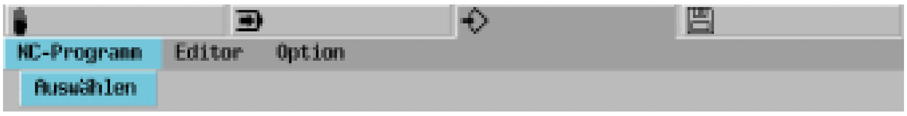
Exiting the data input mode during programming often results in an incomplete program.
The relevant feature must then be deleted and reprogammed.

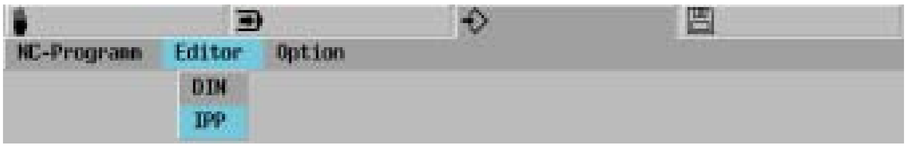
21.4.4 IPP program list

The program window only displays the names of the features used in the parts program.

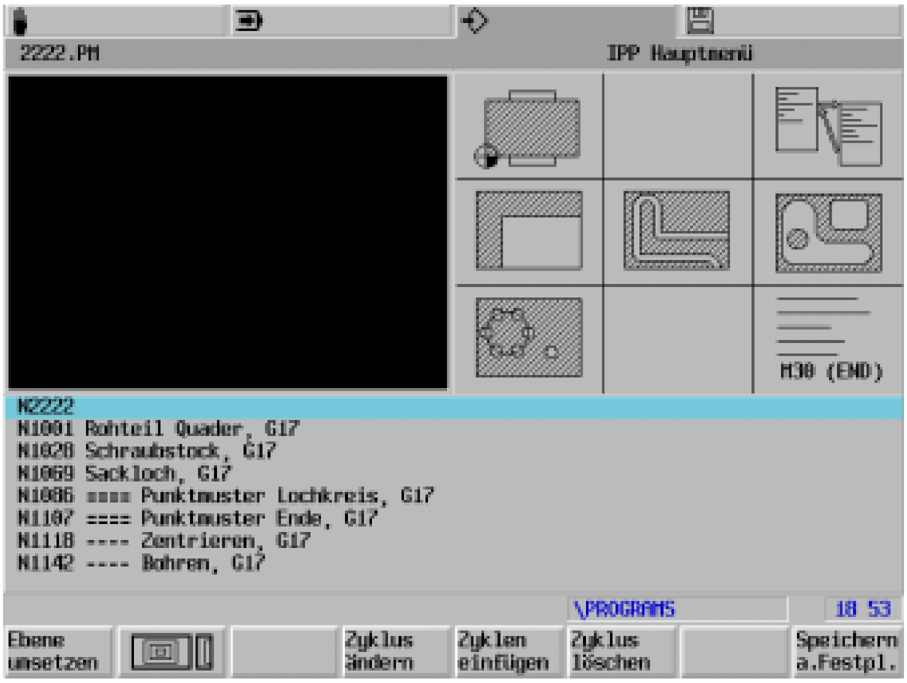


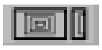
21.5 Editing IPP programs (change line)

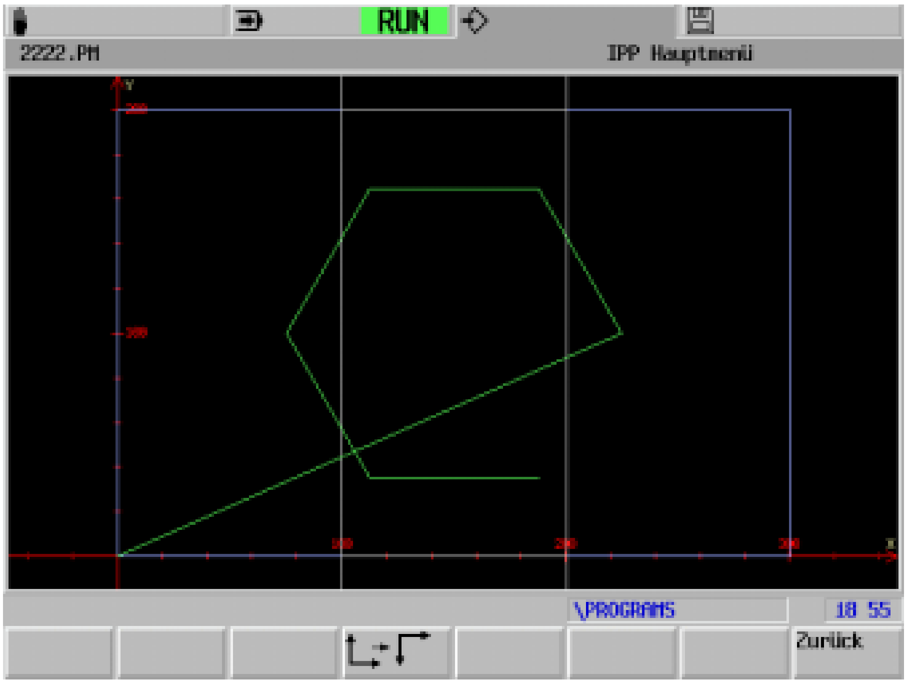


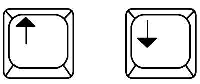


21.5.1 Change features



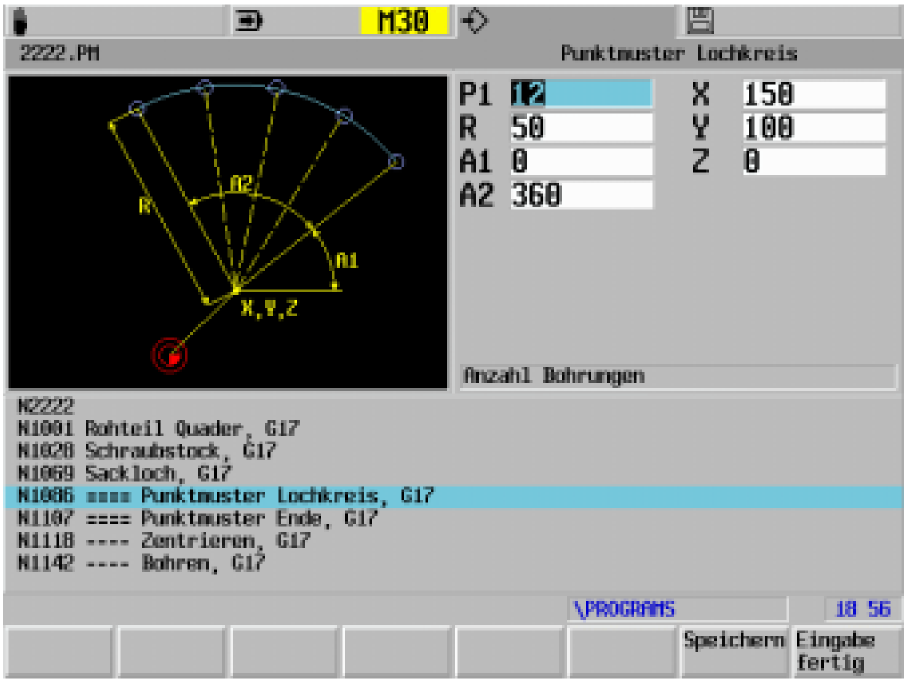




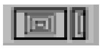


Select the feature to be changed.

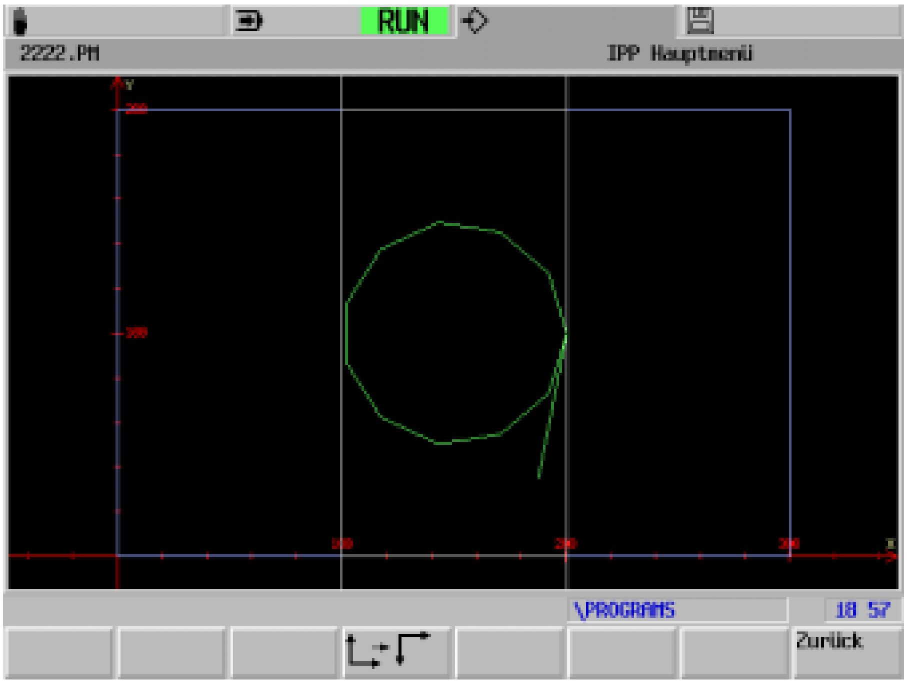
The feature can be redefined.
For example, it is now possible to change an address value.
Enter address values.



The feature is directly generated.

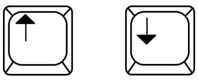


Checking changes with graphic.



Have all changes in the program been made?

If not, select next feature.



Next feature.

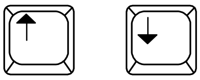
Note

If a feature is changed within an IPP program block, the entire IPP program block

must be run through. Changes that have been made will be accepted in the subsequent features of the IPP program block.

21.5.2 Inserting a feature

When an IPP feature is inserted, the feature is inserted after the position indicated.



Select insert position of feature.

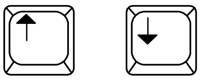
Defining a feature and entering program data.

Note

Macro number 8000 is suggested for pocket milling. If the macro number already exists, change the number.

21.5.3 Delete feature

When deleting an IPP feature, all the accompanying instructions in the program are deleted.



Select the feature to be deleted.

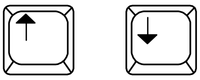
The feature to be deteted is cleared immediately.

21.5.4 Select tool during editing

```
P0
P1 T1 L0 R3.3 G1 Q3=1
P2
P3
P4 T2 L0 R8 G2 Q3=1
P5
P6
P7
P8
P9
P10 T3 L40 R2 G31 Q3=17 Q4=2
P11 T4 L40 R0.98 G31 Q3=17 Q4=2
P12 T5 L60 R2.25 G31 Q3=17 Q4=2
P13 T6 L60 R3 G31 Q3=17 Q4=2
```

D	10	M	6
T	201	M1	1
W4	2	F	41
B1	118	S	255
L	55.004		
W1	2		
W2	0		
X	0		

Durchmesser

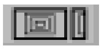


Select tool.

Copy tool in dialogue box.

21.5.5 Graphic display of contour (test run)

Check the parts program briefly for the right sequence and to see whether it is correct.



grafwin.sk

Return to input.

21.5.6 Executing IPP programs

Before executing a parts program the operator must:

Enter all the tools created with the aid of IPP into the magazine and the current tool table.

21.5.7 Converting active plane G17 <-> G18

Programs in IPP are basically made in active plane G17 (XY plane). If the machining operation on the machine has to be carried out in active plane G18 (XZ plane), the program must first be converted from G17 to G18. Reconversion is possible.
Editing is also only possible in G17.

21.6 IPP programming tips

21.6.1 Using ICP to define contours

Once one of the options for the freely designed pocket contour or the contour recess has been selected, ICP is loaded automatically.

The program is checked beforehand to see whether a traversing movement has been programmed at least for the X- and Y-axes. If not, the user is requested to enter a traversing movement.

21.6.2 IPP proposals

The proposals generated during data entry in IPP are based on the table data stored in the CNC (tool and technology tables) and on a special IPP start macro. The proposals made in the IPP start macro can be adapted to suit individual requirements.

21.6.3 Maximum feed rates and spindle speeds

The feed rates and spindle speeds proposed in IPP operation are calculated from the data contained in the technology tables. If the limitations of the machine tool used are not taken into account in this process, the possibility exists that the proposed feed rates and spindle speeds will exceed the maximum values applicable to this machine tool.

For this reason the data stored in the technology tables should take account of the limitations of the machine tool used.

The machine constant memory contains the maximum allowable values of the feed speeds and spindle speeds for this machine tool.

21.6.4 Optimizing programming and machining times

1. Centre drill hole, change tool and drill. Repeat operation for each drill hole.
2. Centre all drill holes, change tool and finish all drill holes.

Note

Always decide on the optimization strategy **before** IPP programming, never afterwards!

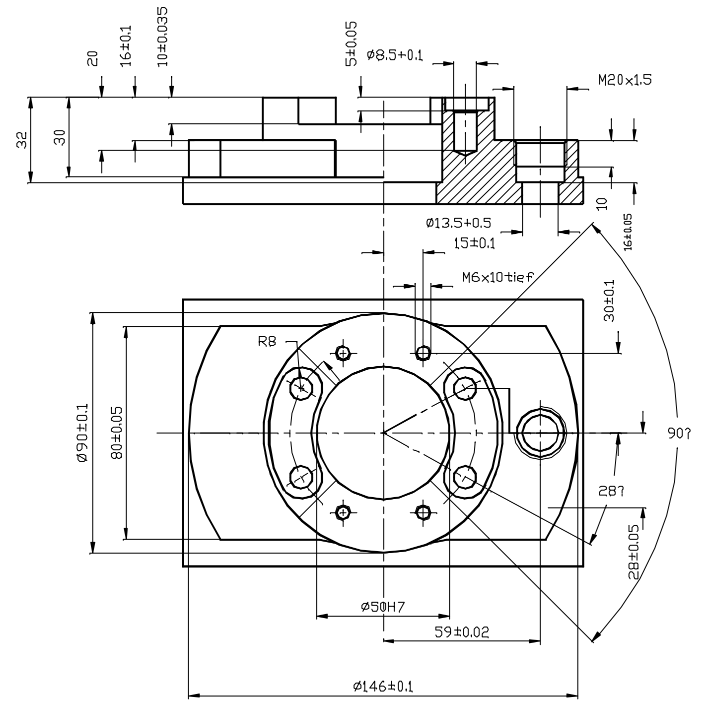
21.6.5 Changing IPP programs using the DIN editor

We would like to advise you to change all IPP-generated programs with the aid of IPP. If this should not be possible or desirable, then it is easy to change programs manually thanks to the standard DIN code programs generated by IPP.

Program changes which are made manually are lost if a feature which has been changed manually is later modified in the IPP mode "Change line", to be precise for the reason that IPP deletes the entire feature and regenerates it.

21.7 IPP program examples

This example discusses some features of IPP. The workshop drawing shown below represents a workpiece whose manufacturing sequence is determined by the programmer.



The following sequence was selected for this example:

-	Definition of blank	21.7.2
-	Clamping	21.7.3
-	Facing	21.7.4
-	Tapping a rectangle	21.7.5
-	Milling freely formed pocket	21.7.6
-	Milling circular groove	21.7.7
-	Milling round pocket (for thread)	21.7.8
-	Milling round pocket (50 mm dia.)	21.7.9
-	Milling freely formed contour	21.7.10
-	Boring and countersinking (8.5 mm dia.)	21.7.11
-	Boring and threading (M6)	21.7.12
-	Threading (M20 x 1.5)	21.7.13

21.7.1 Preparations for programming the examples

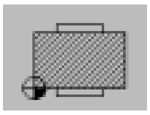
To complete the present example, the tool table should at least contain the following tools:

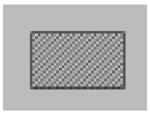
-	End milling cutter	radius 10 mm
-	Groove milling cutter	radius 5 mm
-	Drill bit	radius 4.25 mm
-	Drill bit	radius 2.5 mm
-	Screw tap	M20 x 1.5
-	Screw tap	M6
-	Centre drill	radius 6 mm

Make sure that the tool table is completely defined, so that the recommended feed rates and spindle speeds are given.
Adoption of the IPP program example requires creation of a new program.

21.7.2 Definition of blank

The dimensions of the blank are: 150 mm, 100 mm, 45 mm.





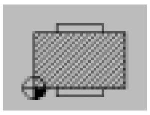
Enter: I150, J100, K45, Q1=12, X75, Y50, Z0, Z1=1

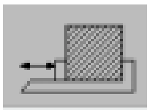
Note

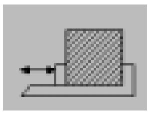
Z1 is the offset of the upper surface for facing.
The zero point is placed at the centre of the blank. The graphic window is based on the co-ordinates and the zero point of the blank.

21.7.3 Clamping

Select the IPP feature for a vice:







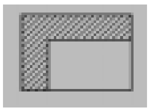
Enter: I1, V100, L34

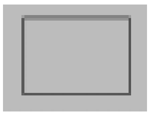
Note

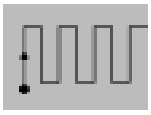
The workpiece should protrude at least 33 mm from the top edge of the vice.

21.7.4 Facing

In this stage the offset (Z1 of the definition of the blank) is removed.







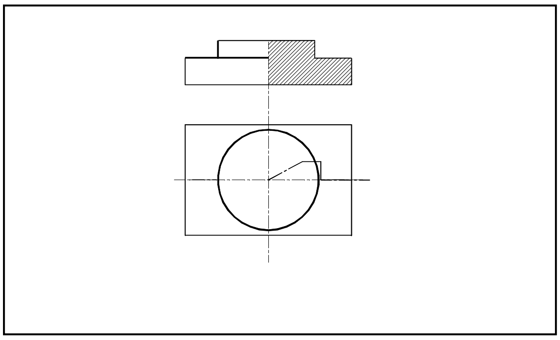
Enter: U150 V100 W1 X0 Y0 Z0

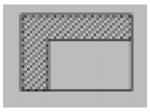
Facing tool R10

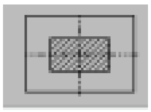
Enter: K1=1 K4=67 W5=10 W1=2 W2=0 F148 S254

21.7.5 Tapping a rectangle

Mill the round cam on the rectangular blank.







Enter: U1=90 V1=90 R1=45 U150 V100 R0 L16 A1=0 X0 Y0 Z0

Enter: A4=2 K2=0.1 K3=0.1 W1=2 K5=1 K4=80 K1=5

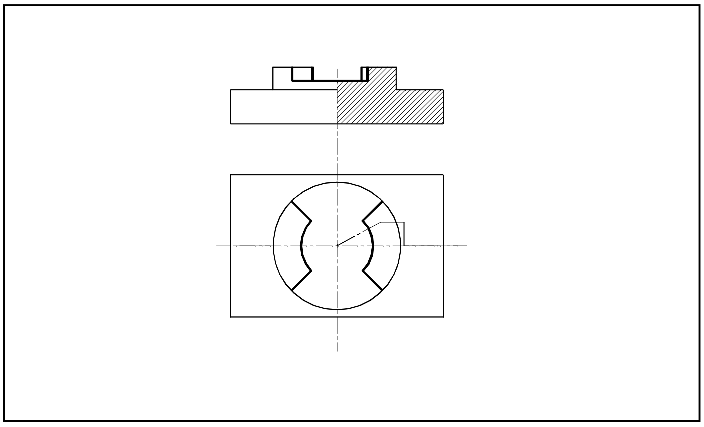
Facing tool R10

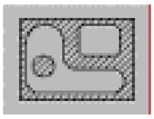
Enter: F148 S254

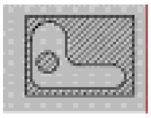
Facing tool R10

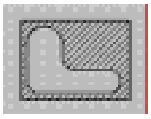
Enter: F148 S254

21.7.6 Milling a freely formed pocket



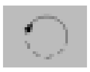




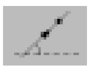


Enter: X0 Y50 Z0 L10

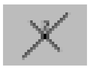
A freely formed contour is created with the aid of ICP.

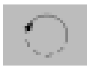
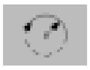
 

Enter: I0 J0

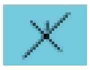
 

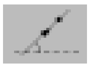
Enter: X0 Y0 B1=-135



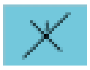
 

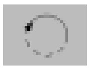
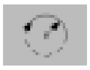
Enter: I0 J0 R25



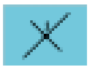
 

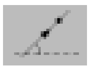
Enter: X0 Y0 B1=-45



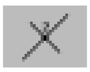
 

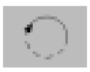
Enter: I0 J0 R50



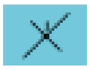
Enter: X0 Y0 B1=45

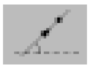


Enter: I0 J0 R25

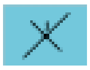
Eingabe fertig

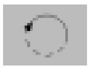


Enter: X0 Y0 B1=135

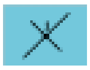
Eingabe fertig



Enter: X0 Y50 I0 J0

Eingabe fertig



Zurück

ICP beenden

Enter: A3=0 A4=1 N=8000 K2=0.2 W1=2 K5=1 K4=50 K1=5

Speichern

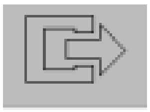
WZ-Nummer auswählen — Groove milling cutter R5

Enter: F90 S509 F1=90

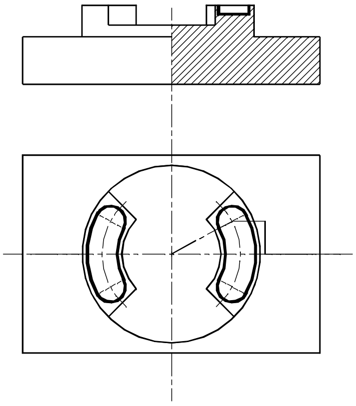
Speichern

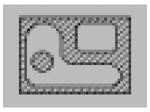
WZ-Nummer auswählen — Groove milling cutter R5

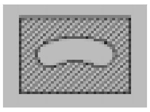
Enter: F90 S509



21.7.7 Milling circular groove







Enter: A1=-28 A2=56 R8 R1=35 L5 X0 Y0 Z0 P1=2 A3=124

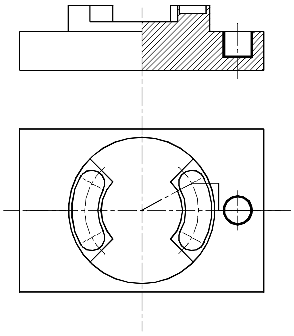
Enter: A1=0 A2=1 A3=0 W1=2 K1=5 K5=1 A5=90 K2=0

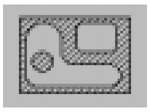
Groove milling cutter R5

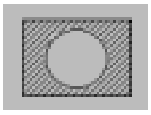
Enter: F90 S509 F1=90

21.7.8 Milling round pocket (for thread)

Mill round pocket, into which the thread (M20 x 1.5) is subsequently cut.



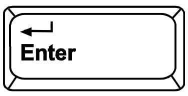




Enter: R9.25 L16 X59 Y0 Z-16

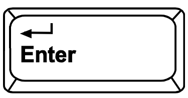
Enter: A3=0 A4=2 A5=90 K2=0.1 K3=0.1 W1=2 K5=1 K4=50 K1=5

Groove milling cutter R5



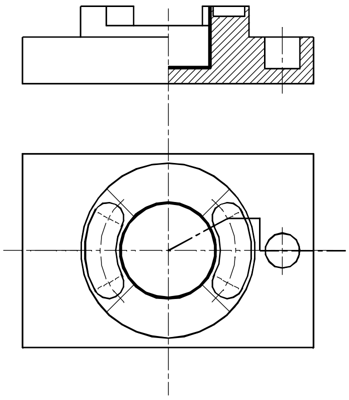
Enter: F90 S509 F1=90

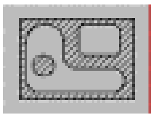
Groove milling cutter R5

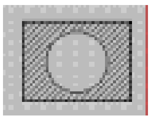


Enter: F90 S509 F1=90

21.7.9 Milling round pocket (50 mm dia)







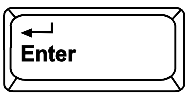
Enter: R25.01 L22 X0 Y0 Z-10

Enter: A3=0 A4=1 A5=90 K2=0.2 K3=0 W1=2 K5=1 K4=50 K1=5

Groove milling cutter R5

Enter: F90 S509 F1=90

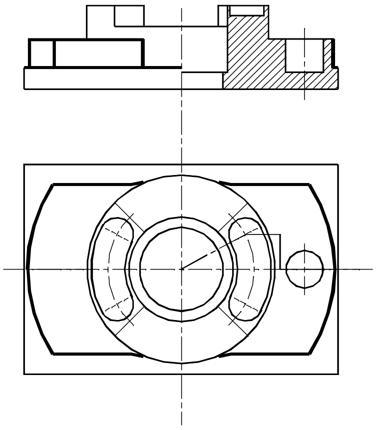
Groove milling cutter R5

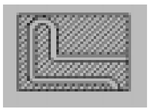


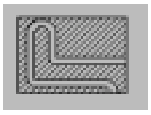
Enter: F90 S509


Speichern

21.7.10 Freely formed contour

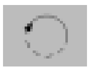






Enter: X-73 Y0 Z-16 I2=1 R2 L14 K5=1

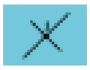
A freely formed contour is created with the aid of ICP.

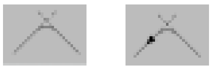
 

Enter: I0 J0

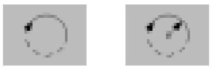
 

Enter: X0 Y40

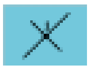


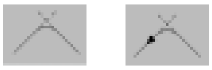


Enter: R15

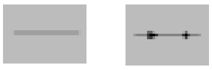


Enter: I0 J0 R45

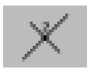


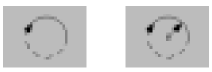


Enter: R15

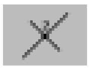


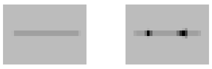
Enter: X50 Y40



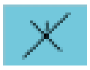


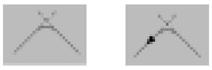
Enter: I0 J0 R73



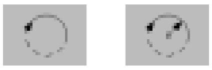


Enter: X0 Y-40

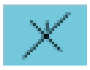


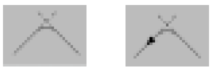


Enter: R15

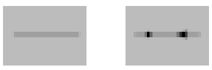


Enter: I0 J0 R45

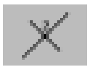


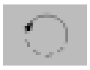


Enter: R15



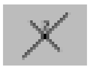
Enter: X-50 Y-40



Enter: X-73 Y0 I0 J0

Eingabe fertig



Zurück

ICP beenden

Enter: A4=1 K2=0.2 R2 W1=2 K1=5

Speichern

WZ-Nummer auswählen — End milling cutter R10

Enter: F148 S254 F1=148

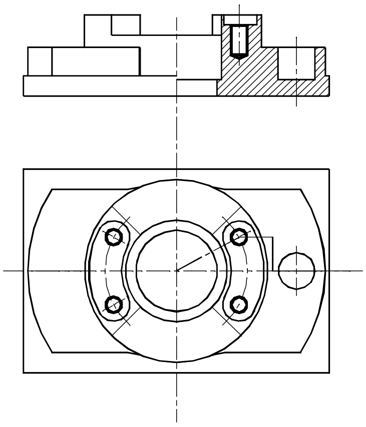
Speichern

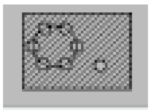
WZ-Nummer auswählen — End milling cutter R10

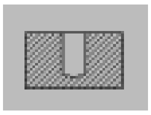
Enter: F148 S254

Speichern

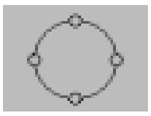
21.7.11 Boring and countersinking (8.5 mm dia)



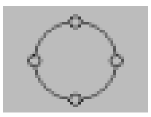




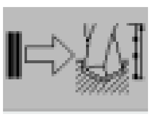
Enter: D8.5 L1=15 I1

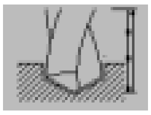


Enter: P1=2 R35 A1=-28 A2=56 X0 Y0 Z-5



Enter: P1=2 R35 A1=152 A2=56 X0 Y0 Z-5



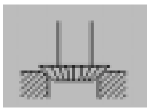


Drill R4.25

Enter: W1=2 W2=5 K5 F40 S299

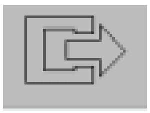
Note

Value 5 is to be entered for W2, as the zero point is located in the Z axis, 5 mm above the holes.

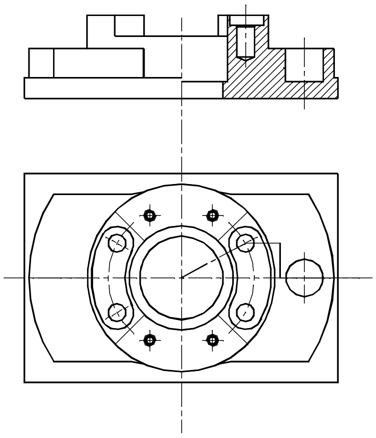


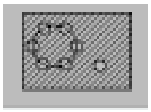
Centre drill R6

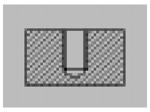
Enter: W2=5 F30 S238 F1=30



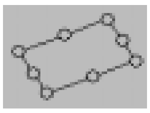
21.7.12 Boring and threading (M6)



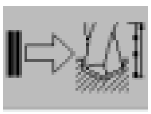


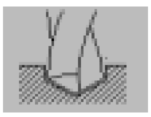


Enter: D6 L2=10 I0



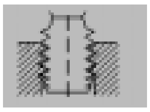
Enter: P1=2 U30 A1=0 P2=2 V60 A2=90 X-15 Y-30 Z-10





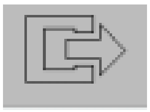
Drill bit R2.5

Enter: W1=2 W2=10 F40 S509

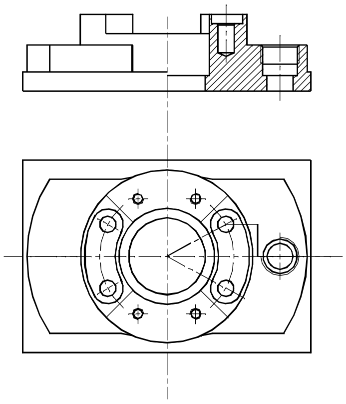


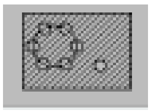
Screw tap M6

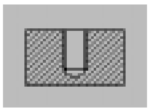
Enter: W1=2 W2=10 F318 S318



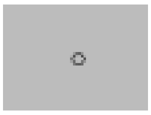
21.7.13 Threading (M20 x 1.5)



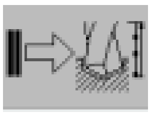


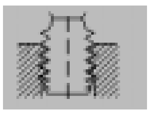


Enter: D20 F1.5 L2=10 I0



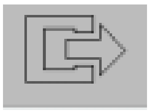
Enter: X59 Y0 Z-16





Screw tap M20 x 1.5

Enter: W2=16 F142.5 S95



21.7.14 End of program

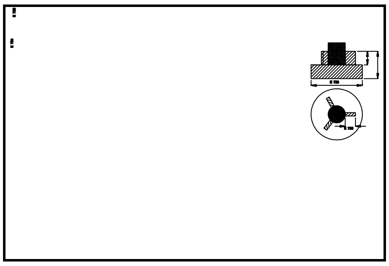
21.8 IPP start macro

IPP operation calls for a special file in the macro memory before a complete program can be created, to be precise the macro N9999998. This file is to be created specially for the machine tool and the most common application. Users can tailor this file if necessary to suit their own requirements.

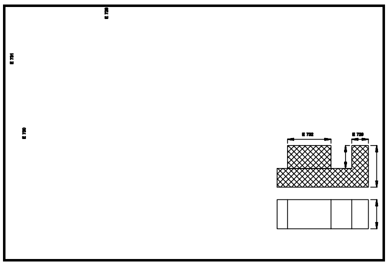
The values contained in the start file are purely initial values (or proposals).

An example of a standard start macro is shown below. The values assigned to the parameters are used in IPP as standard values for specific addresses.

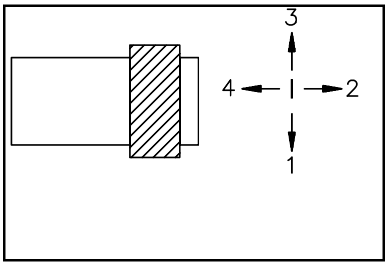
N9999998 (IPP Milling default setup macro)
N1
N1 (Graphic of blank contour)
N1
N100 E701=100 (X Length component)
N101 E702=100 (Y Length component)
N102 E703=100 (Z Length component)
N103 E704=1 (Stock on top plane)
N104 E705=100 (Diameter component)
N1
N1 (Shifting of blank)
N1
N105 E707=-50 (X Distance to zero point)
N106 E708=-50 (Y Distance to zero point)
N107 E709=0 (Z Distance to zero point)
N108 E710=1 (Zero point material 0=Middle 1=Left/Front)
N1
N1 (General parameters)
N1
N109 E712=15 (Extra space window)
N110 E713=10 (Label number)
N111 E714=50 (Retract after action)
N112 E715=1 (Coolant 0=Off 1=M8 2=M7)
N113 E717=1 (Auto-tool generate 0=No 1=Yes)
N1
N1 (Machine table dimensions)
N1
N114 E720=900 (X Length machine plate)
N115 E721=480 (Y Length machine plate)
N1
N1 (Fixture for jaw chuck)
N1
N116 E723=210 (Round clamp diameter)
N117 E724=30 (Yaw height)
N118 E725=130 (Round clamp height)
N119 E726=40 (Yaw length round clamp)



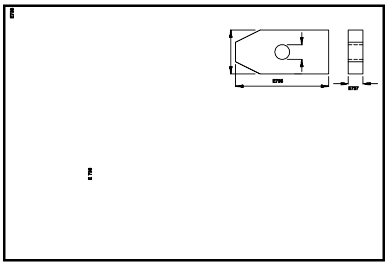
N1
N1 (Fixture for vice)
N1
N120 E728=100 (Clamp length)
N121 E729=40 (Yaw width)
N122 E730=30 (Clamp height)
N123 E731=100 (Bench clamp thickness)
N124 E732=150 (Bench clamp length)



N125 E733=1 (Clamping orientation)



N1
N1 (Gripping jaws)
N1
N126 E735=160 (Clamping bar length)
N127 E736=40 (Clamping bar width)
N128 E737=20 (Clamping bar thickness)
N129 E738=12 (Clamping bold diameter)



N1
N1 (Facing)
N1
N130 E740=10 (Incremental depth facing)
N131 E741=115 (Toolnumber for facing)

N132 E742=67 (Cutting width % face milling)
N133 E743=10 (Clearance in plane face milling)
N134 E744=2 (Clearance tool axis face milling)
N135 E745=7 (Machining code face milling)
N136 E746=0 (Facing-tool 0=On 1=Outside 2=Rapid)

N1
N1 (Drillings)
N1
N137 E750=0 (Chamfer)
N138 E751=118 (Drill top angle)
N139 E752=90 (Sink top angle)
N1
N1 (Reaming)
N1
N140 E755=10 (Tooltype code reaming)
N141 E756=5 (Machining code reaming)
N142 E757=3 (Space for blind hole reaming)
N143 E758=5 (Overshoot for hole reaming)
N1
N1 (Sinking)
N1
N144 E760=174 (Toolnumber sinking)
N145 E761=3 (Machining code sinking)
N146 E763=2 (Tooltip distance sink)
N1
N1 (Centering)
N1
N147 E765=0 (Dwell time general operations)
N148 E766=172 (Toolnumber centering)
N149 E767=1 (Machining code centering)
N150 E768=3 (Depth center)
N1
N1 (General cycles)
N1
N151 E769=2 (Clearance in tool axis)
N152 E770=0 (Safety clearance in tool axis)
N1
N1 (Drilling)
N1
N153 E773=3 (Tooltype code drilling)
N154 E774=2 (Machining code drilling)
N155 E775=2 (Overshoot for drilled through hole)
N1
N1 (Deep-hole drilling)
N1
N156 E777=10 (Incremental depth deep first drill)
N157 E778=0.5 (Incremental decrease after each step)
N158 E779=0.1 (Clearance after each step)
N1
N1 (Boring)
N1
N159 E783=13 (Tooltype code boring)

N160 E784=2 (Clearance boring in tool axis)
N161 E785=0 (Extra clearance boring in tool axis)
N162 E788=6 (Machining code boring)
N163 E789=3 (Space for blind hole boring)
N164 E790=4 (Overshoot for hole boring)
N165 E792=0.2 (Dwell time boring)
N1
N1 (Tapping)
N1
N166 E795=9 (Tooltype code tapping)
N167 E796=0 (Dwell time tapping)
N168 E797=3 (Clearance tapping in tool axis)
N169 E798=4 (Machining code tapping)
N170 E799=2 (Minimum thread runout tapping)
N171 E800=2 (Number of revolutions for stop tapping)
N172 E801=5 (Safety clearance tapping in tool axis)
N173 E802=3 (Space for blind hole tapping)
N1
N1 (Milling or countersinking of blind hole)
N1
N174 E805=23 (Toolnumber milling)
N175 E806=7 (Machining code milling)
N176 E807=10 (Incremental depth milling)
N177 E808=83 (Cutting width % milling)
N178 E809=1 (Drill or mill operation)
N1
N1 (Countersinking or bevelling of blind hole)
N1
N179 E811=133 (Toolnumber for drilling)
N180 E812=2 (Machining code drilling)
N1
N1 (General milling, pocket, recess and contours)
N1
N181 E813=86 (Toolnumber roughing)
N182 E814=7 (Machining code roughing)
N183 E815=7 (Machining code rough/drill)
N184 E816=87 (Toolnumber finishing)
N185 E817=7 (Machining code finishing)
N1
N1 (Pocket)
N1
N186 E820=10 (Incremental depth pocket)
N187 E821=8000 (Macro number counter)
N188 E822=50 (Cutting width % pocket)
N189 E823=1 (Finishing pocket 0=No 1=Yes)
N190 E824=1 (Mill direction 1=Climb -1=Conv)
N191 E825=0.4 (Finishing allowance pocket)
N192 E826=1 (Drilling pocket 0=No 1=Yes)
N193 E827=0 (Radius of the corner)
N1
N1 (Contours)
N1
N194 E830=10 (Start distance contour)
N195 E831=10 (Incremental depth contour)
N196 E832=0.4 (Finishing allowance contour)
N197 E833=1 (Finishing contour 0=No 1=Yes)
N198 E834=0 (Drilling contour 0=No 1=Yes)

```
N1
N1 (Recess milling)
N1
N199 E838=0.4 (Finishing allowance edge)
N200 E839=10 (Incremental depth edge)
N201 E840=1 (Finishing edge 0=No 1=Yes)
N202 E841=83 (Cutting width % edge)
N1
N1 (Tapping)
N1
N203 E899=10 (Number tap diameters max 16)
N204 E900=2 (Diameter tap 1)
N205 E901=0.4 (Pitch 1)
N206 E902=1.6 (Diameter drill hole 1)
N207 E903=3 (Diameter tap 2)
N208 E904=0.5 (Pitch 2)
N209 E905=2.5 (Diameter drill hole 2)
N210 E906=4 (Diameter tap 3)
N211 E907=0.7 (Pitch 3)
N212 E908=3.3 (Diameter drill hole 3)
N213 E909=5 (Diameter tap 4)
N214 E910=0.8 (Pitch 4)
N215 E911=4.2 (Diameter drill hole 4)
N216 E912=6 (Diameter tap 5)
N217 E913=1 (Pitch 5)
N218 E914=5 (Diameter drill hole 5)
N219 E915=8 (Diameter tap 6)
N220 E916=1.25 (Pitch 6)
N221 E917=6.8 (Diameter drill hole 6)
N222 E918=10 (Diameter tap 7)
N223 E919=1.5 (Pitch 7)
N224 E920=8.5 (Diameter drill hole 7)
N225 E921=12 (Diameter tap 8)
N226 E922=1.75 (Pitch 8)
N227 E923=10.2 (Diameter drill hole 8)
N228 E924=16 (Diameter tap 9)
N229 E925=2 (Pitch 9)
N230 E926=14 (Diameter drill hole 9)
N231 E927=20 (Diameter tap 10)
N232 E928=2.5 (Pitch 10)
N233 E929=17.5 (Diameter drill hole 10)
N234 E930=20 (Diameter tap 11)
N235 E931=2.5 (Pitch 11)
N236 E932=17.5 (Diameter drill hole 11)
N237 E933=20 (Diameter tap 12)
N238 E934=2.5 (Pitch 12)
N239 E935=17.5 (Diameter drill hole 12)
N240 E936=20 (Diameter tap 13)
N241 E937=2.5 (Pitch 13)
N242 E938=17.5 (Diameter drill hole 13)
N243 E939=20 (Diameter tap 14)
N244 E940=2.5 (Pitch 14)
N245 E941=17.5 (Diameter drill hole 14)
N246 E942=20 (Diameter tap 15)
N247 E943=2.5 (Pitch 15)
N248 E944=17.5 (Diameter drill hole 15)
N249 E945=20 (Diameter tap 16)
```

```
N250 E946=2.5 (Pitch 16)
N251 E947=17.5 (Diameter drill hole 16)
N1
N1 (Material code)
N1
N252 E950=12 (Material code)
N1
N1 (Reserved parameter)
N1
N253 E961=1 (Reserved for internal use)
N254 E962=0 (Reserved for internal use)
N255 E963=0 (Reserved for internal use)
N256 E964=0 (Reserved for internal use)
N257 E965=0 (Reserved for internal use)
N258 E966=0 (Reserved for internal use)
```

Note

Before installation of IPP the IPP Startup Macro must be removed from the Startup directory

22. Program structure and block format

22.1 Program extract

```
%PM9001
N9001
N1 G17 S630 T1 M6
N2 G54
N3 G0 X60 Y30 Z-8 M3
N4 G1 Z-10 F50
N5 G43 X80 F100
N6 G42
:
M30
```

22.2 Memory identifier

Main program: program number.PM or %PM
Subprogram: program number.MM or %MM

22.3 Program number

N1 - N9999999

22.4 Program block

A program block consists of a several program words (max. 255 characters). Each address can only be used once in the program block.

1	Block number	N1
2	Geometric commands	G17 S630
3	Technological commmands (S,F,T,M)	T1 M3
Total		N1 G17 S630 T1 M3

22.5 Block number

N1 - N9999999
The sequence of the block numbers is unimportant.
The blocks are executed in the programmed sequence.

22.6 Program word

Address, sign, number
(Positive sign can be deleted)

Positive word	X21.43
Negative word	Y-13.8
Indexed word	X1=15.3
Calculated word	=12.5+30
	Y=2^5
	Y=sqrt(25)

22.7 Input formats for axis addresses

Metric	6.3	X123456.789
Inch	5.4	X12345.6789

23. G-Functions

23.1 Rapid traverse G0

N... G0 [axis coordinates]

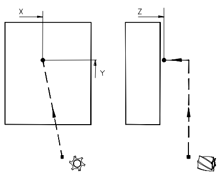
Parameters

G	Eilgang
X	Endpunktkoordinate
Y	Endpunktkoordinate
Z	Endpunktkoordinate
A	Endwinkel
B	Endwinkel
B1=	Winkel
B2=	Polarwinkel
?90=	Endpunkt abs. (X,Y,Z..)
?91=	Endpunkt inkr. (X,Y,Z..)
L1=	Streckenlänge
L2=	Polarlänge
P1=	Punktedefinitionsnummer

Example

N... G0 X25 Y15 Z30

Simultaneous movement in main plane XY, followed by movement in tool axis Z



Notes

At the start of a program and following each change of tool or swivel head, each active axis must be programmed in a program block for traversing movements. Every axis is thus in the start position.
The positioning logic determines the sequence of the traverse movements in rapid traverse.

Tool movement:	to workpiece G17,18,19			away from workpiece G17,18,19		
1st axis movement	4.+5	4.+5	4.+5	Z	Y	X
2nd axis movement	X+Y	X+Z	Y+Z	X+Y	X+Z	Y+Z
3rd axis movement	Z	Y	X	4.+5.	4.+5.	4.+5.

23.2 Linear interpolation G1

Linear interpolation in the main level:
N.. G1 {X..} {Y..} {Z..} {F..}

3 D interpolation:
N.. G1 X.. Y.. Z.. {F..}

One axis of rotation:
N.. G1 {A..} {B..} {C..} {A40=..} {B40=..} {C40=..} {F...}

Several axes:
N... G1 {X..} {Y..} {Z..} {A..} {B..} {C..} {A40=..} {B40=..} {C40=..} {F...}

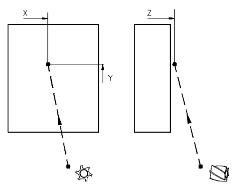
Parameters

G	Linearinterpolation
X	Endpunktkoordinate
Y	Endpunktkoordinate
Z	Endpunktkoordinate
A	Endwinkel
B	Endwinkel
B1=	Winkel
B2=	Polarwinkel
?90=	Endpunkt abs. (X,Y,Z..)
?91=	Endpunkt inkr. (X,Y,Z..)
L1=	Streckenlänge
L2=	Polarlänge

P3=	Punktedefinitionsnummer
P4=	Punktedefinitionsnummer

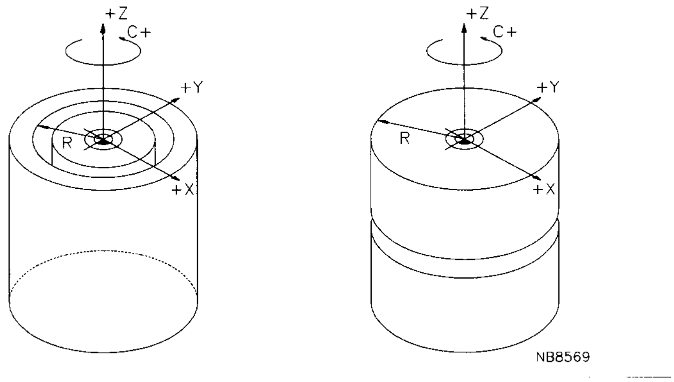
Examples

1. 3 D interpolation

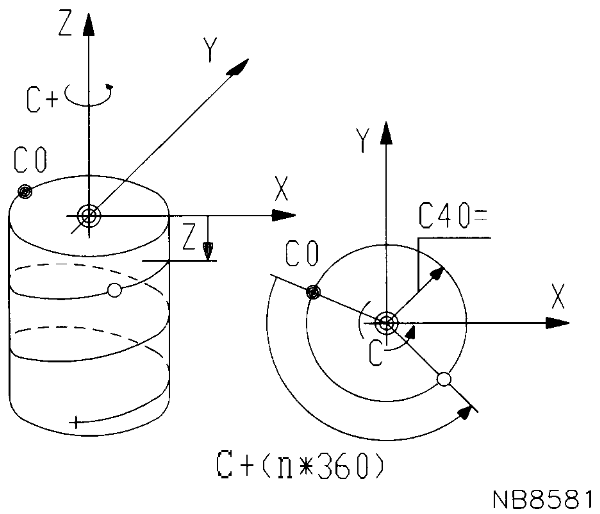


:
N14 G0 X10 Y5   Z20
N15 G1 X20 Y10 Z40 F100   Simultaneous movement of the axes
:
:

2. Programming rotary axes, with and without linear axis

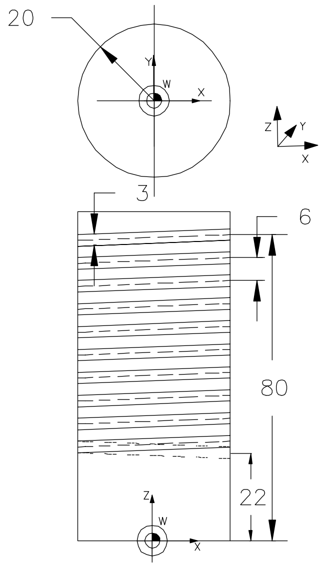


One rotary axis and one linear axis:



Z and C axis
(X and A axis)
(Y and B axis)

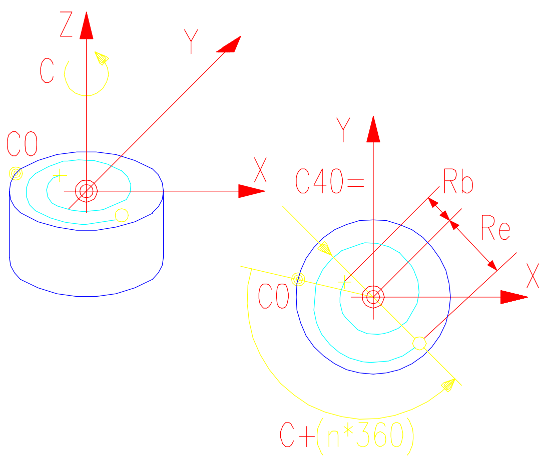
Thread on a cylindrical surface:



3563

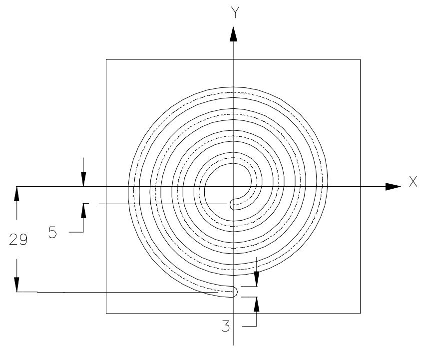
```
:
N10  G18
N11  T1  M6  S2000  F200          Change tool
N12  G0  X0  Z80  Y22  C0  M3
N13  G1  Y18                      Set tool to required position
N14  Z20  C3600  C40=18           Mill spiral, 10 turns
N15  G0  Y25
:
```

Linear axis with additional rotary axis:



C40=..(central path radius)
C40=(Rb+Re)2
Rb(start radius)
Re(end radius)

Spiral:




```
:
N10 G17 T1 M6                 Change tool
N11 G54                       Zero offset
N12 G0 X0 Y5 Z3 C0 S200 M3
N13 G1 Z-2 F100               Approach start position
N14 Y29 C1440 C40=17 F200     Mill spiral, 4 turns
N15 G0 Z100
:
```

Note:

MACHINES WITH KINEMATIC MODEL **(as of V410)**

The radius of the rotating axis is calculated automatically in machine tools with a kinematic model. A40=, B40= or C40= no longer need to be programmed. The new option is programmed with G94 F5=1.

23.3 Circular clockwise / circular counter clockwise G2/G3

Full circle:
N.. G2/G3 [centre point]

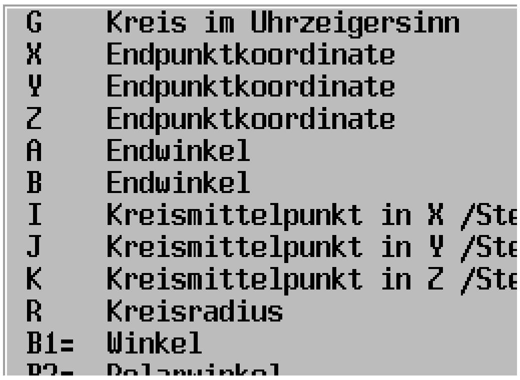
Circular arc less than or equal to 180:
N.. G2/G3 [end point] R..

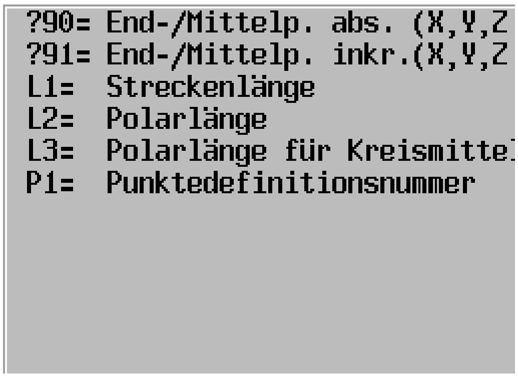
Circular arc greater than 180 :
N.. G2/G3 [centre point] [end point]
N.. G2/G3 [centre point] B5=..

2.5D-Interpolation:
N... G2/G3 [centre point] [end point of arc] [end point on the linear or rotary axis]

Spiral:
N... G2/G3 [centre point] [end pointof arc] [end point on the linear or rotary axis] [pitch]
N... G2/G3 [centre point] [pitch] B5=...

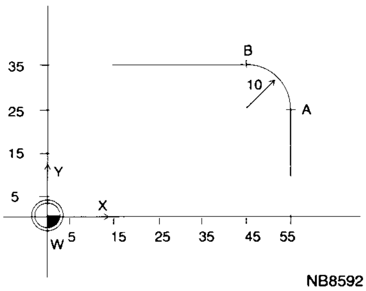
Parameters G2 / G3





Examples

Circular arc less than or equal to 180°

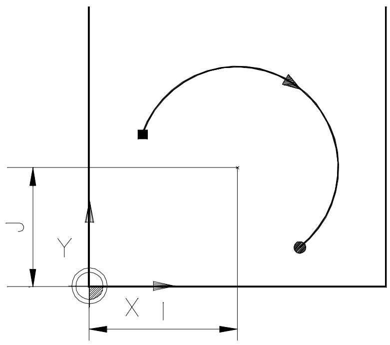


N10 G1 X55 Y25 F100 Linear movement
N20 G3 X45 Y35 R10 Circular counterclockwise movement

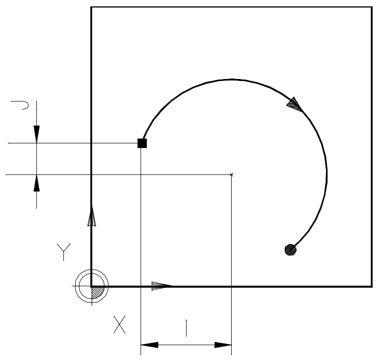
Circular arc greater than 180°

Centre point coordinates:
G17
N.. G2/G3 I.. J..
G18
N.. G2/G3 I.. K..
G19
N.. G2/G3 J.. K..

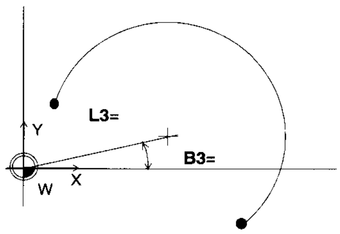
Absolute centre point coordinates (G90):
Centre point coordinates in relation to program zero point



Incremental centre point coordinates (G91):
Centre point coordinates in relation to the start point



Polar centre point coordinates
N.. G2/G3 L3=.. B3=.. (G17/G18/G19)

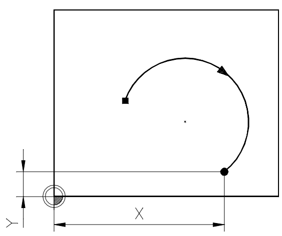


NB8711

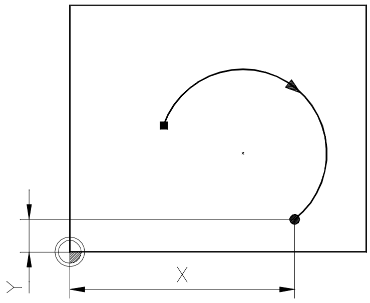
End point coordinates:

Cartesian end point coordinates
G17
N.. G2/G3 X.. Y..
G18
N.. G2/G3 X.. Z..
G19
N.. G2/G3 Y.. Z..

Absolute end point coordinates (G90):
End point coordinates in relation to the program zero point

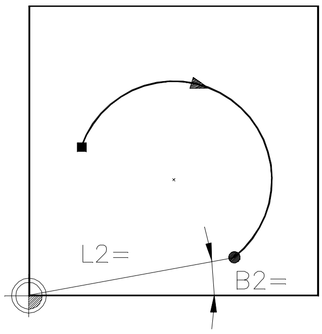


Incremental end point coordinates (G91):
End point coordinates in relation to the start point

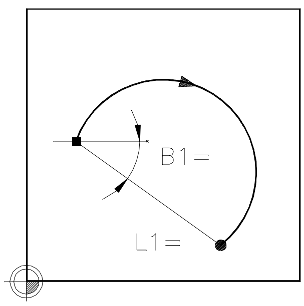


Polar end point coordinates:

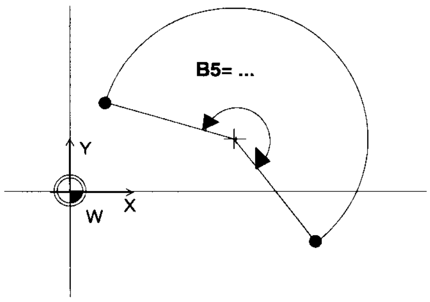
End point coordinates in relation to the program zero point
N.. G2/G3 L2=.. B2=..   (G17/G18/G19)



End point coordinates in relation to the start point
N.. G2/G3 L1=.. B1=..   (G17/G18/G19)



Angle of circular arc:
N2.. G2/G3 B5=..	(G17/G18/G19)



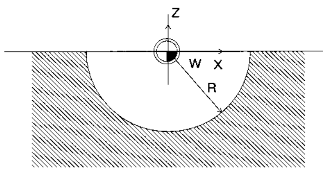
NB8714

Circular movement not in the main level

Circular arc smaller than or equal to 180°:
N2.. G2/G3	[end point coordinates of the linear axes] R..
N2.. G2/G3	[cartesian coordinates of circle centre point]

Circular arc greater than 180°:
N2.. G2/G3	[cartesian coordinates of the end point andcircle centre point]

Use of the radius correction is not possible.



NB8574

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

Circle in the main level:
N.. G2/G3	[circle definition][tool axis]

Level	G17	G18	G19
Tool axis	Z	Y	X

Circle not in the main level:
N.. G2/G3	[cartesian coordinates of end point and circle centre point] [tool axis]

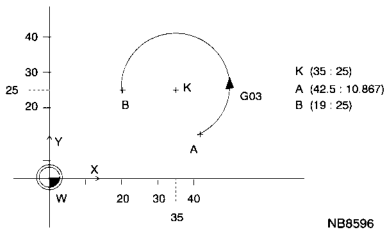
Level	G17	G18	G19
End point	X..Y..	X..Z..	Y..Z..
Centre point	I..J..	I..K..	J..K..
Tool axis	Z	Y	X

Spiral interpolation

Level	G17	G18	G19
Tool axis	Z	Y	X
Centre point	I..J..	I..K..	J..K..
	/	/	/
	B3=..L3=..	B3=..L3=..	B3=..L3=..
Circular arc angle	B5=..	B5=..	B5=..
Spiral pitch	K	J	I

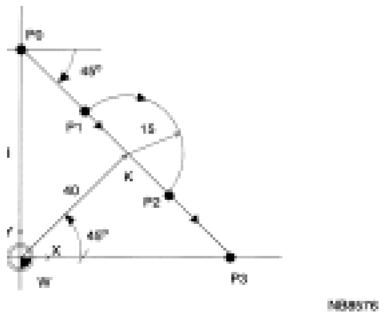
The value of (B5=) can lie between 0 und 999999 degrees (approx. 2777 rotations)

Level	G17	G18	G19
Tool axis	Z	Y	X
Circle end point	X..Y..	X..Z..	Y..Z..
Centre point	I..J..	I..K..	J..K..
Spiral pitch	K	J	I

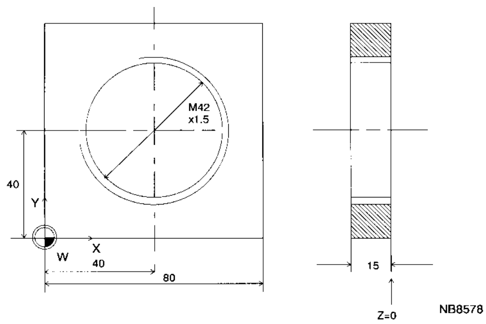


Absolute coordinates
N82000
N1 G17
N2 G98 X0 Y0 Z10 I60 J60 K-30
N3
N4 G0 X0 Y0 Z-10
N5
N6 G1 X42.5 Y10.867 F200 — Linear movement
N7 G3 X19 Y25 I35 J20 — Circular counterclockwise movement (absolute)
N8
N9 G0 Z100 M30

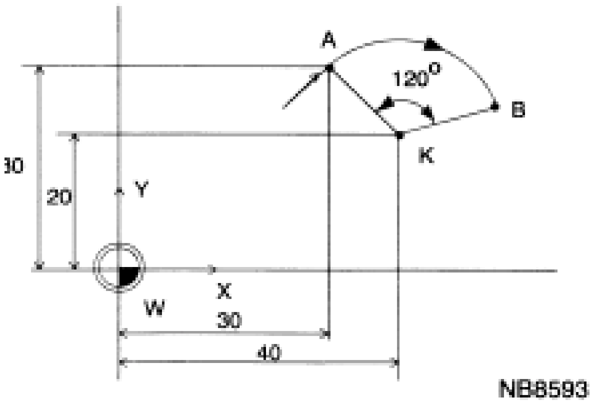
Incremental coordinates
N82001
N1 G17
N2 G98 X0 Y0 Z10 I60 J60 K-30
N3
N4 G0 X0 Y0 Z-10
N5
N6 G1 X42.5 Y10.867 F200 — Linear movement
N7 G91 — Incremental size programming
N8 G3 X-23.5 Y14.133 I-7.5 J9.133 — Circular counterclockwise movement (incremental)
N9
N10 G0 Z100 M30



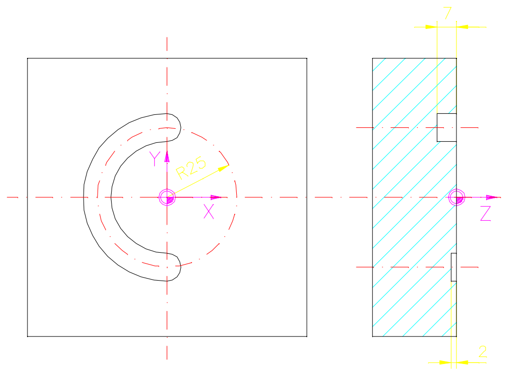
N82030
N1
N2 G17
N3 G98 X-10 Y-10 Z10 I80 J80 K-30 Definition of graphic window
N4
N5 G0 X0 Y56.568 Z0
N6 G1 F200 B1=-45 L1=25
N7 G2 B1=-45 B3=45 L1=30 L3=40 Circular clockwise movement
N8 G1 B1=-45 L1=25
N9
N10 G0 Z100 M30



N82040
N10 G17 T1 M6 Active plane, change tool
N11 G0 X40 Y40 Z1.5 S400 M3
N12 G1
N13 G43 Y61 F120 Tool radius compensation to end point
N14 G42 Tool radius compensation to the right
N15 G2 I40 J40 K1.5 B5=4320 Circular clockwise movement (thread)
N16 G40 Cancel tool radius compensation
N17 G1 Y40
N18 G0 Z100 M30




```
:
N10  G1  X30  Y30  F500
N11  G2  I40  J20  B5=120        Circular clockwise movement
:
```

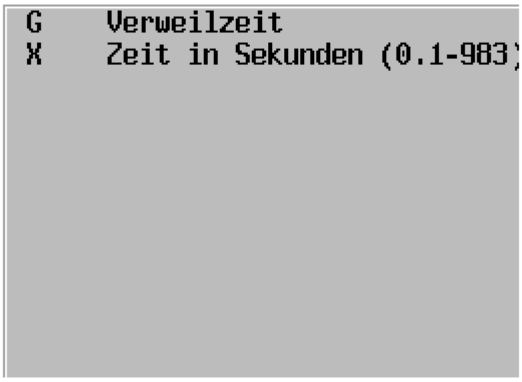



```
N85770
N1 G17
N2 G54
N3 G98 X20 Y50 Z10 I-100 J-100 K-20
N4
N5
N6 S650 T1 M6                   Change tool
N7 G0 X0 Y-25 Z5 M3             Spindle ON clockwise; rapid traversing movement
N8 G1 Z-2 F100                  Move to machining depth
N9 G2 X0 Y25 Z-7 I0 J0 F200     Circular clockwise movement
N10 G1 Z5                       Retract tool from material
N11
N12
N13 M30
```

23.4 Dwell time G4

N... G4 X...

Parameters



Dwell time: 0.1 - 983 seconds (approx. 16 minutes)

Example

:
N50 G4 X2.5 Dwell time 2.5 seconds
:

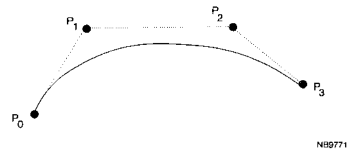
23.5 Spline Interpolation G6

Spline interpolation allows the programmer to create a uniform and smooth curve by entering a few points.

Formats with Bezier splines

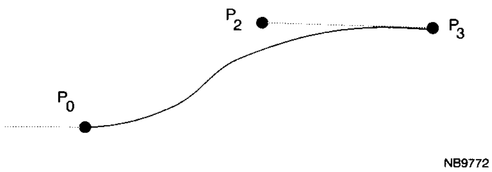
Spline with three vertices:

G6 X61=.. Y61=.. Z61=.. X62=.. Y62=.. Z62=.. X.. Y.. Z..



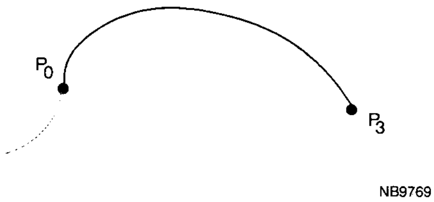
Spline with two vertices and constant tangents to the spline:

G6 X62=.. Y62=.. Z62=.. X.. Y.. Z..

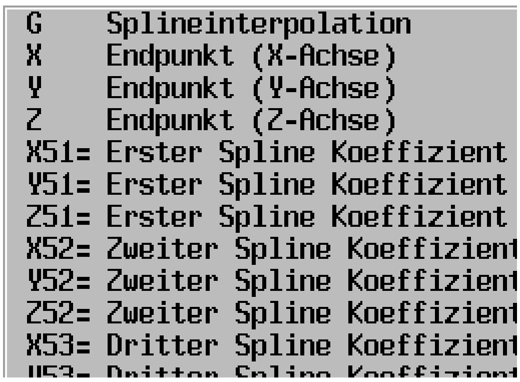


Spline with constant curvature to the previous spline:

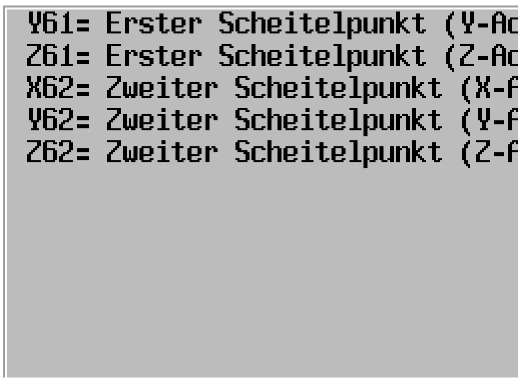
G6 X.. Y.. Z..



Parameters Bezier-Splines



G Splineinterpolation
X Endpunkt (X-Achse)
Y Endpunkt (Y-Achse)
Z Endpunkt (Z-Achse)
X51= Erster Spline Koeffizient
Y51= Erster Spline Koeffizient
Z51= Erster Spline Koeffizient
X52= Zweiter Spline Koeffizient
Y52= Zweiter Spline Koeffizient
Z52= Zweiter Spline Koeffizient
X53= Dritter Spline Koeffizient



Y61= Erster Scheitelpunkt (Y-Ac
Z61= Erster Scheitelpunkt (Z-Ac
X62= Zweiter Scheitelpunkt (X-A
Y62= Zweiter Scheitelpunkt (Y-A
Z62= Zweiter Scheitelpunkt (Z-A

Formats with cubic splines

Spline with all coefficients defined:

G6 X51=.. Y51=.. Z51=.. X52=.. Y52=.. Z52=.. X53=.. Y53=.. Z53=..

Spline with constant tangents to the previous spline:

G6 X52=.. Y52=.. Z52=.. X53=.. Y53=.. Z53=..

Spline with constant curvature to the previous spline:

G6 X53=.. Y53=.. Z53=..

Parameters

Cubic splines

X51=,Y51=,Z51= First spline coefficient

52=,Z52= Second spline coefficient

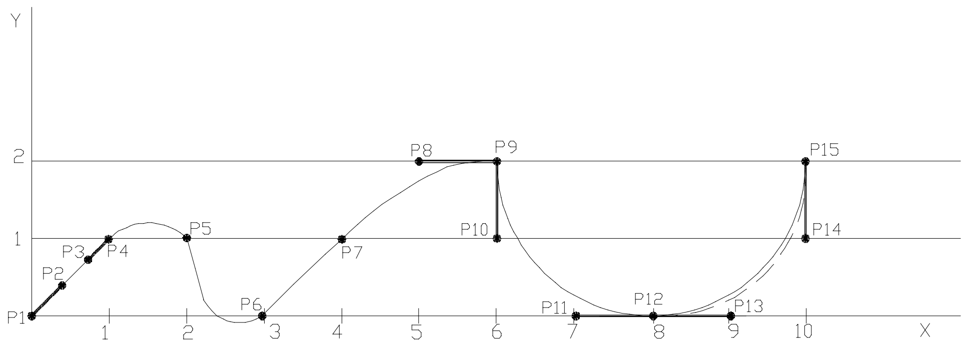
53=,Z53= Third spline coefficient

Example:

Bezier splines

Spline mit konstanter Krümmung mit der vorherigen Spline:

G6 X53=.. Y53=.. Z53=..



N17001 (Spline Kurve)
N1 G98 X2 Y-6 Z-2 I10 J10 K10
N2 G17
N101 G0 X0 Y0 Z0 F500
N102 G6 X1 X61=0.3 X62=0.7 Y1 Y61=0.3 Y62=0.7 Z0.001 Z61=0 Z62=0
N103 X2 Y1.001 Z0
N104 X3 Y0 Z0.001
N105 X4 Y1 Z0
N106 X6 X62=5.7 Y2 Y62=2 Z0.001 Z62=0

```
N107 X8 X61=6 X62=7.5 Y0 Y61=1.5 Y62=0 Z0 Z61=0 Z62=0.001
N108 X10 X61=8.5 X62=10 Y2 Y61=0 Y62=1.5 Z0.001 Z61=0.001 Z62=0
N109 G0 X0 Y0 Z0
N110 M30
```

N101: Approach start position (P1)
N102: First element. Straight line. Touches P1-P2 and P3-P4. End point is P4. All coordinates must be entered. For this purpose, select a straight line.
N103: Curve passes through P5
N104: Curve passes through P6
N105: Curve passes through P7. If the curve does not have the required shape, add more points.
N106: Curve passes through P9 and touches line P8-P9.
N107: New curve with sharp transition is defined. First curve element starts in P9 and touches P9-P10 and P11-P12. End point is P12.
N108: New curve with tangential transition is defined. First curve element starts in P12 and touches P12-P13 and P14-P15. End point is P15. The radius of curvature can be adjusted in P15 by changing distance P14-P15.

Note:

In G6 identical coordinates must be different in two blocks (Z0 and Z0.001)

23.6 Tilt operating planes G7 (starting with V400)

Programming of a tilt operating plane for four or five axis machines.

The position of the operating plane can be tilted using the function "Tilt operating plane". The operation which has then been programmed in the principal plane (G17, G18) can then be implemented within the tilt operating plane. The tool axis is then orientated vertically in the new plane.

The tilt of the operating planes is defined and implemented using the G7 function.

Format

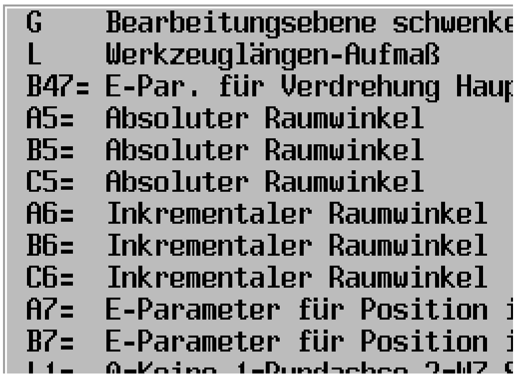
N.. G7 {A5=.. | A6=..} {B5=.. | B6=..} {C5=.. | C6=..} {A7=..} {B7=..} {C7=..} {B47=..} {L1=..} {L..}

Parameters

Raumwinkel
A5=,B5=,C5= Absoluter Raumwinkel
:,C6= Inkrementaler Raumwinkel

Parameternummer
A7=,B7=,C7= E-Parameter für berechnete Position
E-Parameter für Verdrehung Hauptebene

Bewegungsarten
L1= 0=Keine,1=Rundachse,2=WZ-Spitze
Werkzeuglängenaufmaß



Ancillary function

G FUNCTIONS WHICH ARE NOT PERMITTED, IF G7 IS SWITCHED ON
If G7 is switched on, the following (modal) G functions are not allowed to be active:
G6, G9, G19, G41, G42, G43, G44, G61, G64, G73, G141, G182, G197, G198, G199, G200, G201, G203, G204, G205, G206, G207, G208

G FUNCTIONS WITHIN G7 WHICH ARE NOT PERMITTED
If G7 is active, the following G functions are not permitted:
G6, G19, G182

G FUNCTIONS WHICH ARE NOT PERMITTED, IF G7 IS SWITCHED OFF

If G7 is switched off, the following (modal) G functions are not allowed to be active:

G9, G41, G42, G43, G44, G61, G64, G73, G141, G197, G198, G199, G200, G201, G203, G204, G205, G206, G207, G208

If one of these G functions which are not permitted is active, the fault report P77 "G function and Gxxx are not permitted" is generated.

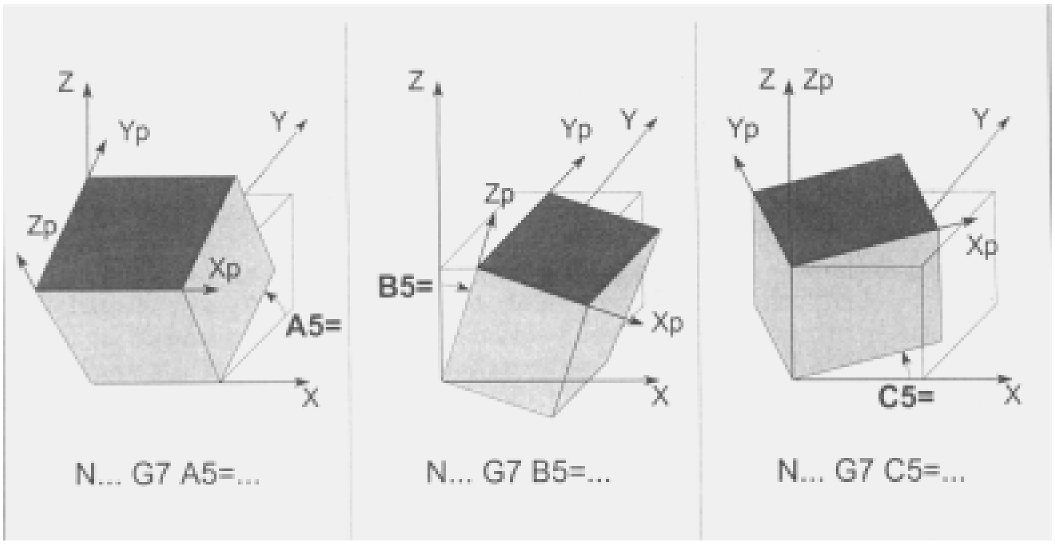
Type of function

modal

Notes and employment

G7 FUNCTION

The freely programmable operating plane is defined using the new G7 function:
The new plane becomes active with the original null point.
The tool is orientated vertically in the new plane. The axes which move depend upon the machine configuration and the programming.
The display shows the coordinates in the new (tilt) plane.
The manual operation is orientated in accordance with the new plane.



DEFINITION OF NEW OPERATING PLANES

Tilting of the operational plane can be defined in two ways:

- Programming with A5=, B5= or C5= parameters. In this way, the absolute tilts about the corresponding positive axes are defined. The tilts are implemented as follows:
  1. the active G7 tilt is raised
  2. C5= tilt about the machine fixed positive Z axis
  3. B5= tilt about the positive Y axis
  4. A5= tilt about the positive X axis

- Programming with A6=, B6= or C6= parameters. The incremental tilts about the current corresponding positive axes are defined in this way. The tilts are implemented as follows:
  1. C6= tilt about the current G7 positive Z axis
  2. B6= tilt about the current G7 positive Y axis
  3. A6= tilt about the current G7 positive X axis

The programming is independent of the machine configuration. The plane tilt is calculated with reference to the current null point. The movement is dependent upon the machine configuration.

ABFRAGEN EINER BERECHNETEN WINKELPOSITION

A7=, B7=, C7= Enthält die Nummer des E-Parameters, in dem der errechnete Winkel der entsprechenden Rundachse gesetzt wird.

B47= Enthält die Nummer des E-Parameters, in dem der errechnete Winkel der Hauptebene gesetzt wird.

TOOL VERTICAL ON THE DEFINED TILT PLANE
The G7 tilt movement takes place interpolating with the power traverse. It tilts the tool axis to the defined plane. The axes which are moved depend upon the type of movement L1=:

- L1=0 The rotary axes do not move (start position).

Comment:

The tilt movement can then be implemented, using the E parameters loaded into A7=, B7= or C7=. This movement must then be programmed manually.

- L1=1 Interpolate only the rotary axes, which do not move the linear axes.
- L1=2 Interpolate the rotary axes and to that end execute a "compensatory movement". In this way the tool tip remains in the same position with respect to the workpiece.

WERKZEUGLÄNGENAUFMAß (L)
Wenn die Schwenkbewegung um die Werkzeugspitze stattfindet (L1=2), definiert L ein Aufmaß in der Werkzeugrichtung, zwischen dem programmierten Endpunkt und der Werkzeugspitze.

SWITCHING OFF THE G7 FUNCTION
The operation of G7 remains active until G7 is switched off. G7 is switched off by the programming of G7 without parameters or by G7 L1=1 positioning of the rotary axes on the workpiece null point.

G7 is not switched off by M30 or <Program interrupt>. After switching on the control G7 is permanently active. Travel in the G7 plane is then possible. G7 is switched off in accordance with reference point travel or <Reset CNC>.

Note:

It is recommended that, at the start of every G7 program, that a G7 without parameters is programmed. In this way, during the start-up of the program (interrupt within the tilted plane and the new start), the plane is always reset. Without this G7 at the start, the first part of the program will be implemented in the tilted plane rather than in the untilted plane.

This programming is similar to programming with G17/G18 - different null points or different tools.

ROTARY AXES
Rotary axes can be programmed in the tilted planes in the normal way. It is the programmer's responsibility to ensure compatibility of the rotary axes with the G7 tilt.

ABSOLUTE POSITION G74
If G7 is active, the "Absolute position" G74 is referred to the machine coordinates. This is the same as in V3.3x.

GRAPHICS
The graphics display the G7 plane as the main picture. The screen is refreshed whenever G7 becomes active.
If G7 is active, the position between tool and workpiece is displayed.

DISPLAY
If G7 is active, a yellow icon is displayed on the screen behind the tool number. By means of a small "p" on the right next to the "axes characters", an indication is given as to whether the display relates to the tilted operating planes or to the machine coordinates. The operating status has been enhanced with the current reading of the programmed G7 space angle.

A new soft key (Jog to the G7 plane) appears in the "Jog operation type" soft key group. This soft key is used to switch between the tilted operating planes and the machine coordinates. If the position is displayed in machine coordinates, the actual position of the tool tip is shown.

CHANGE OF TOOL
If G7 is active, a tool change is not permitted (fault report). G7 must first be deselected. G7 must then be selected again, in order to resume work in the tilted plane following the tool change.

Example:

N100 G7 B5=45 L1=1	(plane is set)
N110 T14	(tool preselect)
..	
N200 G0 Z200	(the tool axis is withdrawn)
N210 G7 B5=0 L1=1	(deselect G7)
N220 M6	(tool change)
N230 G0 X.. Y.. Z..	(power traverse to the new start position)
N240 G7 L1=1 B5=45	(face is rotated again in the G7 plane)

PALETTE, TILT FACE OR TOOL CHANGE
While G7 is active a change of palette, tilt face or tool cannot be implemented. A fault is displayed and the program must be interrupted. Before such a change, G7 must be deactivated.

TILT OPERATING PLANE WITH M53/M54
During mixed operation with G7 and M53/M54, the tilt face positioning M53/M54 with M55 must be relinquished before programming G7. In this way, the face offset which is active under these conditions is relinquished.

M FUNCTIONS WHICH ARE NOT PERMITTED IF G7 IS SWITCHED ON
If G7 is switched on, the following M functions are not permitted to be active:
M53, M54

M FUNCTIONS WHICH ARE NOT PERMITTED WITHIN G7
If G7 is active, the following M functions are not permitted:
M6, M46, M53, M54, M60, M61, M62, M63, M66

FAULT REPORTS

P77 G-Funktion und Gxxx nicht erlaubt
Dieser Fehlertext gibt an, welche Kombination von G-funktionen nicht zugelassen ist. Z.B. wird G7 programmiert, wenn G41 aktiv ist, kommt der Fehler P77 'G-Funktion und G41 nicht erlaubt'.

P306 Plane not clearly defined
The G7 plane is defined with a mix of absolute (A5=, B5=, C5=) and incremental (A6=, B6=, C6=) angles.
Resolution: Use only absolute or incremental angles. If required, several G7 definitions with incremental angles behind one another can be defined.

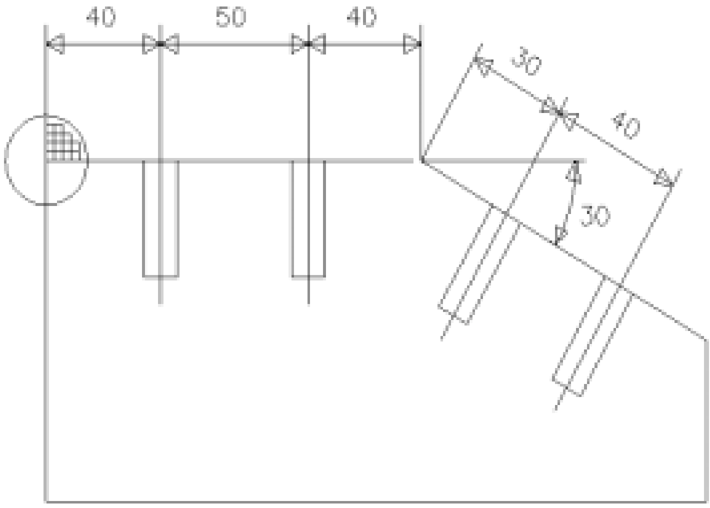
P307 Programmed plane not attainable
The defined G7 tilt position, owing to the restricted range of the rotary axis, cannot be attained.

MACHINE SETTINGS
MC312 free operating plane (0=off, 1=on)
activates the free operating plane. The G7 function can be programmed.

MC755 free operating plane: rotation (0=coordinates cross,1=axes)
If the desired rotation of the operating plane is compatible with the rotation of a rotary axis, an adjustment may be made to determine whether the relevant rotary axis or the coordinates cross is rotated.

e.g. on a machine with (real C axis) the programming G7 C5=30 and MC755=0 generates a rotation of the coordinates cross through -30° and, if MC755=1, a rotation of the C axis through 30°.

Example 1 Workpiece with tilted operating plane.



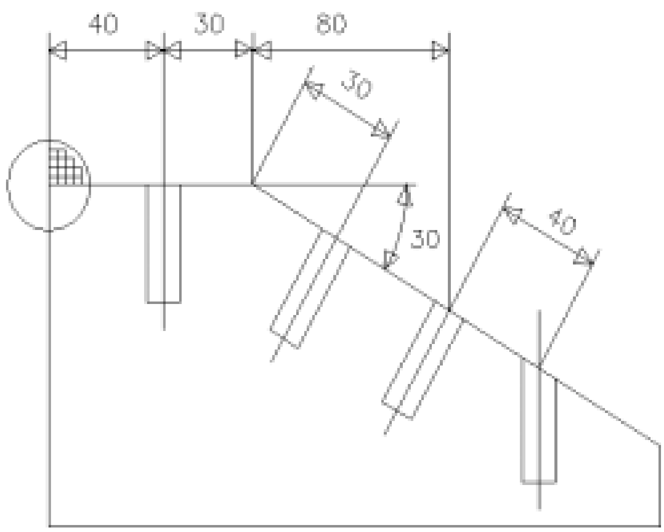
N10 G17
N20 G54
N30 M55
N40 G7 L1=1
N..
N100 G81 Y1 Z-30
N110 G79 X40 Z0
N120 G79 X90
N..
N200 G0 X130 Z50
N210 G93 X130
N220 G7 B5=30 L1=2 L50
N230 G79 X30 Z0
N240 G79 X70
N..
N300 G7 L1=2 L50

Explanation:

N10	define operating plane	
N20	null point insertion	
N30	deselection of M53/M54	
N40	reset G7	
N100	drill cycle definition	
N110	drill the first hole in the horizontal plane	
N120	drill the second hole in the horizontal plane	
N..	other movements in the horizontal plane	
N200	tool is set at the safety distance.	
N210	null point is set at the start of the tilted operating plane.	
N220	G7	define new operating plane
	B5=30	angle of rotation
	L1=2	tool/table is rotated about the tool tip
	L50	extra oversize in the direction of the tool. In this way, the tool is rotated about the null point. The distance from the tool tip to the null point is 50 mm.
N230	drill the first hole in the tilted operating plane	
N240	drill the second hole in the tilted operating plane	

N.. other movements in the tilted operating plane
N300 reverse rotation in the horizontal plane.

Example 2 Workpiece using tilted operating plane.



N10 G17
N20 G54
N30 M55
N40 G7 L1=1
N..
N100 T1 M6
N110 G81 Y1 Z-30
N120 G79 X40 Z0
N..
N200 T2 M6
N210 X70 Z50
N220 G93 X70
N230 G7 B5=30 L1=2 L50
N240 G1 X0 Z0
N250 X150
N..
N300 T1 M6
N310 G79 X30 Z0
N320 G93 X=80:cos(30)
N330 G79 X0 Z0
N..
N400 G93 X=40
N410 G0 X0 Z50
N420 G7 B5=0 L1=2 L50
N430 G79 X0 Z0
N..
N500 M30

Explanation:
N10 define operating plane
N20 null point insertion
N30 deselection of M53/M54
N40 reset G7

N100	change the drill	
N110	define the drilling cycle	
N120	drilling of a hole in the horizontal plane	
N..	other movements in the horizontal plane	
N200	change the mill	
N210	tool is set at the safety distance	
N220	null point insertion	
N230	G7	define new operating plane
	B5=30	angle of rotation
	L1=2	tool/table is rotated about the tool tip
	L50	extra oversize in the direction of the tool. In this way the tool is rotated about the null point. The distance of the tool tip from the null point is 50 mm.
N240	positioning of the mill in the tilted plane	
N250	mill in the tilted plane	
N..	other movements in the tilted operating plane	
N300	change the drill	
N310	drill the first hole in the tilted operating plane	
N320	null point insertion	
N330	drill the second hole in the tilted operating plane	
N..	other movements in the tilted operating plane	
N400	null point insertion	
N410	tool is set at the safety distance	
N420	G7	deselect "Tilt operating plane" reverse rotation in the horizontal plane
	B5=0	angle of rotation
	L1=2	tool/table is rotated about the tool tip
	L50	extra oversize in the direction of the tool. In this way, the tool is rotated about the null point. The distance of the tool tip from the null point it 50 mm.
N430	drill the third hole in the horizontal operating plane	
N..	other movements in the horizontal operating plane	
N500	end of program.	

23.7 Tilting of the operating plane (from V400)

23.7.1 Introduction

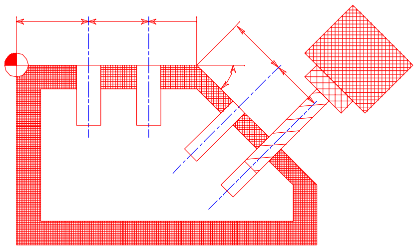
The control supports the tilting of operating planes on tool machines with tilting faces or tilting tables. Please consult your user handbook.

Typical applications, for example, are oblique drilling or contours which lie obliquely in the operating area. In this way, the operating plane is always tilted about an active null point. Normally, the operation is programmed in a principal plane, e.g. X/Y plane; however, it is executed in the plane which was tilted to the principal plane.

Consult the description of the G7 function for the programming of the freely programmable operating plane.

The tilting of the operating plane is defined and implemented using the G7 function. The G7 function is made up of two components:

- definition of new operating planes, rotation of the coordinate system.
- in the event that it is programmed, tilt the tool vertically to the defined operating plane.



An operation on an oblique workpiece plane is programmed in local coordinates. In this way, the local X and Y coordinates lie in the oblique plane and the Z coordinate is fixed vertically in the plane.
The machine recognizes the link between the local coordinates and the true machine axes and calculates this. The control calculates the tool correction factor.

Millplus distinguishes two machine types during tilting of the operating plane:

1) Machine with a tilting table
The location of the transformed machine axis is not changed with reference to the coordinates system fixed in the machine. If you rotate your table, and thus the workpiece, through 90°, for example, the coordinate system is not rotated through 90° with it. If, in the operating mode "Manual operation", you press the axis direction key Z+, the tool travels in the direction Z+.
2) Machine with a tilting face
the location of the tilted (transformed) machine axis is changed with reference to the coordinates system fixed in the machine:
If you rotate the tilting face of your machine and hence the tool, e.g. in the B axis about +90°, the coordinates system is rotated with it. If, in the operating mode "Manual operation", you press the axis direction key Z+, the tool travels in the direction Z+ and X+ of the coordinates system fixed in the machine.

Using the G7 function you define the location of the operating plane by the input of tilt angles. The angles entered describe the angular components of a space vector.

If you program the angular components of the space vector, the control automatically calculates the angular position of the tilt axes. MillPlus calculates the location of the space vector and thus the location of the spindle axis, by means of rotation about the coordinates system fixed in the machine. The sequence of rotations for the calculation of the space vector is fixed: MillPlus turns the A axis first, next the B axis and finally the C axis.

The G7 function is effective from the start of its definition in the program.

MillPlus can only position controlled axes automatically.
In the G7 definition, you can, in addition, input a safety distance to the tilt angles, with which the tilt axes are positioned.
Use only pre-set tools (full tool length in the tool table).
During the tilting process the position of the tool tip opposite the workpiece remains virtually unchanged (depending on the type of movement L1=).

MillPlus implements the tilting process using the power traverse.

23.7.2 Machine types

Milling machines with four or five axes can be used for the oblique machining of a workpiece.
Depending on the plane which is tilted, other types of machine are needed for the working. At least two rotary axes and three linear axes are needed, in order to reach all sides and planes (except the under surface) without the need for remounting.

The possible types of machine are:

90° tilting face and turntable
: The tilting face can be in two states. The upper and reverse sides can be worked by means of the tilting face. The four side surfaces can be worked using the turntable (C axis). The machine is only suited to all oblique operating planes if the tilting face can also be set in the oblique position manually.

Double turntable
: The tables (A and C axes) are stacked. In this way, all sides and oblique operating planes can be worked.

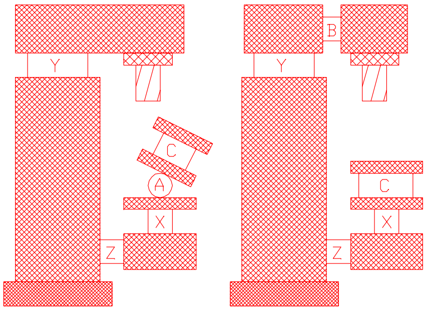
Double turntable and 45° tilting face
: The tables (A and C axes) are stacked. The A axis is limited in its angular movement. In conjunction with the two tilting face states all sides and oblique operating planes can be worked.

45° double turntable
: The tables (B and C axes) are stacked. The B axis is fixed in this way at an angle of 45°. All sides and oblique operating planes can be worked.

Turntable and rotating face
: The face (B axis) can be freely positioned. In conjunction with the table (C axis) all sides and oblique operating planes can be worked.

Turntable and 45° rotating face
: The face (B axis) is set at an angle of 45°. In conjunction with the table (C axis) all sides and oblique operating planes can be worked.

Outline of the most suitable machine types for use with oblique operating planes.

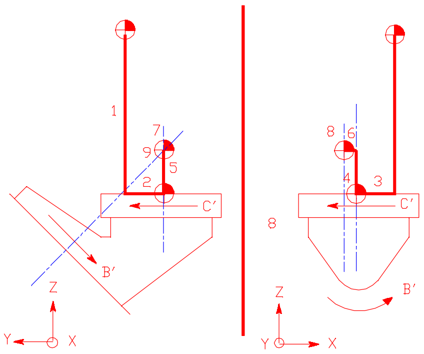


23.7.3 Kinematic model

The control needs a kinematic model of the machine in order to reset the programmed local coordinates in the oblique plane to the movements of the machine axes. A kinematic model describes the "construction" of the axes and the exact position of the different turning points on the rotary axes.

As an example, a kinematic model of the DMU 50 V machine is shown.The kinematic model is made up of a chain stretching from the work piece to the machine frame. It is not necessary to describe the chain from the workpiece to the machine frame, because it includes no rotary axes.

Kinematic model for the DMU 50 V



Explanation of the drawing:

-1,2,3	three elements in the X, Y, and Z directions in order to fix the (absolute) centre position of the workpiece table with reference to the marker positions.
-4	element for definition of the C axis. It is only necessary to describe the rotating axis of a rotary axis, not the centre point.
-5,6	two elements in order to obtain the rotating axis of the second (incremental) rotary axis.
-7	element for definition of the (incremental) direction of the second rotating axis. This direction is -45° in the A axis (all around the X axis).
-8	element for definition of the B axis.
-9	element in order to raise the -45° tilt (Element 7) again. In this way, the kinematic chain ends without rotation.

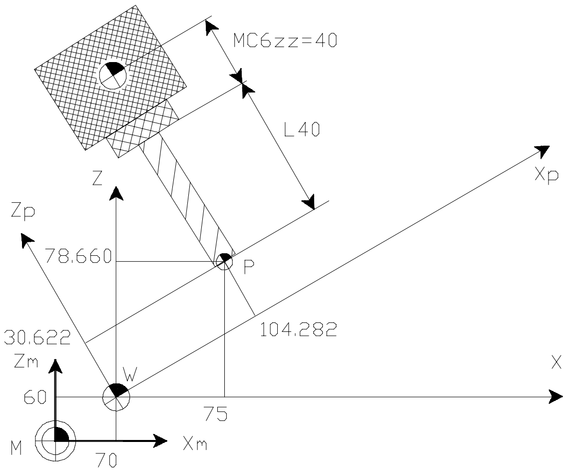
The kinematic model is entered by means of the machine settings MC600 to MC699.

To determine the connection between the operating planes and the positions of the axes, the stacking and the exact position of the different rotating points of the rotary axes are needed. A description of this stacking is called the kinematic model. The kinematic model is defined in the form of two "chains". One chain defines the axis stacking of the tool as far as the machine frame, the other chain from the workpiece to the machine frame. In this way, it is only necessary to describe a chain if it contains rotary axes.
A kinematic chain defines, by means of displacements and tiltings, the way in which the rotary axes lie with respect to one another. Every displacement or tilting is determined as an element of the

kinematic chain in three machine settings. Thus, a total of 25 elements of the kinematic chain can be determined. All rotary axes and positioning axes which are present should be described.

Only the machine types with rotary axes in the X, Y or Z direction, in which the sequence of rotary axes from workpiece to tool is as follows, are supported:

- A C
- C A
- C B



The position display on the screen can change between the position in the G7 plane (Xp,Zp) or in machine coordinates (X,Z).
Both are based on the active null point G52 + G54 + G92/G93.

23.7.6 Axis selection/positioning axis

An axis which is not regulated must be set to the correct position by hand. However, either before or after this, the oblique setting of the tool must also be entered by means of G7. Otherwise it will not be checked.

Comment: The expected position of the rotary axes is set parametrically in G7 using n7=<parameter number>. An axis selection or a positioning axis can be set manually using this information.

The axis selection or the positioning axis should also be followed in the kinematic model.

23.7.7 Reference point

If the reference point is approached during G7, the rotary axes remain in their reference position following the approach. The G7 plane is deactivated and the G17 plane is activated.

After running up the machine, but before approaching the reference point, the G7 plane is still active.
After < reset CNC> the G7 plane is deactivated.

23.7.8 Interruption

If the G7 movement is interrupted, the exact position of the tool tip is displayed on the screen. Following interruption, the axes can be used in manual operation [mode].

Following <Start> a move in position back to the interrupted point is effected. At the same time the axes run with positional logic corresponding with the G7 plane. Concurrently, the rotary axes rotate to the initial position.

23.7.9 Fault reports

P306 plane not clearly defined

The G7 plane is defined using a mix of absolute angles (A5=, B5=, C5=) and incremental angles (A6=, B6=, C6=).

Solution: Use only absolute or incremental angles. If necessary, several G7 definitions with incremental angles can be defined, one behind the other.

P307 program plane not attainable

The G7 oblique setting defined cannot be attained, on account of the limited range of the rotary axes.

Solution: Machines with a tilting face should tilt the face (by means of the M function) from the instantaneous position (horizontal or vertical) following the other position

O256 Machine type not recognized

The kinematic model in MC600 to MC699 is defining a type of machine which is not supported for the oblique operating plane (G7). Only machine types with the following sequence of rotary axes, as viewed from the workpiece to the to tool, are supported:

- A C
- C A
- C B
- C A fixed B -A fixed
  (A fixed is a fixed tilt in the direction of the A axis, as, for example, the DMU50V has with -45°)
- C
- Axis change variants (C becomes B, and B becomes C) are also possible.

Solution: The kinematic model should be entered in detail, with at least a description of those rotary axes present. The control must be run up once more.

23.7.10 Machine settings

MC312 free operating plane (0=off, 1=on)

activates the free operating plane. The G7 function can be programmed.

MC600 - MC699

There are 100 new machine settings (MC600 – MC699) for the description of the kinematic model. The model is described using a maximum of 25 elements, in which each element is described by means of four machine settings.
The following machine settings are used:

MC600 Kinematic chain (0=end,1=tool,2=workpiece)

MC601 Element (0,1=X,2=Y,3=Z,4=A,5=B,6=C)

MC602 Element type (0=incremental,
1=absolute)

MC603 Element insertion [:m/mdegrees]

MC604, 608, 612, 616, 620, …. , 696	as MC600
MC605, 609, 613, 617, 621, …. , 697	as MC601
MC606, 610, 614, 618, 622, ... , 698	as MC602
MC607, 611, 615, 619, 623, …. , 699	we MC603

MC755 Free operating plane: rotation (0=coordinates cross,1=axes)

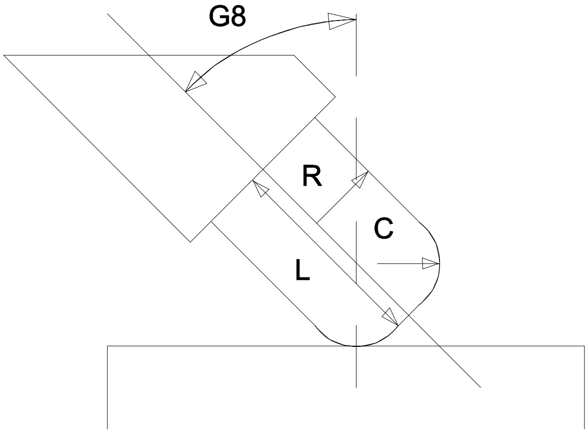
If the desired rotation of the operating plane corresponds with the rotation of a rotary axis, the control has the choice between rotten using the relevant rotary axis or rotation using the coordinates cross. This choice is made with MC755.
E.G on a machine with a (true) C axis the program instruction G7 C5=30 and MC755=0 produces a rotation of the coordinates cross over -30° and MC755=1 a rotation of the C axis over 30°.

23.8 Swivel tool G8 (from V410)

To program a swivelled tool for four or five-axis machines.

With the function "Swivel tool" the tool axis can be set obliquely relative to the machining plane. This makes angle milling possible and substantially improves cutting conditions and thus surface quality.

The programming of G8 is identical to G7. G7 should therefore also be read.



L, R and C from the tool table.

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

Parameters

G	Werkzeug nachführen
L	Werkzeuglängen-Aufmaß
A5=	Absoluter Raumwinkel
B5=	Absoluter Raumwinkel
C5=	Absoluter Raumwinkel
A6=	Inkrementaler Raumwinkel
B6=	Inkrementaler Raumwinkel
C6=	Inkrementaler Raumwinkel
A7=	E-Parameter für Position i
B7=	E-Parameter für Position i
L1=	0=Keine,1=Rundachse,2=WZ-S

Associated function

G FUNCTIONS NOT PERMITTED WITHIN G8

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

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

- Programming with A5=, B5= or C5= parameters. The absolute rotations about the corresponding positive axes are defined by this. The rotations are calculated as follows:
  1. the active G8 rotation is cancelled
  2. C5= rotation about the positive Z axis fixed with respect to the machine
  3. B5= rotation about the positive Y axis
  4. A5= rotation about the positive X axis

- Programming with A6=, B6= or C6= parameters. The incremental rotations about the corresponding current positive axes are defined by this. The rotations are calculated as follows:
  1. C6= rotation about the current G8 positive Z axis
  2. B6= rotation about the current G8 positive Y axis
  3. A6= rotation about the current G8 positive X axis

Programming is independent of the machine configuration. The plane rotation is calculated relative to the current zero point. The motion is dependent on the machine configuration.

SCANNING A CALCULATED ANGULAR POSITION

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

SWIVEL MOTION

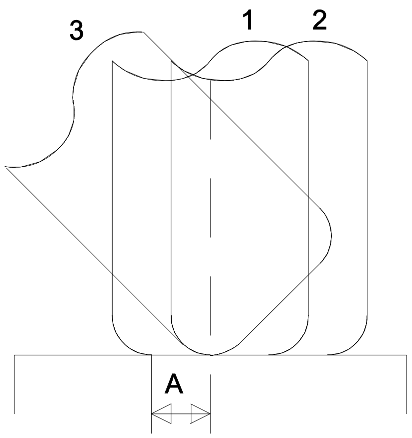
The G8 swivel motion is performed by interpolating with feed (F6=). It swivels the tool axis onto the defined plane. Which axes move depends on the type of motion L1= :

- L1=0 The rotary axes do not move (default).

Note: The swivel motion can be programmed or performed manually by means of the E parameters that are loaded with A7=, B7= or C7=.

- L1=1 Only the rotary axes swivel, while the linear axes do not move.
- L1=2 The rotary axes swivel and the linear axes perform a movement. This means that the contact point position remains X, Y, Z.

If the contact point lies on the tool corner radius, the movement is only a rotation.
If the contact point is the tool tip and the corner radius (C) is smaller than the tool radius (R), a compensating movement occurs so that the contact point is shifted from the tool tip to the corner radius.



With cylindrical cutters (with corner radius C < cutter radius R), the following particular point applies:
When swivelling from the vertical (1) to the oblique (2--> 3) position or vice versa, the contact point moves from the centre of the cutter to the corner radius (A) and vice versa. A compensating movement at the tool tip ensures that the current contact position X, Y, Z remains unchanged.

TOOL LENGTH_ALLOWANCE

If the swivel motion takes place about the tool contact point (L1=2), L defines an extra allowance in the tool direction between the rotation point and the tool tip.

TOOL CORRECTION

During the function "swivel tool" (G8) the values L, R and C for the tool are corrected.
This G8 tool correction is independent of G40, G41, G42, G43, G44 and is always effective.
At the beginning and end of the tool correction, a compensation movement is carried out if the corner radius (C) is smaller than the tool radius (R).
If the tool dimensions (L,R,C) change with G8 active, the current position of the linear axes is re-calculated.

TURNING OFF THE G8 FUNCTION

G8 remains active until it is cancelled. G8 is cancelled by programming G8 without angle parameters.

G8 is not cancelled by M30 or <program abort>. After the control is turned on, G8 is still active. After search for reference points or <CNC reset> G8 is cancelled.

Note: At the start any program that uses G8, we recommend that a G8 is programmed with no parameters. This ensures that the tool direction is always reset as the program is starting up (abort with swivelled tool and new start). Without this G8 at the beginning, the first part of the program is executed in the swivelled instead of the unswivelled plane.
The programming is similar to programming with G7/G17/G18 - different zero points or different tools.

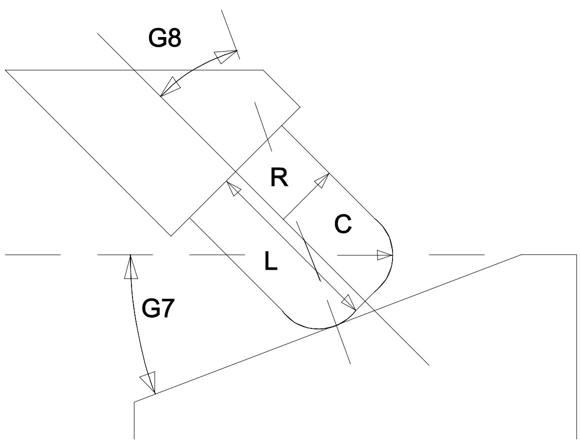
CONFIGURATION

Swivel tool (G8) can be used for machines where a kinematic model is defined and entered. See description of the kinematic model.

DISPLAY

When G8 is active, a yellow icon is displayed in the display behind the tool number.
A small 'p' on the right, next to the 'axis letters', is used to display whether the position of the tool tip is displayed or the position in machine coordinates.

Example 1 Workpiece with oblique machining plane and oblique tool.



N10 G17
N20 G54
N30 M55
N40 G7 L1=1
N50 G8 L1=1
..
N100 G0 X130 Z50
N110 G93 X130

N120 G7 B5=-30 L1=2
N130 G8 B5=30 L1=2
..
N200 G8
N210 G7 L1=2

Key:

N10	Define machining plane	
N20	Zero point offset	
N30	Deselect M53/M54	
N40	Reset G7	
N50	Reset G8	
N100	Tool set to safety distance	
N110	Zero point set to the beginning of the swivelled machining plane.	
N120	G7	Define new oblique position of the tool.
	B5=-30	Angle of rotation
	L1=2	Tool/table revolves about the tool tip
N130	G8	Define new oblique position of the tool.
	B5=30	Angle of rotation
	L1=2	Tool rotates about the tool tip and a compensation movement is performed.
N200	Turn tool perpendicular to the machining plane again (rotary and compensation movement).	
N210	Rotate back to the horizontal plane.	

23.9 Defining polar point (measurement reference point) G9 (from V320 onwards)

Programming a polar point. If a polar point has been programmed, program blocks with polar programming (angle and length) no longer relate to the zero point but to the last programmed polar point.

N.. G9 X.. Y.. {X90=...} {X91=...} {Y90=...} {Y91=...} {Z90=...} {Z91=...}

N.. G9 X0 Y0 Deactivate pole (same as workpiece zero point)

N.. G9 B2=.. L2=.. {B1=..} {L1=..} (polar point in polar coordinates)

Parameters

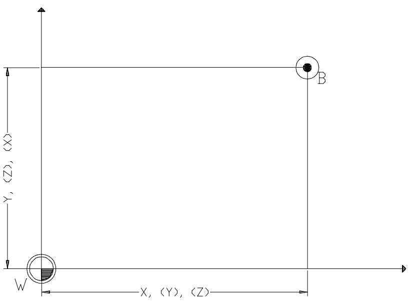
End point coordinates

```
G    Polpunkt definieren
X    Polkoordinate
Y    Polkoordinate
Z    Polkoordinate
B1=  Winkel
B2=  Polarwinkel
?90= Polkoordinate abs. (X,Y,Z
?91= Polkoordinate inkr. (X,Y,Z
L1=  Streckenlänge
L2=  Polarlänge
```

Wortweise Absolut-/Inkremental-Programmierung
X90=,Y90=,Z90= Polkoordinate absolut
)1=,Z91= Polkoordinate inkremental

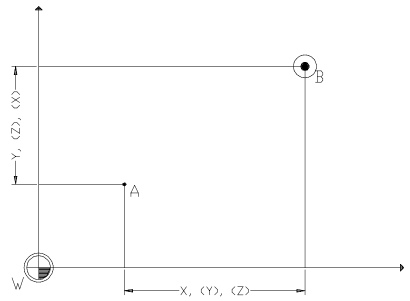
Notes and usage

Polar point in absolute coordinates:



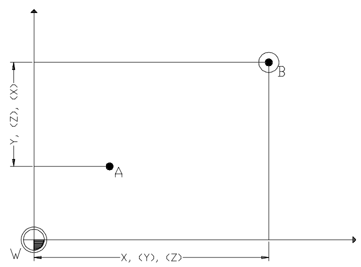
B = polar point
N.. G9 X.. Y..

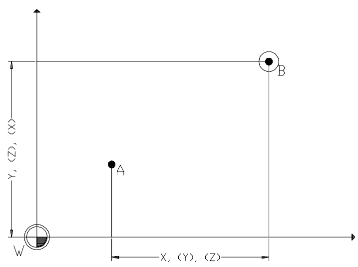
Polar point in incremental coordinates:



A = existing polar point B = new polar point
N... G9 X91=... Y91=...

Polar point in combined absolute/incremental coordinates:

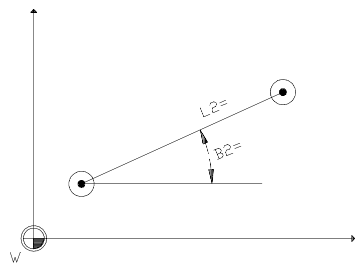




A = existing polar point B=new polar point
N... G9 X... Y91=...

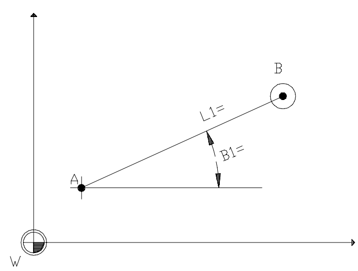
N.. G9 X91=.. Y..

Polar point in absolute polar coordinates:



A = existing polar point B = new polar point
N.. G9 B2=.. L2=..

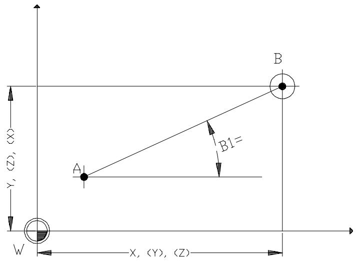
Polar point in incremental polar coordinates:



A = end point of last movement B = new polar point

N.. G9 B1=.. L1=..

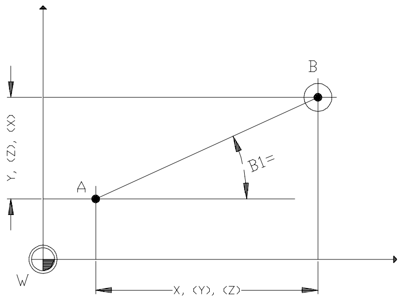
Combined programming: Cartesian absolute/polar:



A = existing polar point B = new polar point

N.. G9 X.. B1=..

Combined programming: Cartesian incremental/polar:



A = existing polar point B = new polar point

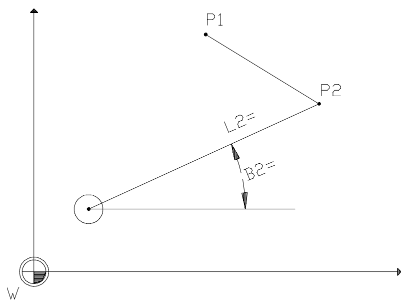
N.. G9 X91=.. B1=..

- pole definitions are only permitted in the active working plane
- before the G9 block is called, the polar point is located at the workpiece zero point (polar point = 0)
- when the plane is changed using G17, G18, G19, the polar point is zeroed (0).

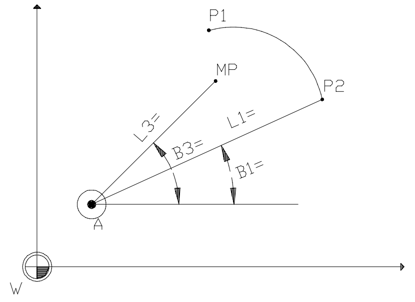
Polar end point definition:

During absolute polar programming polar length L2= or L3= and polar angle B2= or B3= no longer relate to the zero point, but to the polar point.

Polar point definition

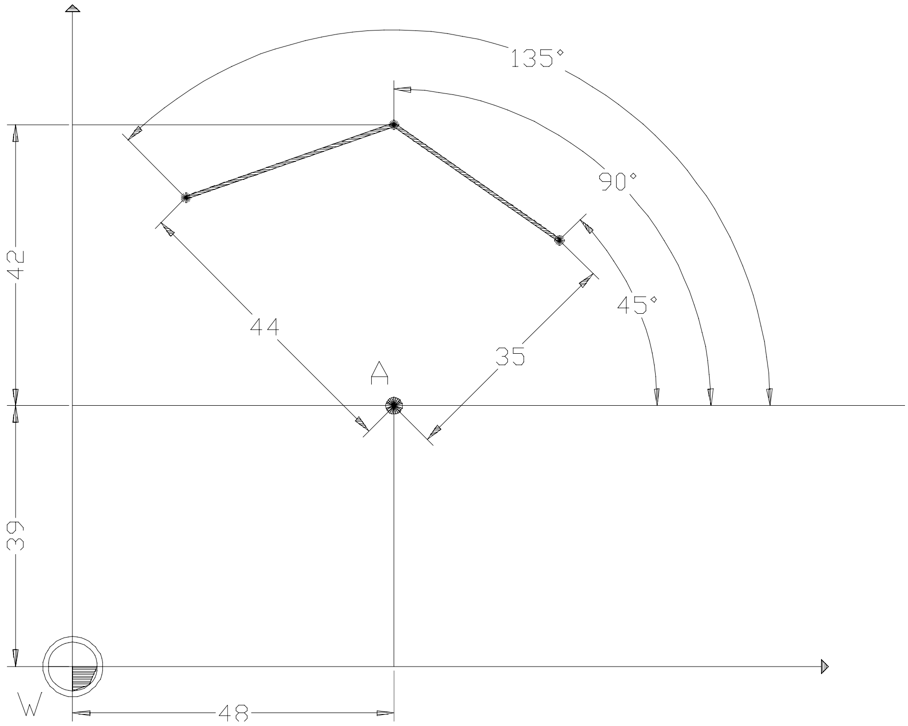


Polar circle definition
In G2 and G3 blocks polar programming with polar point of centre and end point is possible.



ICP/geometry calculation G64
G1, G2 and G3 blocks with B2=, B3= and L3= programming may be programmed in G64 and ICP. They relate to the active polar point. However, the polar point itself can only be changed in G64, **not** in ICP.

Example



A = new polar point

N30 G9 X48 Y39	Definition of new polar point
N40 G1 B2=135 L2=44	Definition of end point coordinate related to the new polar point
N50 G1 B2=90 L2=42	
N60 G1 B2=45 L2=35	

23.10 Polar coordinate, Rounding, Chamfering G11

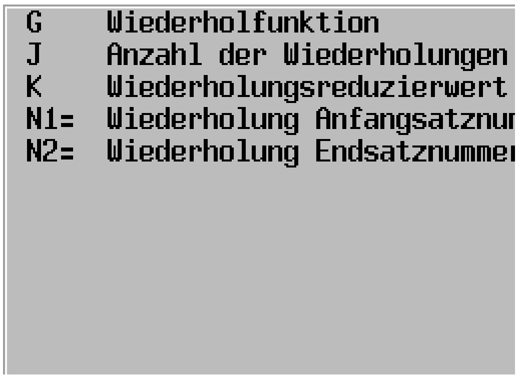
The use of the function is limited to programs that have been created with previous types of controller.

Programs that require geometric calculations can be easily created by the user with the Interactive Contour Programming (ICP).
(See chapter on Interactive Contour Programming)

23.11 Repeat function G14

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

Parameters



Beispiel
Example

Repear program blocks N12-N19 four times. (2 methods)

:
N12
:
N19
:
N90 G14 N1=12 N2=19 J4
Repeat program blocks N12-N19 four times
:

:
N5 E2=4
:
N12
:
N19
:
N90 G14 N1=12 N2=19 E2
Repeat program blocks N12-N19 four times
:

Note

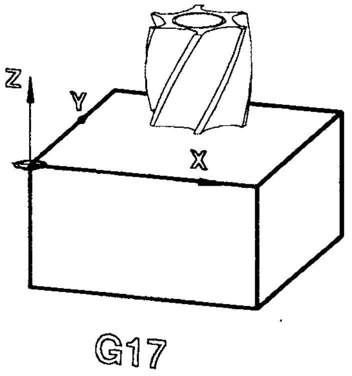
The block numbers of N1=.. and N2=.. must both be in the same part program or subprogram.
If N2=is not programmed, only the block marked N1= will be repeated.

If parameters J and E are not programmed, the block sequence will only be repeated once. A repeating block sequence can be contained in another repeating block sequence (can be nested four times).

A repeat only takes place in a G14 block if E>0. If the K parameter is not programmed, the CNC uses the standard value K1.

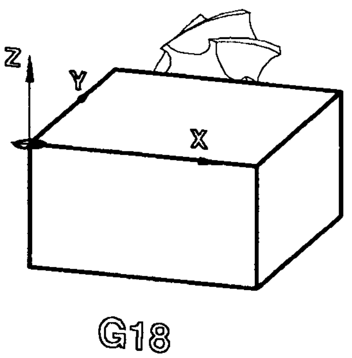
23.12 Main plane XY, tool Z G17

N... G17



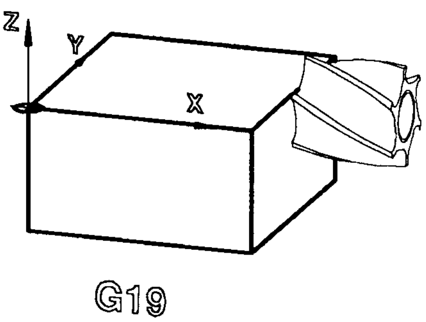
23.13 Main plane XZ, tool Y G18

N... G18



23.14 Main plane YZ, tool X G19

N... G19



23.15 Macro call G22

Call subprogram:
N... G22 N=..

Call subprogram if E..>0:
N... G22 E.. N=.. {E..=..}

Parameters

G	Makroaufruf
E	Parameterdefinition
N=	Makronummer

Example

PM	MM	MM
N9451	N9001	N9002
N1	N1	N1
N2	N2	N2
N3 G22 N=9001	N3 G22 N=9002	N3
N4	N4	N4
N5	N5	N5
	N6	N6
	N7	N7

Note

A subprogram can be called from another subprogram (can be nested eight times).

23.16 Main program call G23

N.. G23 N=..

Parameters

G	Hauptprogrammaufruf
N=	Programmnummer

Example

```
PM
N9451
N1
.....
N3 G23 N=9001
N4
:
N50 M30
```

```
PM
N9001
N1
N2
N3
N4
N5
:
N200 M30
```

Notes

The main program or subprogram must not contain any G23 functions, so it may not be nested.

Programs larger than 100 KByte must not contain any jump commands.

23.17 Enable/disable feed and spindle override G25/G26

Activates (G25) or deactivates (G26) the feed and spindle override, for the purpose of the programmed feed and spindle movements. This is fixed at 100% with the feed and spindle override deactivated.

Enable feed and spindle override:
N... G25

Disable feed override (F=100%):
N... G26 I2=1 or without I2=

Disable spindle override (S=100%):
N... G26 I2=2

Disable feed and spindle override (F and S= 100%):
N... G26 I2=3

Parameters

Example

N66 G26 I2=1	Deactivate feed override, i.e. fix at 100%
:	
N70 G25 I2=2	Activate feed override
:	
N68 G26 I2=3	Deactivate feed and spindle override that is to say F and S fixed at 100 %
:	
N70 G25	Activate feed override and spindle override

Note

Reactivate feed override and spindle override using G25, M30, soft key Cancel program or soft key Clear control.

23.18 Reset/activate positioning functions G27/G28

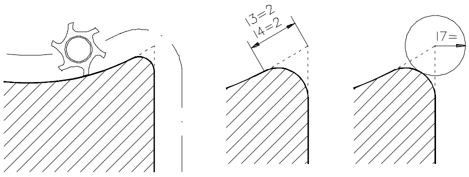
23.18.1 Positioning functions G27/G28 (to V320)

Reduces the corner radius that occurs due to tool lag (lag error) on change of direction.

Activate:
N... G28 {I3=...} {I4=...} {I5=...} {I6=...} {I7=..}

Delete individual parameter:
N... G28 {I3=0} {I4=0} {I5=0} {I6=0}

Delete all parameters:
N... G27



Without In-Position:
The next movement is started when the setpoint position is reached. This may cause corner radii.

With In-Position:
The next movement is only started after all axes have reached the programmed positions.

1. G28 without parameter	
G1,G2,G3 with In-Position	G28
2. Movement with feed	
G2,G3 without In-Position (initial setting)	G28 I3=0
G1,G2,G3 with In-Position	G28 I3=1
G1,G2,G3 with corner release distance (MC136)	G28 I3=2
G1 with programmable contour accuracy	
- Contour accuracy (MC137)	G28 I3=3
- Programmable contour accuracy I7=.. [0-10000mm]	G28 I3=3 I7=..
3. Rapid traverse movements G0	
G0 with In-Position (initial setting)	G28 I4=0
G0 without In-Position	G28 I4=1
G0 with corner release distance (MC136)	G28 I4=2

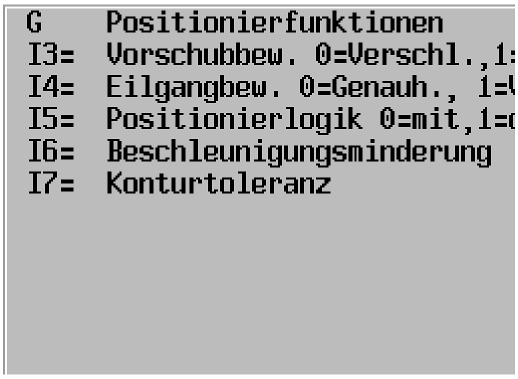
4. Positioning logic with G0

G0 with positioning logic (initial setting)	G28 I5=0
G0 with positioning logic	G28 I5=1

5. Feed limiting for circular movements

G2, G3 with standard value (initial setting)	G28 I6=0
G2, G3 with standard value (MC135)	G28 I6=1

Parameters



23.18.2 Look Ahead Feed (from V320 onwards)

Look Ahead Feed is used for precalculation on the programmed tool path, while taking account of the dynamics of all axes involved. The path speed is adjusted to achieve the highest contour accuracy at the highest possible speed. The programmed feed is, however, never exceeded.

Taking the programmed feed and actual feed override settings into account, special high-performance algorithms ensure a homogeneous feed at fast processing times.

With respect to Look Ahead Feed users need not pay attention to anything else.
This function cannot be influenced.

Existing programs need not be adjusted. They can be run as before.
During Look Ahead Feed the end point and centre of a circle should match to within 64 µm. In this case the centre is corrected automatically. Unlike V310 there is no "compensation movement" at the end point. An error message follows if the end point and centre do not match to within 64 µm. The same applies also to helix interpolation.

The running speed of CAD-generated programs is increased substantially.

Only the G28 function has been changed. The addresses for feed limitation were cancelled (refer to G27/G28 from V320 onwards).

23.18.3 Positioning functions G27/G28 (from V320 onwards)

1. G28 without parameter

G1,G2,G3 with In-Position	G28

2. Movement with feed

G1,G2,G3 without In-Position (initial setting)	G28 I3=0
G1,G2,G3 with In-Position	G28 I3=1

3. Rapid traverse movements G0

G0 with In-Position (initial setting)	G28 I4=0

G0 without In-Position G28 I4=1

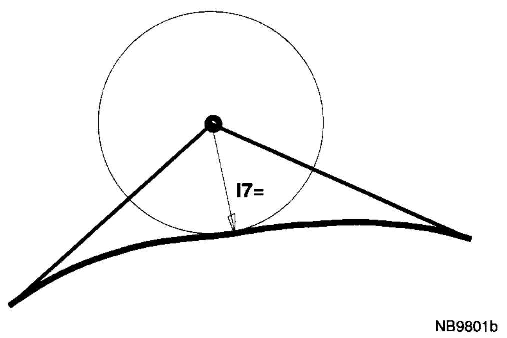
4. Positioning logic with G0

G0 with positioning logic (initial setting)	G28 I5=0
G0 with positioning logic	G28 I5=1

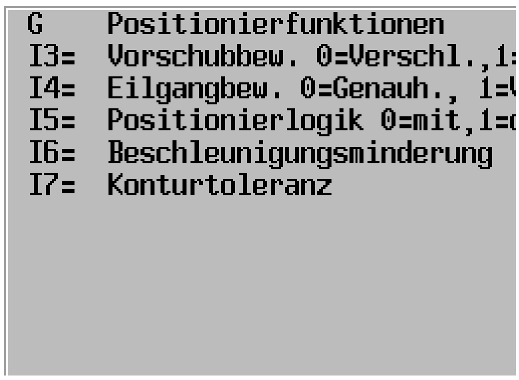
5. Movements with user-definable contour accuracy

G0,G1,G2,G3
-contour accuracy (MC765)
-user-definable contour accuracy
I7=... (0-10000 mm) G28 I7=...

User-definable contour accuracy (rapid motion and feed)



Parameters


G Positionierfunktionen
I3= Vorschubbew. 0=Verschl.,1:
I4= Eilgangbew. 0=Genauh., 1=V
I5= Positionierlogik 0=mit,1=o
I6= Beschleunigungsminderung
I7= Konturtoleranz

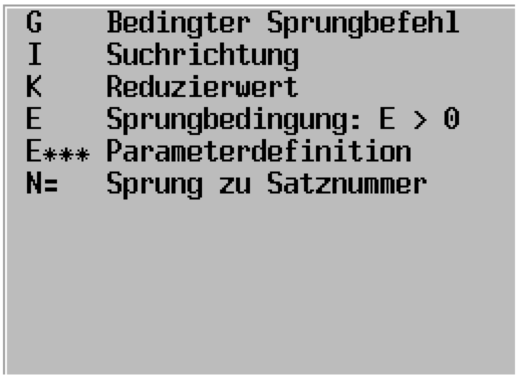
Note

G28 I3= is only active in G74

23.19 Conditional jump G29

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

Parameters



Example

:	
N50 E2=3	Parameter E2 will have value 3
N51	
:	
:	
N100 G29 E2 N=51	At E2 > 0 there will be a jump to N51; E2 is reduced by 1. At E2=0 the program run is continued after N100.
:	

Note

The value of the E parameter is reduced by the value of the K address. The E parameter is used as the new branch condition.
If the K address is not programmed, the E parameter is reduced by 1 after each branch.

Forward and backward jumping is possible in a subprogram or program. This is controlled by parameter I. If I=1 or I=0, searching will only be in forward direction. If I=-1 or no value is shown, there will first be a jump backwards to the beginning of the subprogram or program, which is followed by forward searching for the block number.

23.20 Activate/deactivate offset G39 (from V320 onwards

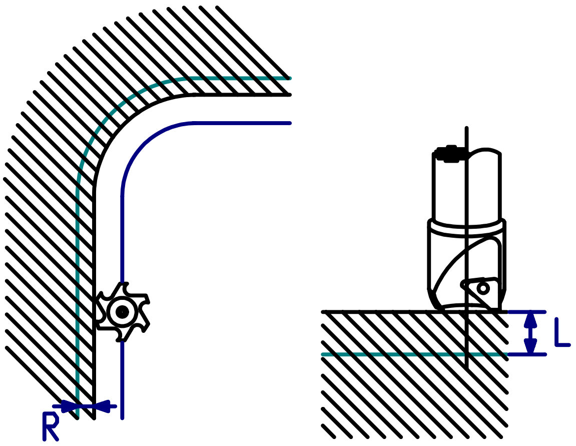
The programmed contour may be changed by offset.

Activate offset:
N... G39 {R...} {L...}
R: tool radius offset
L: tool length offset

Deactivate:
N... G39 L0 and/or R0

Parameters

Notes and usage



Changes made to the tool length offset will be activated with the next in-feed movement.

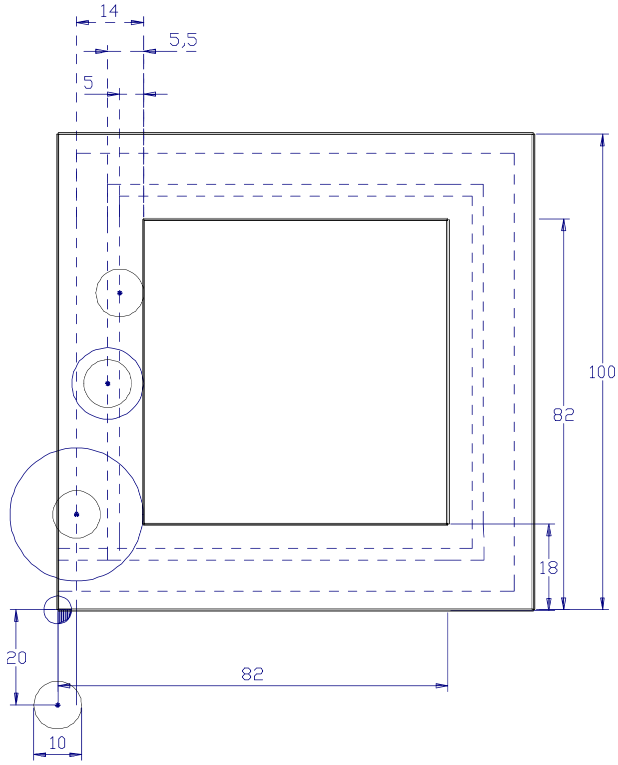
The tool radius offset is only active with active cutter radius compensation.

With inactive cutter radius compensation, tool radius offset changes will be activated when cutter radius compensation (G41/G42, G43/G44) has been activated.
With activated cutter radius compensation, tool radius offset changes will be corrected linearly over the entire path in the next movement block.

Note:
Tool radius offset is suppressed when the following functions have been activated: G6, G83-G89, G141, G182. Length offset remains active. Offset programming should be deactivated before these functions.

Example

Rectangular milling by roughing twice and finishing once

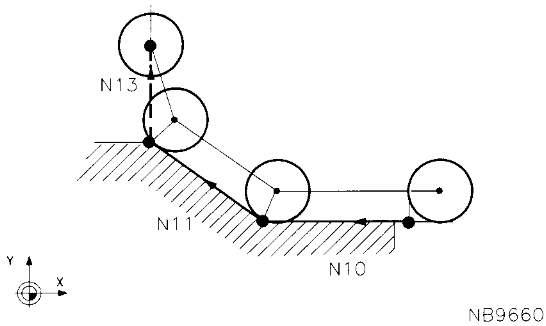


N39001	
N1 G98 X-10 Y-10 Z10 I120 J120 K-60	
	Define the graphic window
N2 G99 X0 Y0 Z0 I100 J100 K-40	
	Define the material
N3 T1 M6	Change tool (cutter radius: 5 mm)
N4 G39 L0 R9	Activate tool radius offset. The offset is 9 mm. (cutter radius for radius compensation is (5+9 =) 14 mm).
N5 F500 S1000 M3	Activate feed and spindle speed
N6 G0 X0 Y-20 Z5	Approach start position
N7 G1 Z-10	Move to depth
N8 G43 X18	Approach contour with radius compensation
N9 G41 Y82	First roughing of the rectangle
N10 X82	
N11 Y18	
N12 X0	
N13 G40	Turn off radius compensation
N14 G39 R0.5	Change tool radius offset. The offset is 0.5 mm. (cutter radius for radius compensation is (5+0.5 =) 5.5 mm)
N15 G14 N1=8 N2=13	Repeat the rectangle (2nd roughing movement)
N16 G39 R0	Change tool radius offset. The offset is 0 mm. (cutter radius for radius compensation is 5 mm)
N17 G14 N1=8 N2=13	Finish the rectangle
N18 G0 Z10	Retract tool from material
N19 M30	End of program

23.21 Cancel tool radius compensation G40

N.. G40

Example



:

:
N9 G42 Activate radius compensation to the right
N10 G1 X..
N11 X.. Y..
N12 G40 Cancel radius compensation
N13 G0 Y..
:

Notes

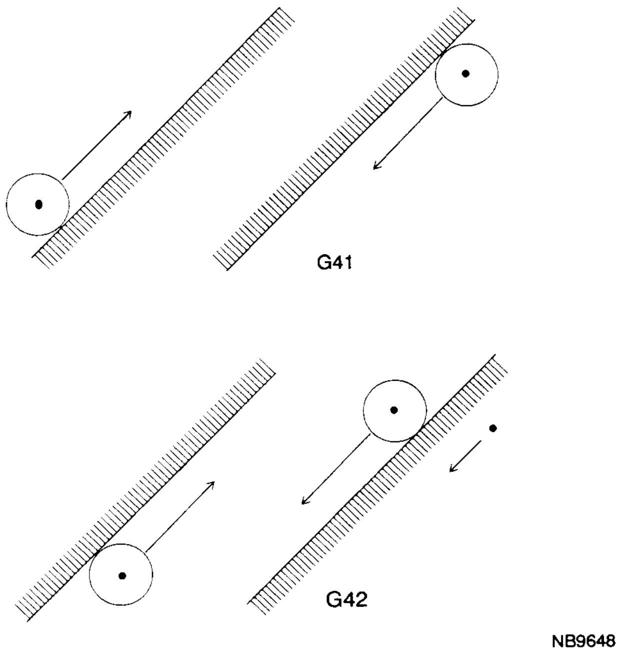
G40 comes in to effect automatically after:

- Switching the controller on
- Softkey Clear control
- Softkey Cancel program
- M30

23.22 Tool radius compensation (left/right) G41/G42

N.. G41/G42

In both cases, the view direction is the direction of tool movement.



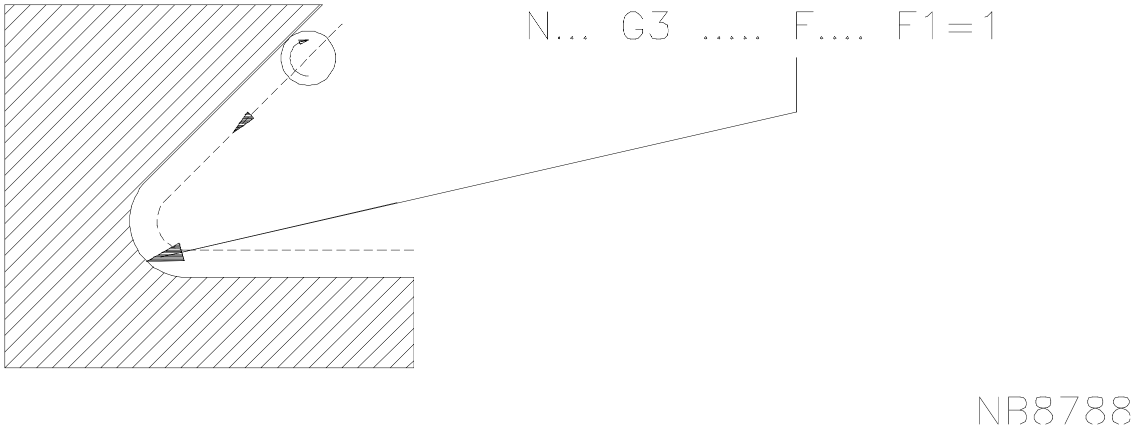
Constant feed for radius compensation of circles

Parameter F1= is used to keep the programmed feed on the workpiece contour constant regardless of the cutter radius and contour shape.

F1=0 feed not constant (switch-on condition, M30, Cancel program softkey or after Clear control softkey). The programmed feed should represent the tool tip speed.

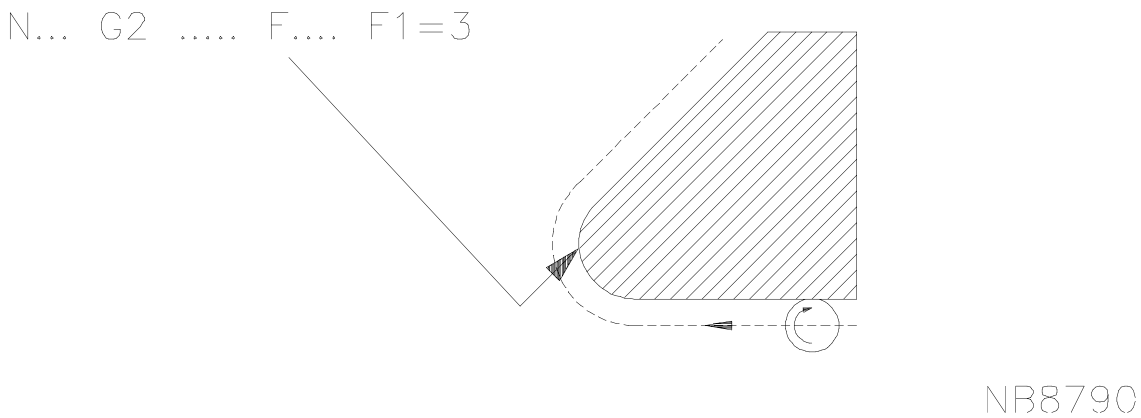
* = feed too large ** = feed too small

F1=1 constant feed only on the inside of circular arcs. The programmed feed is reduced to ensure that the tool tip moves along the inside of a circular arc at reduced speed.

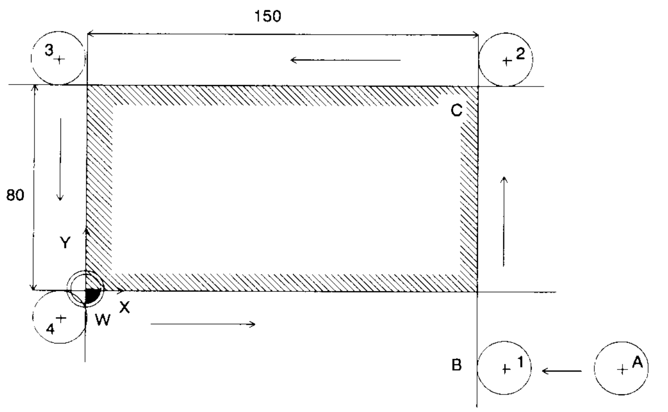


F1=2 constant feed on the inside and outside of circular arcs. The programmed feed is reduced (inside of circular arcs) or increased (outside of circular arcs) to ensure that the tool tip moves at the calculated new speed. If the increased speed exceeds the maximum feed defined by a machine constant, the maximum feed is used.

F1=3 constant feed only on the outside of circular arcs. The programmed feed is increased to ensure that the tool tip moves along the outside of a circular arc at the increased speed.



Example

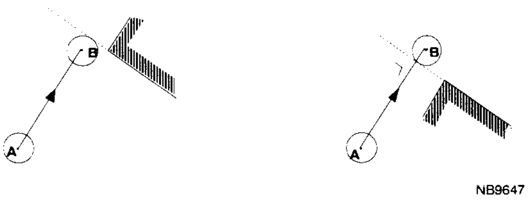


NB8602

N9999	
N1 G17	
N2 G54	
N3 T1 M6	Change tool
N4 G0 X200 Y-20 Z-5 S500 M3	Spindle start, move tool to X120,Y-20 at traversing speed
N5 G43	Radius compensation to end point
N6 G1 X150 F150	
N7 G42 Y80	Activate radius compensation to the right
N8 X0	
N9 Y0	
N10 X150	
N11 G40	Cancel radius compensation
N12 G0 X200 Y-20	

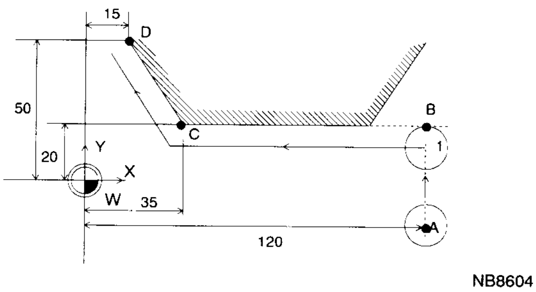
23.23 Tool radius compensation to end point G43/G44

N.. G43/G44



G43 G44

Example



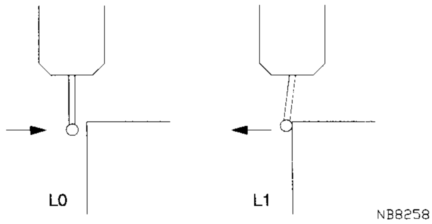
:	
N40 G0 X120 Y-15 Z10	
N41 G1 Z-10 F500	
N42 G43 Y20	Radius compensation to end point
N43 G41 X35	Activate radius compensation to the left
N44 X15 Y50	
:	

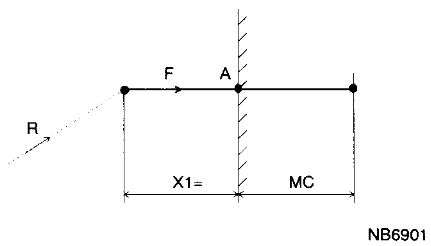
23.24 Measuring a point G45

Determines coordinate values with the measurement probe. The clamping position of the work piece and the work piece dimensions can be measured. The measurement results can be further processed by G49 and G50. The freely programmable measurement cycle G145-G150 can be used as an alternative to G45.

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

The plane of the rotary table is determined by definition of the 4th axis in the machine constant list. (MC117 should be 4 and MC118 should be B(66) or C(67)). L relates to the 4th axis B or C. Rotary axis A is not allowed.





Parameters

Meßposition
X,Y,Z Meßpunktkoordinate
Endwinkel vom Meßpunkt
Punktedefinitionsnummer

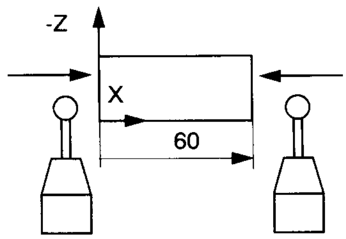
Meßparameter
I Meßrichtung in X
Meßrichtung in Y
Meßrichtung in Z
Meßrichtung in Rund-Achse
Meßpunktvorlauf

Meßergebnisse
E Parameternr. gemessene Koordinate
Punktnr. für gemessene Koordinate

Wortweise Absolut-/Inkremental-Programmierung

X90=,Y90=,Z90= Meßpunkt absolut
0=,C90= Winkel absolut vom Meßpunkt
1=,Z91= Meßpunkt inkremental
1=,C91= Winkel inkremental vom Meßpunkt

Examples



Measuring a point in the X axis :

Measuring in the positive direction
N.. G45 X0 Y20 Z-10 I1 E1 N=1
Measure point, calculate measurement position, store in point memory N= or in parameter E1.

Measuring in the negative direction
N.. G45 X60 Y20 Z-10 I-1 E1 N=1

Notes

- Only one axis coordinate can be measures with a G45 block.
- Measurements can only be made in the negative direction in the tool axis.
- The spindle speed must not be activated or switched on.
- Locate block.

N105 ...
N110 G148 E20
N115 G29 E21=E20=2 E21 N=125
N120 G45/G46
N125 ...

The tool type Q3=9999 can be entered for the measurement key.
M27 Activate measuring probe.

M28 Turn off measuring probe.

Example: P5 T5 Q3=9999 L150 R4
When tool T5 is called, the controller recognises that this tool is the measuring probe. The “spindle on” function (M3, M4, M13, M14) is suppressed and a fault message is issued.

Function G45 operates only parallel to the axis. The function of G145 has improved and now includes measurement unparallel to the axis. We therefore recommend that you should use the new basic measurement movement G145.

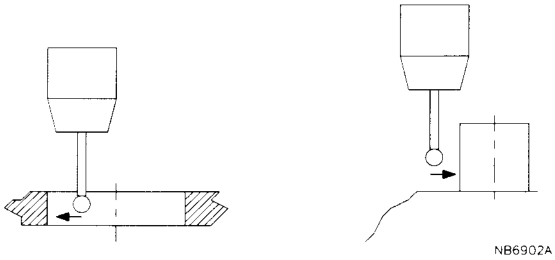
The difference between the measured and the programmed coordinate is calculated and stored internally to be used in operation with G49 or G50.

23.25 Measuring a circle G46

Measures a full circle (internally or externally) with a 4-point measurement. The measurement can be processed further by G49 or G50.

Measure inner circle:
N.. G46 [circle centre point coordinates] R.. {I+1 J+1} {I+1 K+1} {J+1 K+1} {F..} {X1=..} {P1=..} N=.. E..

Measure outer circle:
N... G46 [circle centre point coordinates] R.. {I-1 J-1} {I-1 K-1} {J-1 K-1} {F..} {X1=..} {P1=..} N=.. E..



Parameters

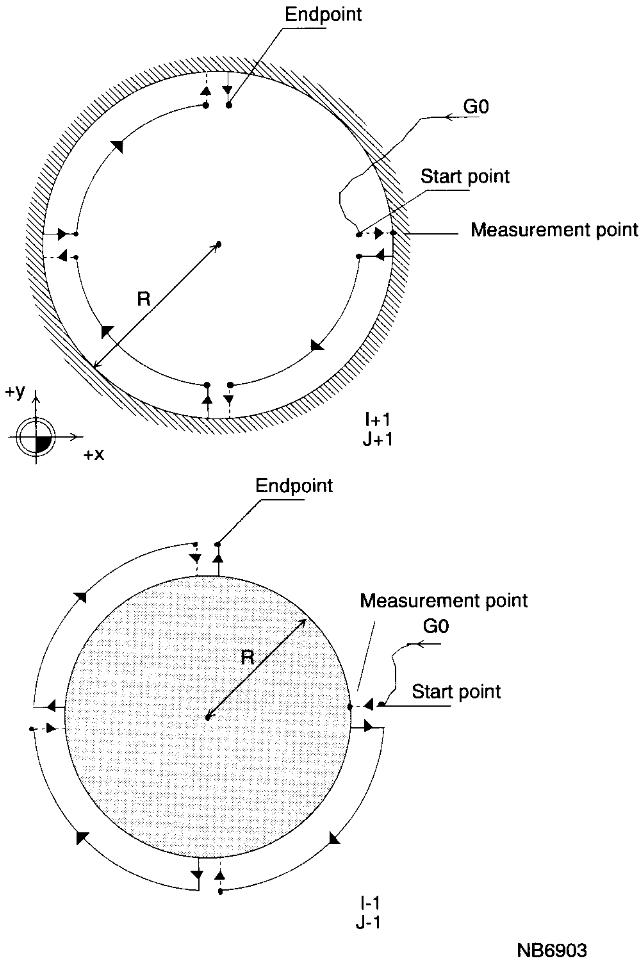
Kreisparameter
X,Y,Z   Lochkreismittelpunkt
Endwinkel vom Meßpunkt
Punktedefinitionsnummer
Kreisradius

Meßparameter
I   Meßrichtung in X
Meßrichtung in Y
Meßrichtung in Z
Vorschub zwischen den Meßpunkten
Meßpunktvorlauf

Meßergebnisse
E   Parameternr. gemessener Radius
Punktnr. gemessener Mittelpunkt

Wortweise Absolut-/Inkremental-Programmierung
X90=,Y90=,Z90=   Mittelpunkt absolut
0=,C90=   Winkel absolut vom Meßpunkt
1=,Z91=   Mittelpunkt inkremental
1=,C91=   Winkel inkremental vom Meßpunkt

Example



Measurement of an internal and an external circle in the XY level:

Internal circle:

N... G46 X30 Y25 Z20 I+1 J+1 R12.5 F3000 N=59 E24 — Measure circle, store centre in point memory N=59 and radii in parameter memory E24.

External circle:

N... G46 X30 Y25 Z20 I-1 J-1 R20 F3000 N=58 E23

Level	Internal circle		External circle	
XY (G17)	I+1	J+1	I-1	J-1
XZ (G18)	I+1	K+1	I-1	K-1
XZ (G19)	J+1	K+1	J-1	K-1

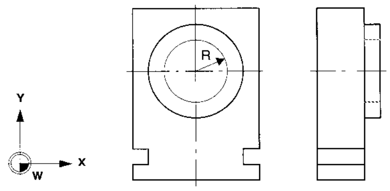
23.26 Calibrating the measuring probe G46 + M26

The measuring probe radius is determined by touching the calibration ring. The controller calculates the probe radius from the measured radius of the calibration ring and the programmed radius. The new radius value is stored in tool memory.

The centre point coordinates and the radius of the calibration ring are entered as machine constants.

Measuring the internal gauge ring:
N... G46 {I+1 J+1} {I+1 K+1} {J+1 K+1} {F...} {X1=...} M26

Measuring the external gauge ring:
N... G46 {I-1 J-1} {I-1 K-1} {J-1 K-1} {F...} {X1=...} M26



Parameters

I+/-1	Measurement in the X-Achse
J+/-1	Measurement in the Y-Achse
K+/-1	Measurement in the r Z-Achse
F	Feed during measurement movement
X1=	Predetermined distance

Example

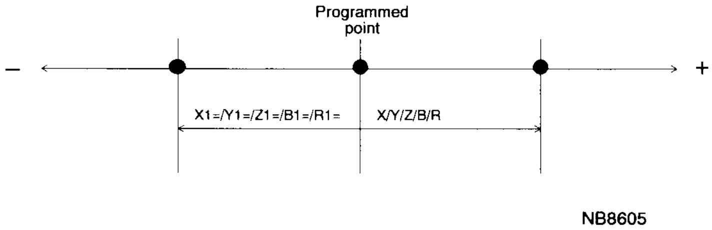
N46002
N1 G17
N2 T1 M6
N3 D207 M19 defined spindle stop
N4 G46 I1 J1 M26 calibrate measuring probe, store measuring probe radius for T1 in tool memory
N5 Z200 M30

23.27 Checking on tolerances G49

Compares whether the difference between the programmed value and the value measured during the G45 or G46 block lies within the dimensional tolerance limits.
If the difference lies within the tolerance limits, program processing continues.
If the difference lies outside the tolerance limits, the following options are available:

Repetition of the program section:
N.. G49 {X.., X1=..} {Y.., Y1=..} {Z.., Z1=..} {B.., B1=..} {R.., R1=..} N1=.. N2=.. {E..}

Conditional branch:
N.. G49 {X.., X1=..} {Y.., Y1=..} {Z.., Z1=..} {B.., B1=..} {R.., R1=..} N=.. E..



The measurement point must lie between the upper limit dimension (X/..) and the lower limit dimension (X1=/..) of the tolerance range.

Parameters

Toleranzwerte
X Positiver Toleranzwert in X
Negativer Toleranzwert in X
Positiver Toleranzwert in Y
Negativer Toleranzwert in Y
Positiver Toleranzwert in Z
Negativer Toleranzwert in Z
Positiver Toleranzwert in B
Negativer Toleranzwert in B
Positive Toleranz für Kreisradius
Negative Toleranz für Kreisradius

Bedingter Sprung
E Sprungbedingung: E > 0
Sprung zu Satznummer

Wiederholung eines Programmteils
N1= Wiederholung Anfangsatznummer
Wiederholung Endsatznummer

Example

```
:
N10  G49  R.02  R1=2  E1  N=13
N11  G49  R2  R1=.02  N1=1  N2=6
:
```

N10 1st tolerance comparison:
If the upper tolerance limit (R0.02) is exceeded (bore too large), a branch to block N13 takes place. The lower tolerance limit must not be reached (conditional branch).

N11 2nd tolerance comparison:
If the lower tolerance limit (R1=0.02) is exceeded (bore too small), the program section between N1 and N6 is repeated. The upper tolerance limit must not be reached (repetition of program section)

Note

Where there are two consecutive G49 blocks in the program, it must be ensured that the conditional branch is in the first block and the program section repetition is in the second block (otherwise an error message will appear!)

23.28 Processing measuring results G50

Changes the zero point offset or tool dimensions according to the correction values derived from the measured difference values.

Zero point offset correction:
With standard zero points or MC84=0:
N.. G50 {X1} {I..} {Y1} {J..} {Z1} {K..} {B1} {C1} {C2} {B1=} {C1=} {L..} N=..

With extended zero points and MC84>0:
N.. G50 {X1} {I..} {Y1} {J..} {Z1} {K..} {B1} {C1} {C2} {B1=} {C1=} {L..} **N=54.00 .. 54.99**

Tool length correction:
N.. G50 T.. L1=1 {I..} {J..} {K..} {T2=..}

Tool radius correction:
N.. G50 T.. R1=1 {X1=..} {T2=..}

Parameters

Nullpunktverschiebungen
N= Meßwert NPV G52-G59 verrechnen
X1: Nullpunktverschiebung in X
Y1: Nullpunktverschiebung in Y
Z1: Nullpunktverschiebung in Z
B1: Nullpunktverschiebung in B
C1: Nullpunktverschiebung in C
Korrekturfaktor für X
Korrekturfaktor für Y
Korrekturfaktor für Z
Korrekturfaktor für Rund-Achse
Prog.Winkel in B nach Verrechnung
Prog.Winkel in C nach Verrechnung

Werkzeugmaße

Notes

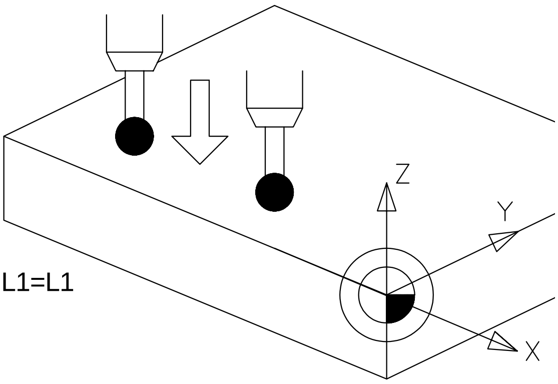
Machine configurations (B1,C1,C2)

B-axis B1: All that is required to align a clamped work piece with a rotary table (B-axis) rotating about the Y-axis is to measure two points on the X-axis:

-the angle of rotation is relative to the X-axis.

-the tool rotates about the Y-axis.

-the tool axis with the probe is the Z-axis or the Y-axis.



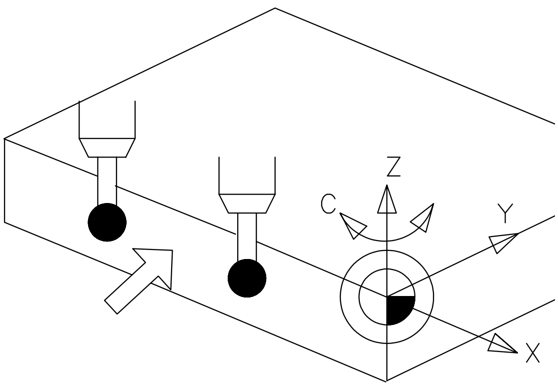
L1=L1

C-axis C1: All that is required to align a clamped work piece with a rotary table (C-axis) rotating about the Z-axis is to measure two points on the X-axis:

-the angle of rotation is relative to the X-axis.

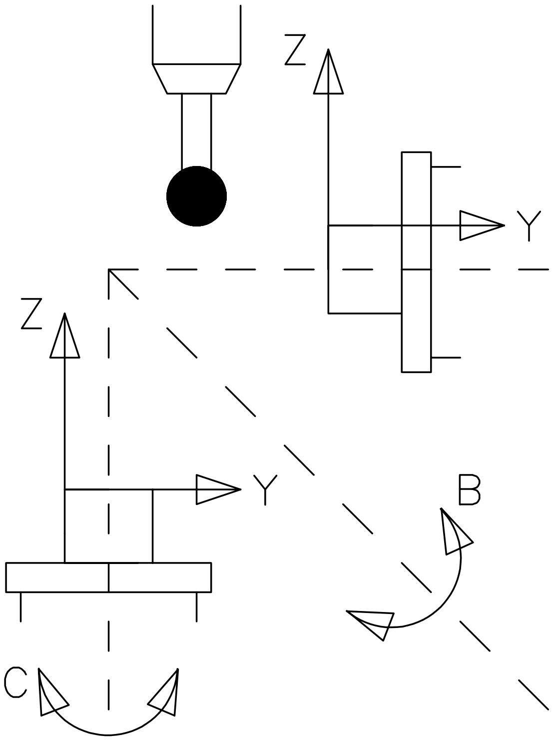
-the tool rotates about the Z-axis.

-the tool axis with the probe is the Z-axis.

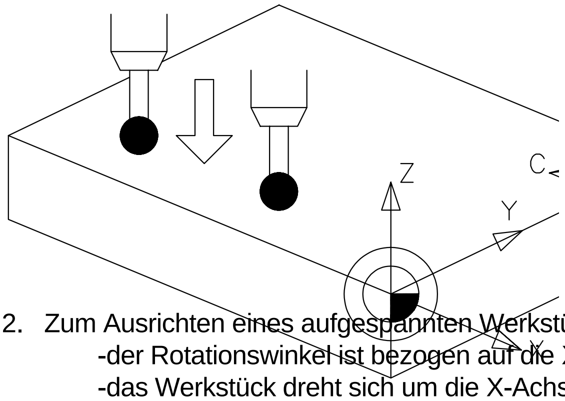


R1=R1

C-axis C2: This is an extension of the C1 option:

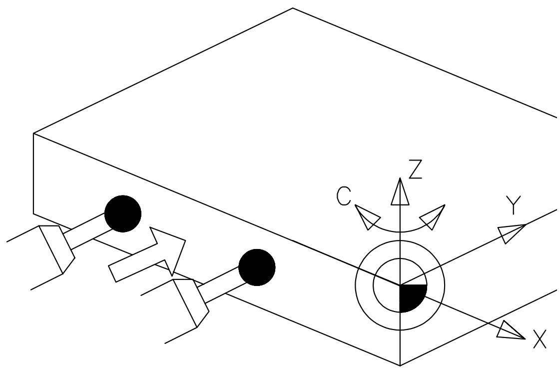


1. The C-axis is rotated through 90 degrees and rotates about the Y-axis instead of the Z-axis.
All that is required to align a clamped work piece with a rotary table (C-axis) rotating about the Y-axis is to measure two points on the X-axis:
   - -the angle of rotation is relative to the X-axis.
   - -the tool rotates about the X-axis.
   - -the tool axis with the probe is the Z-axis.



2. Zum Ausrichten eines aufgespannten Werkstückes auf einen um die Z-Achse drehenden Rund
   - -der Rotationswinkel ist bezogen auf die X-Achse.
   - -das Werkstück dreht sich um die X-Achse.

2. All that is required to align a clamped work piece with a rotary table (C-axis) rotating about the Z-axis is to measure two points on the X-axis:
   - -the angle of rotation is relative to the X-axis.
   - -the tool rotates about the X-axis.
   - -the tool axis with the probe is the Y-axis.



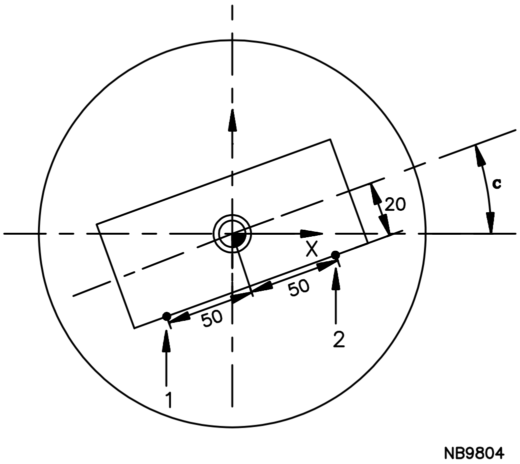
Examples

N.. G50 X1 I0.8 N=54

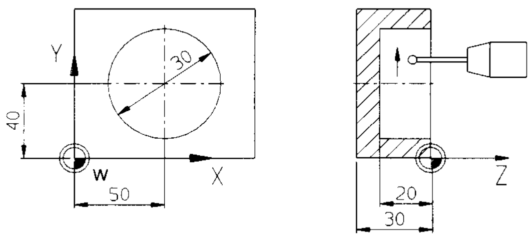
Change the X coordinates of the G54 offset by multiplying the correction value by 0.8 and enter the new X coordinate value of G54 in the zero point memory.

N.. G50 T5 L1=1 K0.97 R1=1

Correct the length of tool 5 by multiplying the difference in Z (tool in Z axis) by 0.97, and enter the new dimension in the tool memory.



N50003
N1 G17 T1 M6
N2 G54
N4 G45 X-50 Z0 Y-20 C0 J1 N=1 Measurement at point 1
N5 G45 X50 Z0 Y-20 J1 N=2 Measurement at point 2
N6 G50 C1 N=54 Settlement of zero offset
N7 G54 Reactivate zero offset
N8 G0 Z100 B0



N50006
N1 G54
N2 G17 T1 M67 (Mill R5)
N3 G89 Z-20 B2 R15 F1000 S50 M3
N4 G79 X0 Y0 Z0
N5 G0 Z50 M5
N6 T31 M67 (Measuring probe)

```
N7 M19
N8 M27                                          Activate measuring probe
N12 G46 X50 Y40 Z-5 R15 I1 J1 F500 E5           Measuring a full circle
N13 G0 Z50
N14 G49 R0.02 R1=2 N=21 E5 (bore > (15+0.02) jump-> N=21)    Tolerance comparison
N15 G49 R2 R1=.02 N=17 E5 (bore < (15-0.02) jump-> N=17)     Tolerance comparison
N16 G29 E10 E10=1 N=23                          Conditional jump to end of program
N17 G50 T1 R1=1                                 Settlement of tool radius
N18 M28                                         Turn off measuring probe
N19 G14 N1=2 N2=5
N20 G29 E1 E1=1 N=23
N21 M0
N22 (Bore outside tolerance area)
N23 M30
```

23.29 Cancel/activate G52 zero point shift G51/G52

Determines the work piece zero point with the stored values.

Activate:
N... G52

Cancel:
N... G51

Note

The use of the functions is limited to programs that were created with previous types of controllers.

Function G52 is cancelled by the Clear control softkey or by programming G51.

Functions G51 and G52 remain active after Cancel program and M30.
If a G54 .. G59 zero point offset is already active, G52 is effective from the point of this offset. If G52 ia active, G54 .. G59 are effective from the point of this offset.

FROM V320

If **MC84 = 0**, G52 is in the ZO.ZO (zero point) memory.
If **MC84 > 0**, G52 is in the PO.PO (pallet offset) memory.

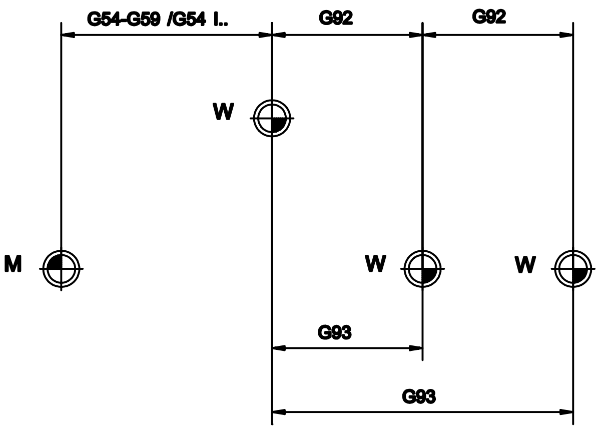
The zero points can be edited in both memories.

23.30 Cancel/activate zero point shift G53/G54...G59

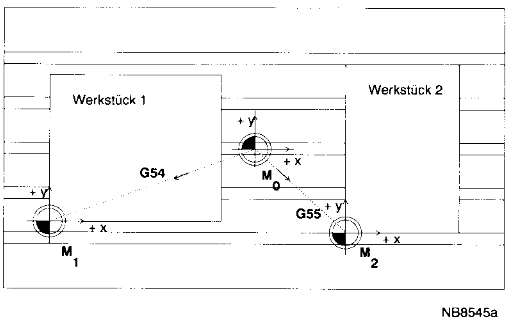
Offsets the work piece zero point to a new position, the coordinate values of which are stored in the zero point memory (under the relevant number).

Activate:
N.. G54
N.. G55
N.. G56
N.. G57
N.. G58
N.. G59

Delete:
N.. G53



Example



:

N60 G54 Activate zero offset G54
:
N600 G55 Activate zero offset G55. The coordinates relate to the new zero point.
:

23.31 Extended zero offset G54 MC84>0 (from V320 onwards)

Additionally to the current zero offset table G54..G59 there is another zero offset table G54 I[nr] with a maximum of 99 zero offsets. The relevant zero offset is selected by machine constant MC84.

- Identifier of zero offset memory Ze.Ze (MC84 > 0)
- Programming (offset values) of zero offset in NC program
- Programming an angle of rotation (B4=) in zero offset
- Entering comment in zero offset memory

Define and call zero offset:
G54 I[nr] [axis coordinates] {B4=..}

Call zero offset:
G54 I[nr]

Parameters

X,Y,Z Nullpunktkoordinate
Maschinenfunktion
Nullpunktwinkel
Nullpunktwinkel
Nullpunktindex

Notes and use

The zero offset table is adjusted during scaling (MC84 > 0). The existing zero points are maintained. Extended zero points are initialised to zero.

Attention: **If MC84 is zeroed, the table is changed (ZE.ZE to ZO.ZO). The new zero point table is initialised to zero.**

Offset values may be entered in the zero point memory in two different ways:

- The values of zero offsets G54 I[nr] are entered in the zero offset memory prior to execution of the program, via the control panel or from a data carrier.
- The values of zero offset G54 I[nr] X.. Y.. Z.. A.. B.. C.. B4=.. are programmed in an NC program block. During execution of the program, the programmed values are accepted and activated in the zero offset memory.

Attention: **If no new zero offset values have been programmed in the program block, the zero offset values already existing in the memory must not be overwritten or deleted. The unprogrammed axis coordinates are taken from the memory. Collision hazard!**

Every zero offset in the table may also involve comments.

Every zero offset in the table may also involve axis rotation. First the offset is carried out, then the coordinate system is rotated through angle B4=.

G52 does not affect the functions G53...G59. If G52 is active, G54..G59 will be active from this offset.

A programmed zero offset (G92 or G93) will be cancelled by any of the G54 I[nr] functions.

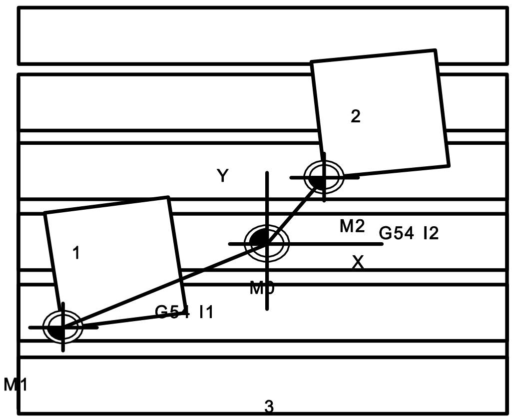
G54 I[nr] is automatically cancelled by the Clear control softkey or by programming G53. The functions G54...G59 are not cancelled by the Cancel Program softkey or M30.

Example

1.

N60 G54 I1	Selection of zero point W1. Its coordinates (X40,Y100,Z300) are taken from the zero offset memory. All programmed coordinates are measured from W1.
N600 G54 I2	Selection of zero point W2. Its coordinates (X200,Y100,Z100) are taken from the zero offset memory. Zero point W1 is cancelled and W2 activated. As a result, all programmed coordinates are measured from W2.
N700 G53	Turn off zero point W2. The coordinates (X0,Y0,Z0) are taken from the G53 zero offset memory. Zero point W2 is cancelled and M is activated. As a result, all programmed coordinates are measured from M.

2.
Axis rotation



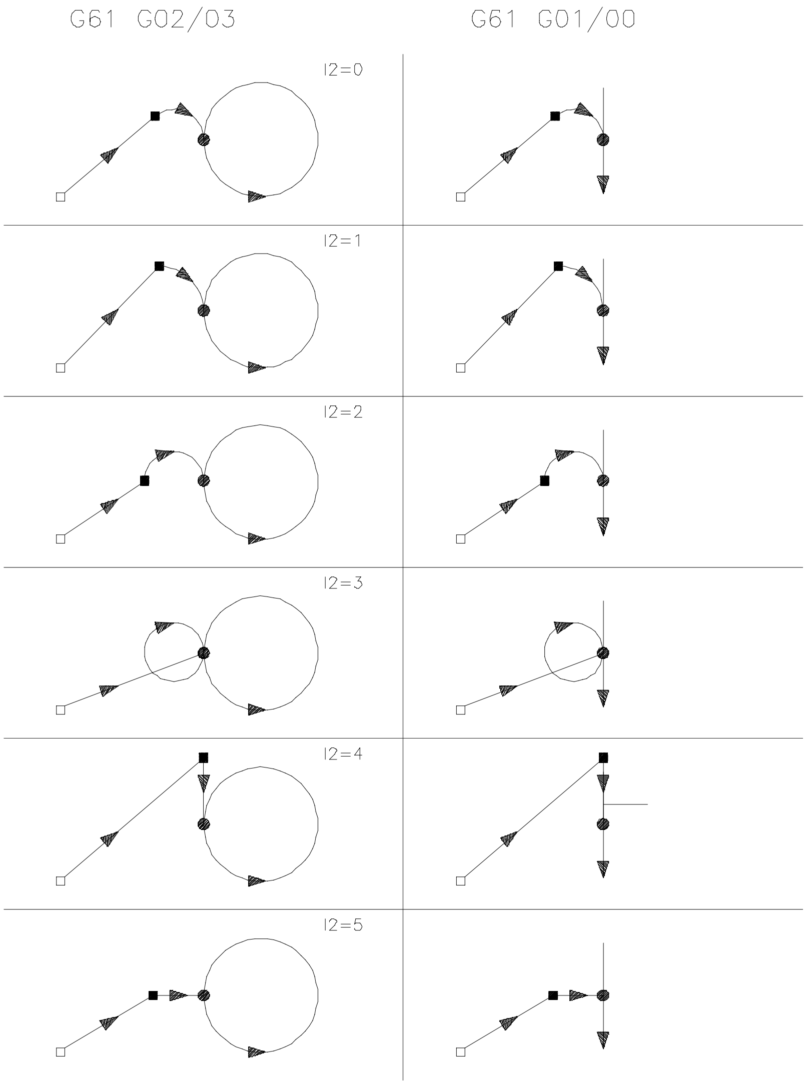
1 Workpiece 1
2 Workpiece 2
3 Machine table

Entry in the zero point table and calling:

N60 G54 I1 X-42 Y-15 B4=14 (Z0 C0)	The zero offset values are entered in the zero offset table. Machine workpiece 1. All programmed coordinates are measured from M1.
N120 G54 I2 X10 Y24 B4=-17	Machine workpiece 2. All programmed coordinates are measured from M2.

23.32 Tangential approach G61

Programs a tangential approach movement between a starting point and the start of a contour.



TANGENTIAL CONTOUR APPROACH G61

Actual position.

Calculated starting position in the plane. Tool axis Z (G17). Z1 can be programmed. If Z1 has not been programmed, Z1=Z.

Contour starting position (X, Y, Z).

N... G61 {I2=..} X... Y... Z... R... [{X1=..} {Y1=..} {Z1=}] {I1=} {F2=}
N... G61 {I2=..} B2=... L2=... Z... R... [{X1=} {Y1=}] {Z1=} {I1=} {F2=}

Parameters

X,Y,Z Konturstartpunkt
Punktedefinitionsnummer
Radius
Startpunkt in Z
Startpunkt in X
Startpunkt in Y
Polarwinkel
Polarlänge
Linearbewegung 0=Eilg.,1=Vorsch.
¼,½-Kreis,Helix,Konturp.,senkr.
Vorschub Kreisbewegung

I2=1 mit Viertelkreis.
I2=2 mit Halbkreis.
I2=3 Helix für Zustellen (für Taschen).
I2=4 Konturparallel.
I2=5 Senkrecht

Wortweise Absolut-/Inkremental-Programmierung
X90=,Y90=,Z90= Endpunkt absolut
)1=,Z91= Endpunkt inkremental

Notes

The control system itself calculates a starting position. The first movement is a positioning movement to the calculated starting point. The approach movement starts from this point.

The approach movement consists of two different movements. First, the rapid traversing or feed movement (determined by I1=) to the (calculated) starting point of the approach movement. Secondly, a feed movement along the approach contour to the starting point of the contour.

The approach side is determined by the active function G41/G42. When G40 is active, there will be an approach movement, similarly to G41.

If radius compensation (G41/G42 without travel in the program block) is activated directly before the G61 block, compensation takes place during linear movement. Depending on the actual position, the movement will be closer to or farther away from the approach circle.
If radius compensation is already active, both the linear and circular movements will be carried out with radius compensation.

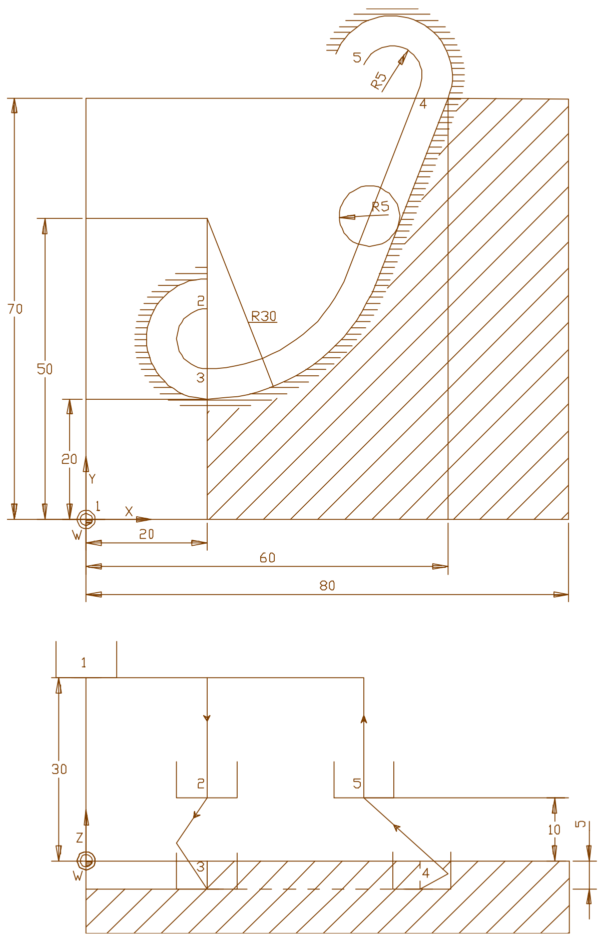
If no G function has been programmed after the G61 block, G1 is not active automatically. The last movement of the G61 function may be G1, G2 or G3.

If the distance between the actual position and the approach circle exceeds the milling radius (I2=0), the approach movement consists of a line and circular arc.
If the distance between the actual position and the approach circle is smaller than the milling radius, I2=0 is changed to I2=1, and the approach movement will be a quarter circle.

G61 programming is subject to the following limitations: G61 is not allowed in the ICP and G64 modes, in the MDI mode and in the G182 mode.

Specific limitations apply to the blocks immediately following the approach movement (G61). Only the following functions G64, G0, G1, G2 and G3 with movements in the active plane are allowed.
Falls nach dem G61-Satz keine G-Funktion programmiert worden ist, wird G1 nicht automatisch wirksam. Die letzte Bewegung der G61 Funktion kann G1, G2 oder G3 sein.

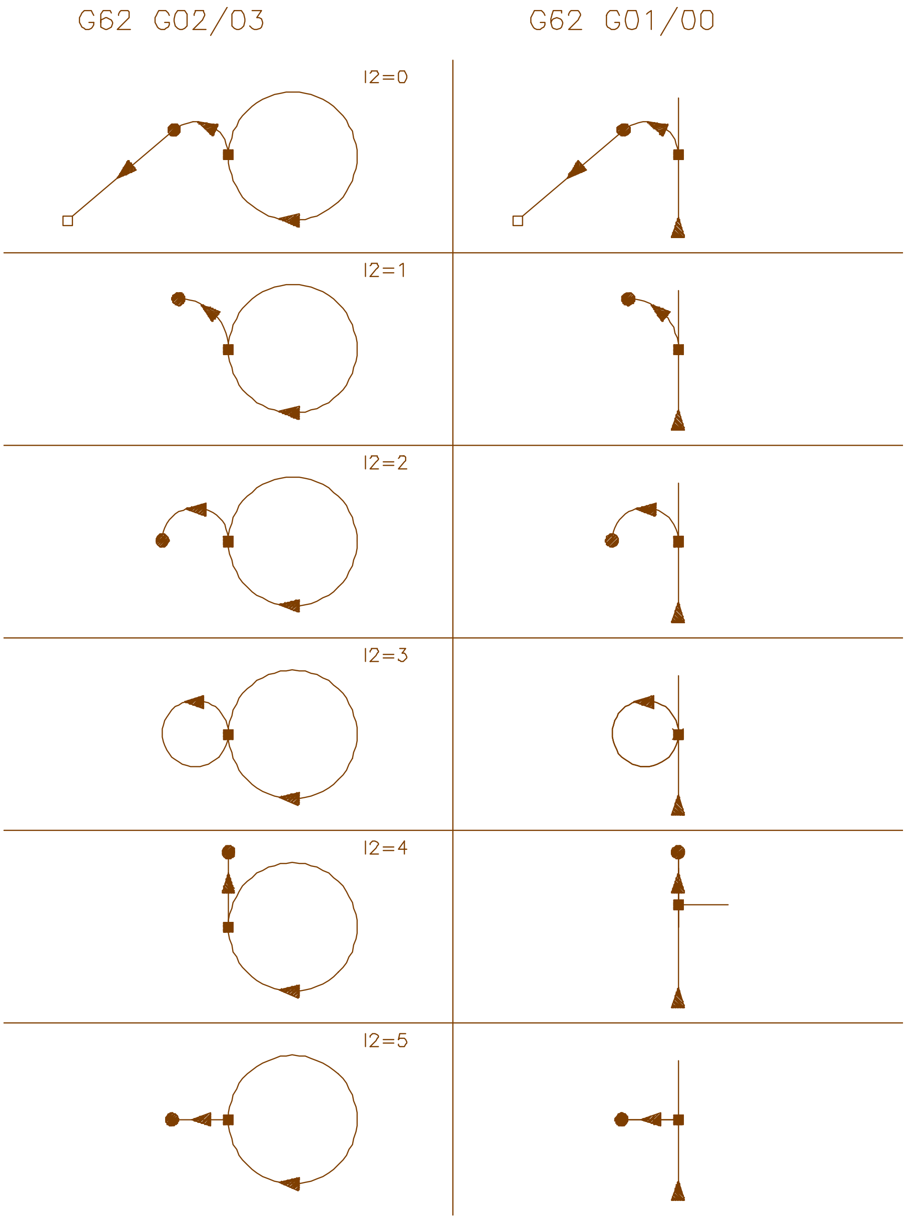
Example



N1 G17	
N2 T1 M6 (cutter R5)	
N3 F500 S1000 M3	
N4 G0 X0 Y0 Z30	Approach starting position. (position 1: X0 Y0 Z30).
N5 G41	Radius compensation to the left.
N6 G61 I2=2 X20 Y20 Z-5 Z1=10 R5 I1=0 F2=200	
	Tangential approach movement (I2=2) with semicircle. The initial part of the approach movement is a rapid traversing movement with positioning logic to the starting point of the semicircle (position 2: X.. Y.. Z10). Radius compensation is activated for this movement. The circular arc is made as a helix. The contour starts at position X20 Y20 Z0 (position 3: X20 Y25 Z-5)
N7 G64	
N8 G3 I20 J50 R1=0	
N9 G1 X60 Y60	
N10 G63	
N11 G62 I2=2 Z1=10 R5	Tangential exit (I2=2) with semicircle. The semicircle is made as a helix. Starting height of Z axis -5. The end height is 10. (position 5: X.. Y.. Z10).
N12 G40	
N13 G0 X0 Y0 Z30	
N14 M30	

23.33 Tangential exit G62

Programs a tangential exit after the end of the contour.



TANGENTIAL EXIT FROM THE CONTOUR G62:

End position of the contour.

□ Calculated end position the plane. Tool axis Z (G17). Z1 can be programmed.

□ The height remains unchanged when Z1 has not been programmed.

Programmed end position of the exit movement (X, Y, Z) (only I2=0).

□

N... G62 I2>0 Z1=... R... {I1=} {F2=}
N... G62 I2=0 X... Y... Z... Z1=... R... {I1=} {F2=}
N... G62 I2=0 B2=... L2=... Z... R... {I1=} {F2=}

Parameters

X,Y,Z Endpunkt tangentiales Wegfahren
Punktedefinitionsnummer
Radius
Startpunkt in X
Startpunkt in Z
Startpunkt in Y
Polarwinkel
Polarlänge
Linearbewegung 0=Eilg.,1=Vorsch.
¼-,½-Kreis,Helix,Konturp.,senkr.
Vorschub Kreisbewegung

I2=0 mit Endpunkt und Kreisbogen
I2=1 mit Viertelkreis
I2=2 mit Halbkreis
I2=3 mit Helix für Zustellen
I2=4 Konturparallel
I2=5 Senkrecht

Wortweise Absolut-/Inkremental-Programmierung
X90=,Y90=,Z90= Endpunkt absolut
)1=,Z91= Endpunkt inkremental

Notes

First read G61 to understand G62.

Notes and usage

If radius compensation (G40 without travel in the program block) is turned off immediately before the G62 block, compensation will be deactivated during the tangential exit movement. If radius compensation with G40 is not deactivated, both the circular and the linear movement will be with radius compensation.

Limitations

Programming of G62 is subject to the following limitations:
- G62 is not allowed in the ICP and G64 modes
- G62 is not allowed in the MDI mode
- G62 is not allowed in the G182 mode

Specific limitations apply to the blocks immediately following the approach movement (G61). Only the following functions are allowed:
- G64
- G0, G1, G2, G3 with movements in the active plane

Example

Refer to example of G61.

23.34 Cancel / activate geometric calculations G63/G64

G63: Cancels the geometry calculation

G64: Activates the geometry calculation

Parameters: G64 active

Note

Programs that require geometry calculation can be easily created by the user with Interactive Contour Programming (ICP).
(see chapter on Interactive Contour Programming)

23.35 INCH / METRIC programming G70/G71

Loads and calls part programs that are described in the alternative unit of measurement to that specified in the CNC (unit of measurement defined in the machine constants)

Inch programming:
N... (PROGRAM NAME) G70

Metric programming:
N... (PROGRAM NAME) G71

Examples

1st unit of measurement: CNC: Metric Program: Inch

```
9001.PM
N9001  G70
:
N50 G1 X2 Y1.5 F8           Read-in ensures storage of X50.8 Y38.1 and F203.2.
:
```

2nd unit of measurement: CNC: Inch Program: Metric

```
9001.PM
N9002 G71
:
N50 G1 X50.8 Z38.1 F203.2   Read-in ensures storage of X2 Y1.5 and F8.
:
```

23.36 Cancel / activate mirror image and scaling G72/G73

Enable zoom/reduce:
N.. G73 A4=..   (factor or percentage, setting in machine constants)

Disable zoom/reduce:
N.. G73 A4=1   (factor)
N.. G73 A4=100 (percentage)

Mirroring about an axis or sign change per axis:
N.. G73 {X-1} {Y-1} {Z-1} {A-1} {B-1} {C-1}

Disable mirroring/sign change per axis:
N.. G73 {X1} {Y1} {Z1} {A1} {B1} {C1}

Disable zoom/reduce and mirroring:
N.. G72

3684 vergrößern verkleinern

G73 A4=2

3684 vergrößern verkleinern

G73 A4=0.5

3685 spiegeln

Parameters

G72 No parameters

G73 Zoom / reduce
Mirroring / sign change
A4= Scaling factor
A4= Maßfaktor

Example

```
N7273  (MIRROR AN ISLAND)
N1 G17
N2 G54
N3 T1 M6 S2000 F200          Change tool
N4 G0 X-60 Y20 Z0 M3
N5 G1  Z-9
N6 G43 Y0
N7 G41 X-10
N8 G3 X0 Y10 R10
N9 G1 X0 Y45
N10 G1 X45 Y45
N11 G1 X45 Y-10
N12 G40
N13 G1 Z10
N14 G73 X-1 Y-1              Mirror coordinates around X and Y axes
N15 G14 N1=4 N2=13           Repeat the blocks 4-13
N16 G72                      Cancel mirroring
N17 S1000 F100 T6 M6         Change tool 6
N18 G81 Y5 Z-20
N19 G79 X30 Y14
N20 G79 X10 Y32
N21 G79 X20 Y32
N22 G79 X30 Y32
N23 G79 X40 Y32
N24 G73 X-1 Y-1              Mirror coordinates around X and Y axes
N25 G14 N1=19 N2=23          Repeat the blocks 19-23
N26 G72                      Cancel mirroring
N27 G0 Z50 M30
```

23.37 Absolute position G74

Rapid movement to a position whose coordinates refer to the reference point

N... G74 X.. Y.. Z.. {X1=..} {Y1=..} {Z1=..} {K...} {L...} {K2=...)

Parameter

X,Y,Z Endpunktkoordinate
Endwinkel
0=Genauhalt,1=verschleifen,2=MC
Inpoc Fenster (0: MC, 1-32767 µm)
0=mit WZ-Länge, 1=ohne WZ-Länge

Notes and usage

The G74 function is mainly used in programming cycles for tool changers, pallet stations etc., if the programmed coordinates should not depend on the coordinates used to define machining of the workpiece.

The end point coordinate may be defined in two different ways:

1) X100: relative position to reference point
2) X100 X1=2: relative position to the absolute position of the machine constant

Machine positions 1 to 10 for the first axis can be determined using the machine settings MC3145 -- MC3154. The machine settings MC3245 -- MC3254 etc are used for the second axis.
If the value in the machine setting being used is zero, no drive movement is implemented.

With G74 there will be travel simultaneously in all programmed axes. The next travel does not start until the required position is reached in all axes.

K0: Allowance is made for an (accurate) stop between the movement of block G74 and the movement in the next block, as is usual for rapid traversing movements.
(K0 is the switch-on position).

K1: No allowance is made for a stop between the movement of block G74 and the movement in the next block (rounding). The next movement starts when the required position has almost been reached in all axes.

K2: No allowance is made for a stop between the movement of block G74 and the movement in the next block. The next movement starts when the required position has almost been reached in all axes. This position is defined by the machine constant (MC136) (K2=0) or the window size (K2=...) for the corner release distance.
K2= window size in mm (0-32.766 mm)

If an incremental movement is programmed after a G74 movement, the coordinates relate to the position indicated in the G74 block.

Tool length compensation is usually not applied in G74 (L0 is switch-on position). L1 must be programmed for tool length compensation.

Radius compensation (G41...G44) should be cancelled before activation of the G74 function.

Geometry function G64 must not be active during G74.

The active zero offset is ignored for the G74 block.

The travel immediately preceding G74 should be programmed with G0 or G1. The travel immediately following G74 is automatically executed with the same G function.

Example

The coordinates of P in relation to R are known. P is programmed as follows:

```
:
N10  G0  X95  Y10
N11  G74  X-120  Y-115 Movement from X95 Y10 to P
```

:

Example of block:
N20 G74 X100 X1=1 Y123.456 Z1=10 K2 K2=25.2

X100 X1=1	Relative position to the absolute position of the machine constant.
Y123.456	Relative position to the reference point.
Z1=10 (Z0)	Position related to the absolute position of the machine constant.
K2	No allowance is made for a stop between the movement of block G74 and the movement in the next block. The next movement starts when the required position has almost been reached in all axes. This position is defined by the window size (K2=...) for the corner release distance.
K2=	Window size in mm

23.38 Bolt hole cycle G77

Executes a previously programmed drilling or milling cycle at points spaced equally on a circular arc or full circle.

Points on a circular arc:
N.. G77 [centre point] R.. J.. I.. K.. {B1=..}

Points on a full circle:
N... G77 [centre point] R.. J.. I.. {B1=..}

Parameter

X,Y,Z Lochkreismittelpunkt
Endwinkel
Anfangswinkel zum ersten Punkt
Anzahl der Punkte
Endwinkel zum letzten Punkt
Punktnr. von Lochkreismittelpunkt
Lochkreisradius
Winkel
Polarwinkel
Streckenlänge
Polarlänge
Punktnr. von Lochkreismittelpunkt
Parameterdefinition

Wortweise Absolut-/Inkremental-Programmierung

X90=,Y90=,Z90= Mittelpunkt absolut
)0=,C90= Endwinkel absolut
)1=,Z91= Mittelpunkt inkremental
)1=,C91= Endwinkel inkremental

Hinweis

B1= hat zwei Bedeutungen:
Es stellt den Winkel für das Drehen einer Tasche bzw. Nute dar, oder die Lage des Kreismittelpunktes (B1= mit L1=, oder X/Y mit B1=).

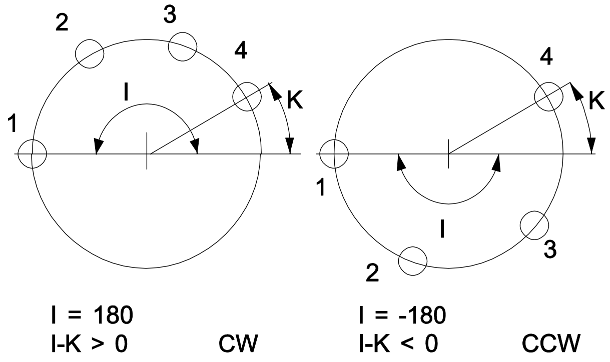
Examples

```
:
N40  G78  P2  X..  Y..  Z..                Second defined point
:
N50  G81  Y1  Z-10  F100  S1000  M3        Define cycle
:
N60  G77  P2  R25  I30  K150  J4           Repeat cycle four times on circular arc
:

:
N41  G78  P1  X..  Y..  Z..                First defined point
:
N50  G81  Y1  Z-10  F100  S1000  M3        Define cycle
:
N60  G77  P1  R25  I0  J6                  Repeat cycle six times on full cycle
:
```

Turned grooves.

N60 T1 M6	Change tool 1 (cutter with radius of 4.8 mm)
N65 G88 X20 Y10 Z-10 B1 F100 S1000 M3	Define the groove as if the sides were parallel to the X and Y axes.
N70 G77 X78 Y56 Z0 R24 I0 J6 B1=30	The turned grooves are milled.

Direction of the bores on a circular arc



N50 G81 Y1 Z-10 F100 S1000 M3
N60 G77 X0 Y0 Z0 R25 **I180 K30** J4
N70 G77 X0 Y0 Z0 R25 **I-180 K30** J4

Explanation:

N50 : Define cycle
N60 : Repeat cycle four times on the circular arc; start at 180 degrees, end at 30 degrees in clockwise direction (CW).
N70 : Repeat cycle four times on the circular arc; start at -180 degrees, end at 30 degrees in counter-clockwise direction (CCW).

23.39 Point definition G78

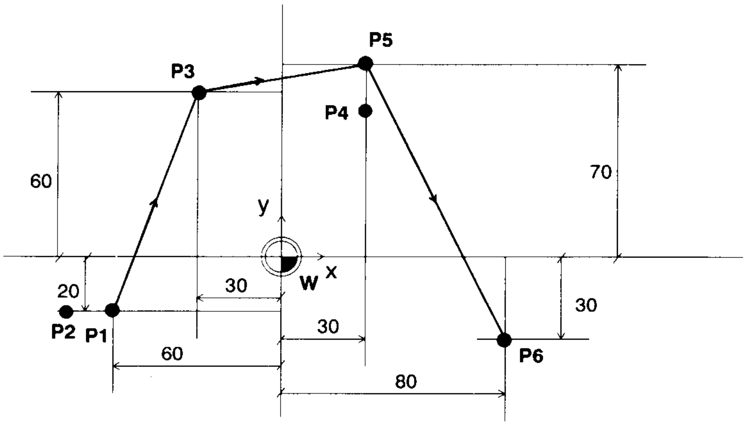
Defines the coordinates of a point once in a program. For subsequent travel to this point, only its number needs to be programmed.

N... G78 P... [Point coordinates]

Parameters

X,Y,Z Punktkoordinate
Punktwinkel
Polarwinkel
Polarlänge
Punktedefinitionsnummer

Example



NB8548

N10 G78 X-60 Y-20 P1	Define point 1
N11 G78 X-70 Y-20 P2	
N12 G78 X-30 Y60 P3	
N13 G78 X30 Y55 P4	
N14 G78 X30 Y70 P5	
N15 G78 X80 Y-30 P6	
:	
N90 G0 P1=1	Move tool in rapid traversing to the position defined by P1.
:	
N91 G1 P1=3 P2=5 P3=6 F1000	Move tool with programmed feed to P3, P5
:	and then to P6.

Note

Only one point can be defined in each G78 block. All the point coordinates are in relation to the active work piece zero point W.

Program blocks with G1 or G79 can contain up to 4 points. In all other cases, there can only be one point in a program block.

Example: N.. G1 P1=9 P2=1 P3=3 P4=8

P adresses with index:
The index value (1-4) determines the priority for the machining sequence (1=highest priority, 4=lowest priority). The entry after the equals sign gives the number of points in the points memory.
Another possibility is to enter the point definitions as parameters, the index again denoting the priority.
P-Adresse mit Index:

23.40 Activate cycle G79

Executes previously programmed drilling cycles (G81, G83-G86) or milling cycles (G87-G89) at defined positions.

N... G79 [point coordinates] {B1=..}

Parameters

X,Y,Z Punktkoordinate
Punktwinkel
Winkel
Streckenlänge
Polarwinkel
Polarlänge
Punktedefinitionsnummer (P1-P4)
=,P3=,P4= Punktedefinitionsnummer

Wortweise Absolut-/Inkremental-Programmierung
X90=,Y90=,Z90=,A90=,B90=,C90= Punkt absolut
91=,Z91= Punkt inkremental
91=,C91= Punktwinkel inkremental

Example

Three holes are to be drilled

```
:
N50  G78  P1  X50  Y20  Z0              Define point
N55  G78  P2  X50  Y80  Z0
N60  T1  M6
N65  G81  Y1  Z-30  F100  S1000  M3     Define drilling cycle
N70  G79  P1  P2                        Drill holes at positions 1 and 2
N75  T2  M6
N80  G79  X50  Y50  Z0  M3              Drill hole
:
```

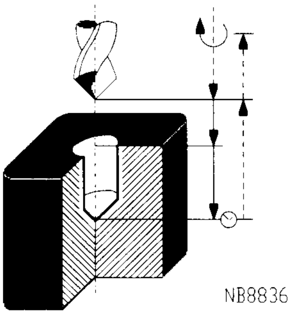
Note

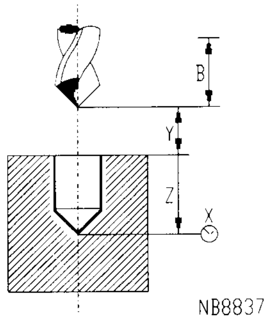
B1= has two meanings:
It represents the angle for machining a pocket or slot, or the position of the circle centre point (B1= with L1=, or X/Y with B1=).

See example G77 "Turned grooves"

23.41 Drilling cycle G81

N.. G81 Z.. {X..} {Y..} {B..}





Eilgangbewegung
Vorschubbewegung

Parameter

Y Sicherheitsabstand
Bohrtiefe
Verweilzeit (s)
Erhöhter Rückzugsabstand

Example

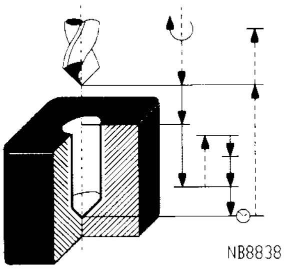
```
:
N50 G78 P1 X50 Y20 Z0               Define point 1
N55 G78 P2 X50 Y80 Z0               Define point 2
N60 G0 Z10 T1 M6
N65 G81 X1.5 Y1 Z-30 F100 S500 M3   Define cycle
N70 G79 P1 P2                       Execute cycle at positions 1 and 2
:
```

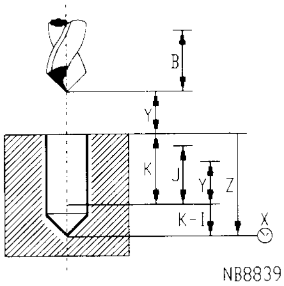
Note

A machining cycle (G81-G89) is executed with G77 or G79.

23.42 Deep hole drilling cycle G83

N.. G83 Z.. {X..} {Y..} {B..} {I..} {J..} {K..} {K1=..}





Parameter

Y Sicherheitsabstand
Gesamte Bohrtiefe
Verweilzeit (s)
Erhöhter Rückzugsabstand
Reduzierwert der Bohrtiefe
Rückzugsabstand nach Bohrschritt
Bohrtiefe für ersten Bohrschritt
Anzahl Zustellungen bis Ausspanen

Examples

1.
:
N5 T1 M6
N10 G83 Y4 Z-150 I2 J6 K20 F200 S500 M3 — Define cycle
N20 G79 X50 Y50 Z0 — Execute cycle
:

2.

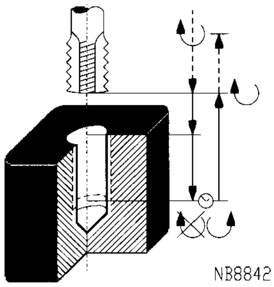
```
:
N..  G83  Y4  Z-150  I2  J6  K20  K1=3        Define cycle
N20  G79  X50  Y50  Z0                        Execute cycle
:
```

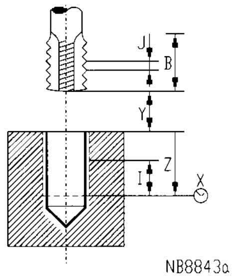
Note

A machining cycle (G81-G89) is executed with G77 or G79.

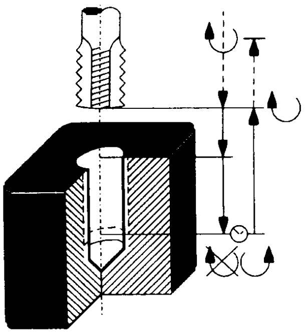
23.43 Tapping cycle G84

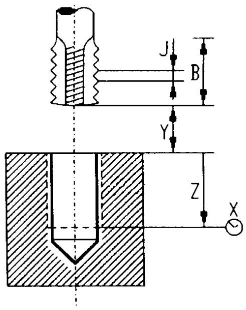
N... G84 Z... {Y...} {B...} {J...} {X...}
or
N... G84 **I1=0** Z... {Y...} {B...} {J...} {X...}





from V400:
Tapping can also be implemented in a closed control loop, as an interpolation between the tool axis and the spindle. The accelerating power of the spindle is taken up in this interpolation. In this way, the running of the spindle in the desired position and at the correct speed in rpm is guaranteed ("synchronous tapping").





N... G84 **I1=1** Z... {Y...} {B...} {J...} {X...}

Parameter

Y Sicherheitsabstand
Gewindetiefe
Verweilzeit (s)
Erhöhter Rückzugsabstand
Gewindesteigung

F(feed) = J(pitch) * S(speed)

Example

```
N14  T3  M6
N15  G84  Y9  Z-22  J2.5  S56  M3  F140          Define cycle
N20  G79  X50  Y50  Z0                           Execute cycle
```

Note

A machining cycle (G81-G89) is executed with G77 or G79.

During call-up of a G84 cycle by means of G79 the soft key Clear control must be set for G94 operation (feed in mm/min) and not for G95 operation (feed in mm/360 degree turn). G94 is always to be programmed before G84.

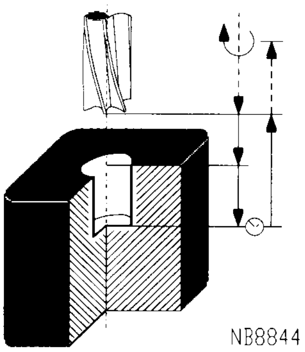
From V400:
Tapping can be programmed with or without interpolation.
I1=0 guided (basic position, open position control loop)
I1=1 interpolating (closed position control loop)
An active "Process level G7 traverse" can only be operated with interpolation (I1=1)

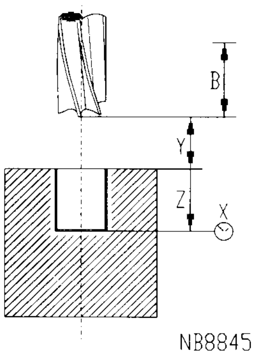
As of V410,
if "Swivel working plane (G7)" is active, and the head has not been swivelled (tool axis is same as the Z axis), guided tapping can also be performed (I1=0).

Machine settings
MC723 and MC727 are no longer required during interpolation. The machine settings of the spindle should be set correctly during tapping. The spindle acceleration is calculated for every operation with the help of MC2491, 2521, 2551, 2581 and MC2495, 2525, 2555, 2585. In every case, MC4430 should be active for good control.

23.44 Reaming cycle G85

N.. G85 Z.. {X..} {Y..} {B..} {F2=..}





Parameter

Y Sicherheitsabstand
Reibtiefe
Verweilzeit (s)
Erhöhter Rückzugsabstand
Rückzugsvorschub

Example

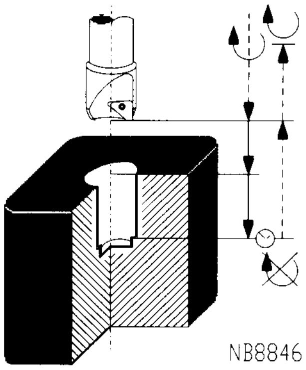
```
:
N25  T4  M6
N30  G85  X2  Y3  Z-30  F50  S100  F2=200  M3  Define cycle
N35  G79  X50  Y50  Z0                         Execute cycle
:
```

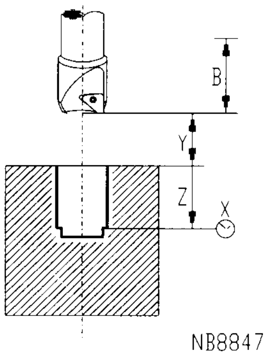
Note

A machining cycle (G81-G89) is executed with G77 or G79.

23.45 Boring cycle G86

N.. G86 Z.. {X..} {Y..} {B..}





Parameter

Y Sicherheitsabstand
Ausdrehtiefe
Verweilzeit (s)
Erhöhter Rückzugsabstand

Example

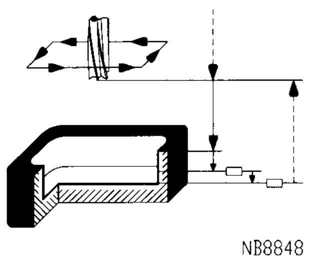
N45 T5 M6
N50 G86 X1 Y9 Z-27 B10 F20 S500 M3 Define cycle
N55 G79 X50 Y50 Z0 Execute cycle

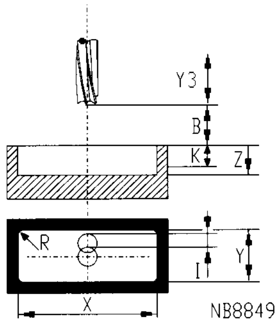
Note

A machining cycle (G81-G89) is executed with G77 or G79.

23.46 Rectangular pocket milling cycle G87

N.. G87 X.. Y.. Z.. {R..} {B..} {I..} {J..} {K..} {Y3=..} {F2=..}





Parameter

Taschengeometrie
X Abmessung parallel zu X
Abmessung parallel zu Y
Gesamte Taschentiefe
Eckenradius
Eintauchvorschub (Satzweise)

Bearbeitungsparameter
B Sicherheitsabstand
Schnittbreite des Fräsers in %
J1:Gleichlauf / J-1:Gegenlauf
Zustelltiefe
Erhöhter Rückzugsabstand

Example

N10 T1 M6	
N20 G87 X200 Y100 Z-6 J+1 B1 R40 I75 K1.5 F200 S500 M3	Define cycle
N30 G79 X120 Y70 Z0	Execute cycle

Note

A machining cycle (G81-G89) is executed with G77 or G79.

23.47 Groove milling cycle G88

N.. G88 X.. Y.. Z.. {B..} {J..} {K..} {Y3=..} {F2=..}

Parameter

Nutengeometrie
X Abmessung parallel zu X
Abmessung parallel zu Y
Gesamte Nutentiefe
Eintauchvorschub (Satzweise)

Bearbeitungsparameter
B Sicherheitsabstand
J1:Gleichlauf / J-1:Gegenlauf
Zustelltiefe
Erhöhter Rückzugsabstand

Example

N10 S500 T1 M6
N20 G88 X55 Y15 Z-5 B1 K1 F350 Y3=10 F2=200 M3 Define cycle

```
N30  G79  X22.5  Y22.5  Z0                              Execute cycle
N40  G88  X15  Y-55  Z-5  B1  K1  Y3=10  F2=200
N50  G79  X90  Y62.528  Z0
```

Notes

A machining cycle (G81-G89) is executed with G77 or G79.
The signs of X and Y determine the direction of the slot from the start point S.

23.48 Circular pocket milling cycle G89

N.. G89 Z.. R.. {B..} {I..} {J..} {K..} {Y3=..} {F2=..}

Parameter

Taschengeometrie
Z Gesamte Taschentiefe
Radius der Kreistasche
Eintauchvorschub (Satzweise)

Bearbeitungsparameter
B Sicherheitsabstand
Schnittbreite des Fräsers in %
J1:Gleichlauf / J-1:Gegenlauf
Zustelltiefe
Erhöhter Rückzugsabstand

Example

```
N10  T1  M6
N20  G89  Z-15  B1  R25  I75  K6  F200  S500  M3        Define cycle
N30  G79  X50  Y50  Z0                                  Execute cycle
N40  G0  Z200
```

Note

A machining cycle (G81-G89) is executed with G77 or G79.

23.49 Absolute/incremental programming G90/G91

G90: Absolute coordinates, measured from the program zero point W.
G91: Incremental coordinates, relative to the last position.

N.. G90/G91

Parameters

Achsenkoordinaten bei G90

X,Y,Z Endpunktkoordinate
Endwinkel

Achsenkoordinaten bei G91

X,Y,Z Endpunktkoordinate
Endwinkel

Example

N88550	
N1 G17	
N2 G54	
N3 G98 X0 Y0 Z60 I100 J100 K-80	Graphic window definition
N4 S1300 T1 M6	
N5 G81 Y2 Z-10 F200 M3	Define cycle
N6 G79 X50 Y50 Z0	Execute cycle
N7 G91	Change to incremental size programming
N8 G79 Y20	Execute cycle
N9 G79 X20	
N10 G79 Y-20	

N11 G90 Change to absolute size programming

Note

An absolute position must be programmed before the entry of the incremental dimension G91.

23.50 Wordwise absolute/incremental programming (from V320 onwards)

Wordwise absolute/incremental programming independently of G90/G91.

absolute programming:
N.. G.. [axis name]90=...

incremental programming:
N.. G.. [axis name]91=...

Parameter

Achsname: X, Y, Z, U, V, W, I, J, K, A, B, C
X90=,Y90=,Z90= Endpunkt absolut
0=,C90= Endwinkel absolut
1=,Z91= Endpunkt inkremental
1=,C91= Endwinkel inkremental

Hinweise und Verwendung

Kartesische Koordinaten:
Die wortweise Absolut-/Inkremental-Programmierung ist unabhängig vom modal gültigen Maßsystem G90/G91.

Polarkoordinaten:
Die Programmierung in Polarkoordinaten wird nicht beeinflußt.

Example

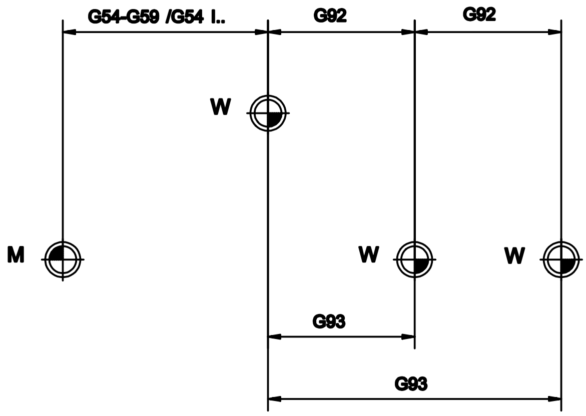
N88550	
N1 G17	
N2 G54	
N3 G195 X0 Y0 Z60 I100 J100 K-80	Define graphic window
N4 S1300 T1 M6 (drill bit R5)	Change tool 1
N5 G81 Y2 Z-10 F200 M3	Define drilling cycle
N6 G79 X50 Y50 Z0	Cycle call for 1st drilling
N7 G79 Y91=20	Cycle call for 2nd drilling, incremental movement
N8 G79 X91=20	Cycle call for 3rd drilling, incremental movement
N9 G79 Y91=-20	Cycle call for 4th drilling, incremental movement
N10 M30	

23.51 Zero point shift incremental / rotation or absolute rotation G92/G93

Zero point offset:
N.. G92 [incremental coordinate(s), in relation to the last program zero point]
N.. G93 [absolute coordinates, in relation to the zero point that was defined by G54-G59]

Rotation of the coordinates system:
N... G92/G93 B4=..

Zero point offset:



Rotation of the coordinate system:

FSP: Driving up from the traverse position by the shortest route

FSP now always outputs an angle between -180 and +180 degrees. This is changed so that an angle is output between the end switches. This angle is then the shortest route. The disadvantage is that the position of the circular axis can climb to very high values which should be reset to a moment.

The disadvantage of very high positions is resolved with a separate function with which the (internal) position is reset to a value between 0 and 360 degrees.

G93 {X},{Y},{Z},{A},{B},{C},{B2=},{L2=},{P},{P1=},{B4=},{A3=1},{B3=1},{C3=1}

in which:
A3=1, B3=1, C3=1
The corresponding axial position is reset to a value between 0 and 360 degrees.

Parameter bei G92

1. Nullpunktverschiebung
X,Y,Z Nullpunktkoordinate
Nullpunktwinkel
Winkel
Streckenlänge

2. Achsendrehung
B4= Inkrementaler Rotationswinkel

Parameter bei G93

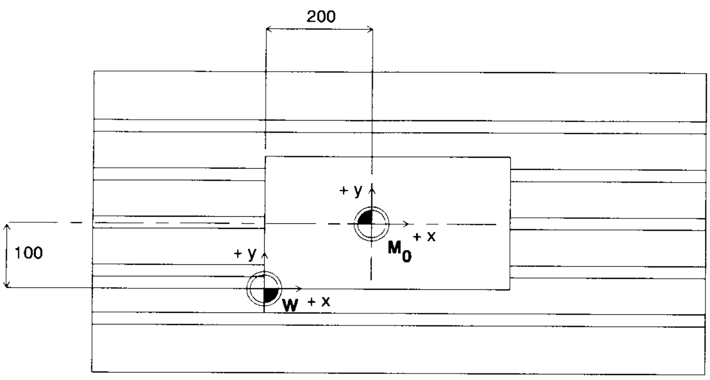
1. Nullpunktverschiebung
X,Y,Z Nullpunktkoordinate
Nullpunktwinkel
Polarwinkel
Polarlänge
Punktedefinitionsnummer

2. Achsendrehung
B4= Absoluter Rotationswinkel

3. Rücksetz Funktion **(ab V400)**
A3=,B3=,C3= Rücksetz Parameter
Mit G93 A3=1 wird die entsprechende Rundachseposition auf einen Wert zwischen 0 und 360 Grad zurück gesetzt.
Beispiel: Eine A-Achse mit der Position 370 Grad, wird nach der Programmierung von G93 A3=1 auf 10 Grad geändert.

Examples

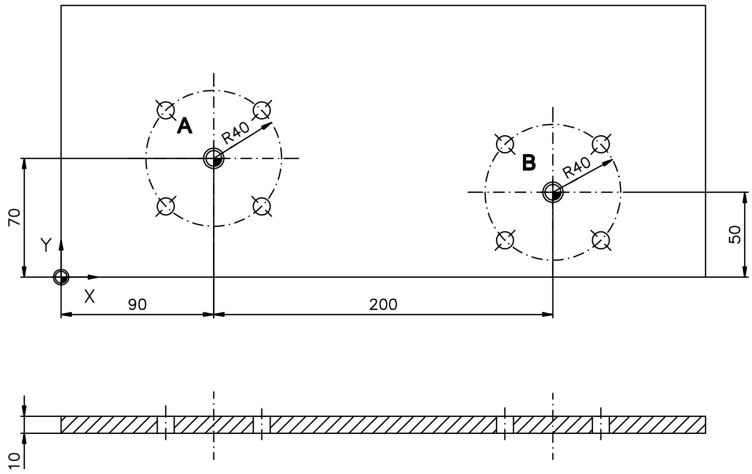
1. The centre point of the work piece coincides with the machine centre point (M). The program zero point (W) is placed in the left corner of the work piece.



NB8556

N30 G93 X-200 Y-100

2. The four holes around points A and B are to be drilled. In the program, the program zero point (W) lies in A and B.



Program with G92
N79560
N1 G17
N2 G54
N3 G98 X-10 Y-10 Z10 I420 J180 K-30
N4 G99 X0 Y0 Z0 I420 J160 K-10
N5 F200 S3000 T1 M6
N6 G92 X90 Y70 Incremental zero offset
N7 G81 Y1 Z-12 M3 Define cycle
N8 G77 X0 Y0 Z0 I45 J4 R40 Call cycle
N9 G92 X200 Y-20 Incremental zero offset
N10 G14 N1=8 Repeat function
N11 G93 X0 Y0 Cancel incremental zero offset
N12 G0 Z100 M30

Program with G93
The program appears as follows in relation to the clamping zero point:
N79561
N1 G17
N2 G54
N3 G98 X-10 Y-10 Z10 I420 J180 K-30
N4 G99 X0 Y0 Z0 I420 J160 K-10
N5 F200 S3000 T1 M6
N6 G93 X90 Y70 Absolute zero offset
N7 G81 Y1 Z-12 M3
N8 G77 X0 Y0 Z0 I45 J4 R40
N9 G93 X290 Y50 Absolute zero offset
N10 G14 N1=8
N11 G93 X0 Y0 Cancel absolute zero offset
N12 G0 Z100 M30

Notes

If no G54-G59 has previously been activated, G92/G93 is effective from the machine zero point.

If rotation of the coordinate system (G92/G93 B4=..) is active, a zero point offset programmed with G92/G93 is no longer allowed.

23.52 Feed in mm/min (inch/min) / mm/rev (inch/rev) G94/G95

Information to the controller about how the programmed feed is to be evaluated.

N.. G94/G95 F..

G94 : Feed in mm/min or inches/min.
G95 : Feed in mm/rev or inches/rev.
N... G94 F5=.

G94 :	Feed in mm/min or inch/min
G95 :	Feed in mm/rev or inch/rev
G94 F5= :	Feed of rotary axes **(V410 and higher)**
	F5=0 degrees/min (default)
	F5=1 mm/min or inches/min

Parameters

F Vorschub
Spindeldrehzahl (U/min)
Werkzeugnummer
Maschinenfunktion

Notes:

MACHINES WITH KINEMATIC MODEL
The G94 F5= function is only present if a kinematic model has been defined for the machine (MC312 must be active).

CALCULATION OF RADIUS OF ROTARY AXIS G94 F5=1
In machines with a kinematic model, the radius of the rotary axis between the centre point of the rotary axis and the workpiece can be calculated. This means that A40=, B40= or C40= no longer need to be programmed.

CANCEL G94 F5=1
G94 F5=1 is cancelled by: G94 F5=0, G95, programming with A40=, B40= or C40= in G0 or G1, M30, <Program abort> or <Reset CNC>.

Examples

```
:
N.. G94                 Feed in mm/min
N.. G1 X.. Y.. F200     Move to X.. Y.. at a feed of 200 mm/min
:

:
N.. G95                 Feed in mm/rev.
N.. G1 X.. Y.. F.5      Move to X.. Y.. at a feed of 0.5 mm/rev.
:
```

23.53 Graphic window definition G98

Defines the position relative to the program zero point W and the dimensions of a 3D graphics window in which the machining of the work piece is to be simulated graphically.

N.. G98 X.. Y.. Z.. I.. J.. K.. {B..} {B1=..} {B2=..}

Parameters

X,Y,Z Anfangskoordinate
Abmessung parallel zu X
Abmessung parallel zu Y
Abmessung parallel zu Z
Rotation Rundachse hor. (3D)
Rotation Rundachse vert. (3D)
Rotation dritte Rundachse (3D)

Example

N9000	
N1 G98 X-20 Y-20 Z-75 I140 J90 K95	Starting point and dimensions of the 3D graphic window
N2 G99 X0 Y0 Z0 I100 J50 K-55	Define blank as 3D area
:	

23.54 Graphic: material definition G99

Defines a three-dimensional blank and its position in relation to the program zero point W. The dimensions are needed for the graphical simulation.

N... G99 X... Y... Z... I... J... K...

Parameters

X,Y,Z Anfangskoordinate
Abmessung parallel zu X
Abmessung parallel zu Y
Abmessung parallel zu Z

Example

```
N9000
N1 G98 X-20 Y-20 Z-75 I140 J90 K95      Starting point and dimensions of the 3D graphic window
N2 G99 X0 Y0 Z0 I100 J50 K-55 Define blank as 3D area
:
```

23.55 3D tool correction G141

Permits the correction of the tool dimensions for a 3D tool path that is programmed in these points by its end point coordinates and standardised vectors perpendicular to the surface.

Activate 3D tool correction:
N.. G141 {R..} {R1=..}

Program straight line movements:
N.. G0/G1 [end point coordinates X.. Y.. Z..] [I.. J.. K..]

Deactivate 3D tool correction:
N.. G40
or M30, Cancel program softkey, Clear control softkey

3221m

Parameters

Im Satz G141:
R Nominaler Werkzeugradius
Nominaler Werkzeugeckenradius

Die Werte von R.. und R1=.. sollten den nominalen Werkzeugmaßen entsprechen, wie sie vom Programmiersystem zur Berechnung der Werkzeugbahn herangezogen werden. Wenn diese Werte nicht programmiert sind, werden sie automatisch Null.

Im Satz G0/G1:
X,Y,Z, Endpunktkoordinaten
I,J,K Achsenkomponenten des normalisierten Vektors (X,Y,Z)

Es können nur absolute oder inkrementale kartesische Maßangaben verwendet werden.

Example

:
N19

N20 G141 R.. R1=.. F..
N21 G1 X.. Y.. Z.. I.. J.. K.. (first scale factor)
:
N300 G141 R.. R1=.. F..
N301 G1 X.. Y.. Z.. I.. J.. K.. (second scale factorr)
:
N2400 G141 R.. R1=.. F..
N2401 G1 X.. Y.. Z.. I.. J.. K.. (third scale factor)

Notes

If G141 is active, the following functions, for example, can no longer be programmed:
G2/G3, G64, G73, G182.

The following are not permitted:
Point definition, E parameter, polar coordinates, programming of rotation axes.

Rotation of the coordinates system (G93 B4=..) is not permitted.

G73 can only be programmed before G141 is activated.

A standardised vector must be calculated for each end point.

The rounding radius in G141 is programmed witht R1=. The rounding radius is stored in the tool memory by the C wordt.

Undercuts or collisions at the 3D surface cannot be recognised by the controller.

23.56 Linear measuring movement G145

Executes a freely programmable linear measurement movement to determine axis positions.

N... G145 [Meesuring point coordinates] [(axis address) 7=..] {S7=..} E.. {F2=..} {K..} {L..} {I3=..}

Parameter

Zu messender Punkt
X,Y,Z Endpunktkoordinate
Endwinkel
Winkel
Polarwinkel
Streckenlänge
Polarlänge
Punktedefinitionsnummer

Achsenadresse
X7= E-Parameter für Meßwert in X
E-Parameter für Meßwert in Y
E-Parameter für Meßwert in Z
E-Parameter für Meßwert in A
E-Parameter für Meßwert in B
E-Parameter für Meßwert in C
E-Parameter für Meßwert in S

Messung
K K0:Werkzeugkorrektur ein / K1:aus
0:Messen bei Berühr.,1:beim Lösen
Status-Überwachung (0=EIN, 1=AUS)
E-Parameter für Meßstatus
Meßvorschub

Wortweise Absolut-/Inkremental-Programmierung
X90=,Y90=,Z90= Endpunkt absolut
)0=,C90= Endwinkel absolut
)1=,Z91= Endpunkt inkremental
)1=,C91= Endwinkel inkremental

Example

A slot is to be milled and its width measured. If the slot width is too small, the milling radius must be corrected and the slot re-worked.

```
N14504  (Milling and measuring a slot)
N1 G17
N2 G54
N3 E15=20.02  (Maximal slot width)
N4 E16=19.98  (Minimum slot width)
N5 E3=(E15+16):2
N6 S1000 T1 M6  (Milling tool d=18 mm)
N7 G0 X-25 Y50 Z-10 B0 F400 M3
N8 G1 X140
N9 G43
N10 G1 Y60
N11 G41
N12 X-25
N13 Y40
N14 X140
N15 G40
N16 Y50
N17 G0 Z50 M5
N18 G149 T0 E30
N19 T30 M6  (Measuring probe)
N20 M19                    (D address optional)
N21 M27
N22 G0 X60 Y50 Z-8 B0
N23 M29
N24 G145 Y65 E10 Y7=1 F2=500
N25 GO Y50
N26 G29 E11=E10=0 E11 N=30
N27 M29
N28 G145 Y35 E10 Y7=2 F2=500
N29 G0 Y50
N30 M28
N31 G29 E11=E10=0 E11 N=41
N32 E5=E1-E2
N33 E6=(E5-E3):2
N34 G29 E20=E5>E15 E20 N=44
N35 G29 E20=E5>E16 E20 N=46
N36 G149 T=E30 R1=4
N37 G150 T=E30 R1=E4+E6
N38 S1000 T1 M6  (Milling tool d=18 mm)
N39 G0 X140 Y50 Z-10 B0 F400 M3
N40 G29 E20 E20=1 N=9
N41 M0
N42 (Measuring probe has not made contact, no measurement carried out)
N43 G29 E20 E20=1 N=46
N44 M0
N45 (Slot width too large)
N46 M30
```

Notes

Tool correction:

K0: Tool correction on.
Measuring positions are corrected to take account of tool length and tool radius. Measuring positions in rotational axes do not take tool data into account.

K1: Tool correction off. Measuring positions will not be corrected.

The following assumptions are made if the measuring positions are corrected to take account of the size of the measuring probe:

- the measuring probe lies parallel to the tool axis
- the measuring probe is completely spherical
- the measuring probe moves vertically in relation to the surface being examined.

E parameter:

The number of the E parameter in which the measured axis position is stored (e.g. X7=2 indicates that the measured value in the X axis will be stored in parameter E2. X7=E1 (E1=5) signifies that the measured value will be stored in E5.

Measuring probe status:

E...=0: the programmed end position has been reached, but no measuring point has been detected. The associated E parameters containing the measured values remain unchanged.

E...=1: a measuring point was detected during the measuring operation. The measuring position has been saved in the E parameters.

E...=2: block G145 was executed during a block search operation, a test run or while in Demo mode.

Status monitoring (I3=)

Monitoring of the measuring key status within G145 can be disabled for certain equipments (laser). The laser has no signal. The standard value is zero.

The functions G145 to G150 must not be used when G182 is being used.

In all the specified operating modes, a value of 2 is allocated to the E parameter for the status of the measuring probe. The use of parameters without measuring data can be prevented by testing the value of this parameter in the measuring macros.

23.57 Reading measuring probe status

N... G148 {I1=...} E...

Parameters

E E-Parameter für Meßtasterstatus
Status-Gruppe (1-3)

Example

:
N110 G148 E27
N115 G29 E91=E27=2 E91 N=300
:
N300 M0 (Present mode: block search, test run, demo)
:
N400 M30

Note

Measuring probe status:
I1=1 or is not programmed (standard value)
E...=0: The programmed end position has been reached, but nor mwasuring point has been detected. The assigned E parameters containing measured values have not been modified.
E...=1: A measuring point has been detected. The measuring position has been stored in the E parameters.
E...=2: The G145 block was executed during a block search, test run or demo.
E...=3: Measuring probe error; no measurement possible.

The priority for the measuring key status codes is as follows:
1 : Code 2 (active mode)
2 : Code 3 (measuring key error)
3 : Code 0 or 1 (measuring key contact)

I1=2
E...= 0: no measuring point has been determined during the measurement
E...= 1: a measuring point has been determined during the measurement

I1=3
E...= 0: information from the IPLC (information program logic control): key/laser not enabled
E...= 1: information from the IPLC (information program logic control): key/laser enabled

See the probe system documentation.

During operation with G182 the use of the functions G148 to G150 is not permitted.

23.58 Reading tool or offset values G149

Interrogate current tool :
N.. G149 T0 E..

Interrogate tool dimensions:
N.. G149 T.. {T2=..} {L1=..} {R1=..} {M1=}

Interrogate tool status:
N.. G149 T.. E..

Interrogate current zero point offsets:
N.. G149 N1=0/1 E..

Interrogate pallet offset values:
N.. G149 N1=0/1 E..

Interrogate saved zero point offsets:
With standard zero points or MC84=0:
N.. G149 **N1=51..59** [(axis address)7=..] {(axis address)7=..}

With extended zero points and MC84>0:
N.. G149 **N1=54.[NR]** [(axis address)7=..] {(axis address)7=..} **{B47=...}**
N.. G149 N1=51..59 [(axis address)7=..] {(axis address)7=..}

Interrogate programmable zero point offsets:
N... G149 N1=92 {93} [(axis address)7=...] {(axis address)7=...}

Interrogate current position values of the axes.
N... G149 [(Axis address)7=...]{(Axis address)7=...}

Parameters

Werkzeugdaten
T Werkzeugnummer
Werkzeugkorrekturindex
E-Parameter
E-Parameter für WZ-Länge
E-Parameter für WZ-Radius
E-param. Wkzg.standzeit

Nullpunktverschiebungen
N1= Nullpunktverschiebung
E-Par. für NPV/Position in X
E-Par. für NPV/Position in Y
E-Par. für NPV/Position in Z
E-Par. für NPV/Position in A
E-Par. für NPV/Position in B
E-Par. für NPV/Position in C
E-Parameter für verdr. in B4=

Notes

The tool status can be loaded from the tool memory into the stated E-parameter.
The tool status can have the following values:

E... = 1	Tool has been released and measured
E... = 0	Tool has been released, but has not yet been measured
E... = -1	Tool is blocked

E... = -2 Tool life has been reached
E... = -4 Tool fracture error
E... = -8 Tool cutting force reached
E... = -16 Tool life programmed shorter than T3
A combination of error messages is possible as well:
E... = -13 means: error message -8 and -4 and -2 and 1.

Examples

1: Interrogate number of current tool.
N100 G149 T0 E1
E1 contains the number of the current tool

2: Interrogate dimensions of the current tool.
N100 G149 T12 L1=5 R1=6
E5 contains the tool length
E6 contains the tool radius

3: Interrogate the active zero point offset function
N100 G149 N1=0 E2
N110 G149 N1=1 E3
E2 contains the current zero point offset (51 or 52)
E3 contains the current saved zero point offset (53...59) **or G54.[nr]**

4: Interrogate the offset G54
N100 G149 **N1=54** X7=1 Z7=2
or
N100 G149 **N1=54.[nr]** X7=1 Z7=2

E1 contains the offset in X
E2 contains the offset in Z

5: Calling G54 offset with rotary angle **(MC84>0)**
N100 G149 **N1=54.[nr]** X7=1 B47=2

E1 contains offset in X
E2 contains rotary angle of coordinate system

6: Call up the remaining tool life M1=:
N100 G149 T1 M1=3 (Store remaining tool life of T1 in parameter E3)
E3 enthält die aktive gespeicherte Nullpunktverschiebung (53...59) oder **G54.[nr]**

Notes

The tool correction index 0, 1 or 2 can be specified. The default value is T2=0.

From V400:
T2=0: Tool radius = radius (R) + dimension (R4=).
Tool length = length (L) + dimension (L4=).
It is better to use G321.

23.59 Change tool or offset values G150

Modify tool data in tool memory:
N.. G150 T.. {T2=..} L1=.. R1=.. M1=..

....
....

Modify tool status in tool memory:
N.. G150 T.. E..

Modify zero point offset data in tool memory:
With standard zero points or MC84=0:
N.. G150 **N1=51..59** [(axis address)7=..] {(axis address)7=..}

With extended zero points and MC84>0:
N.. G150 **N1=54.[NR]** [(axis address)7=..] {(axis address)7=..} **{B47=...}**

Parameters

Werkzeugdaten
T Werkzeugnummer
Werkzeugkorrekturindex
E-Parameter
WZ-Länge Wert in T
WZ-Radius Wert in T

Nullpunktverschiebungen
N1= Nullpunktverschiebung
NP-Verschiebung in X
NP-Verschiebung in Y
NP-Verschiebung in Z
NP-Verschiebung in A
NP-Verschiebung in B
NP-Verschiebung in C
Rotationswinkel in B4=

Notes

The tool status can be loaded from the tool memory into the stated E-parameter.
The tool status can have the following values:

E... = 1	Tool has been released and measured
E... = 0	Tool has been released, but has not yet been measured
E... = -1	Tool is blocked
E... = -2	Tool life has been reached
E... = -4	Tool fracture error
E... = -8	Tool cutting force reached
E... = -16	Tool life programmed shorter than T3

A combination of error messages is possible as well:
E... = -13 means: error message -8 and -4 and -2 and 1.

Examples

1. Modify tool data in tool memory:
   N50 G150 T1 L1=E2 R1=4

2. Modify zero point offset data in tool memory:
   N70 G150 **N1=57** X7=E1 Z7=E6
   or
   N70 G150 **N1=54.[nr]** X7=E1 Z7=E6

3. Changing a zero offset with rotary angle of the coordinate system:
   N70 G150 **N1=54.[nr]** X7=E1 B47=E2

4. Change the remaining tool life M1=:
   N110 G150 T1 M1=10 (Change the new remaining tool life of T1 to 10 minutes)

23.60 Cancel cylinder interpolation or activate basic coordinate system G180

Cancel the cylindrical coordinate system or define the main plane and tool axis (basic coordinate system).

N... G180 [main axis 1] [main axis 2] [tool axis]   Basic coordinate system

Parameters

X,Y,Z   Zylinderebene:2 / Werkzeugachse:3
Zylinderebene:1
Radius Zylinder

General principles

The normal expression is G180 X1 Y1 Z1

Only the following configurations are possible:

Main axis 1	X
Main axis 2	Y
Tool axis	Z or W

The correct procedure depends on three different items of information:

1) G17/G18/G19 determines the tool axis (G17 Z).
2) G180 determines the axes to be changed (G17 W in Z)
3) The machine constants for the tool axis definition should be OK. (Tool axis W belongs to Z).

Example

N12340	
N1 G17 T1 M6	
N2 G54	
N3 F1000 S1000 M3	
N4 G180 X1 Y1 Z1	Activate main plane XY and tool axis Z.
N5 G81 Y2 B10 Z-22	Define cycle.
N6 G79 X0 Y0 Z0	Drill with the feed movement in the Z axis.

Notes and usage

Functions G41...G44, G64, axis rotation (G92/G93 B4=) and G141 should be cancelled before G180 is activated.

Tool length compensation is active in the defined tool axis. Radius compensation is active in the main plane.

The machine constants must be used correctly. If the tool axis is the fourth axis, MC117 should be 3 (MC117 = 3) (same as Z axis). MC3401 = 0 (tool axis is a linear axis).

Only Cartesian coordinates can be used.

If G180 is programmed and radius compensation is still active, it will be cancelled by G180.
We recommend to cancel radius compensation, using G40, and to change to the basic coordinate system.

23.61 Cancel / activate cylinder interpolation G182

Selection of the cylindrical coordinate system. This system simplifies the programming of contours and positions on the curved cylinder surface.

Activate the cylindrical coordinate system:
N.. G182 [cylinder axis] [rotational axis] {tool axis}

Rapid feed when G182 in effect:
N.. G0 [cylinder axis] [rotational axis] (tool axis}

Linear feed movement:
N.. G1 [cylinder axis] [rotational axis] (tool axis} {F..}

Circular feed movement:
N.. G2/G3 [cylinder axis] [rotational axis] R..

Return to basic coordinate system:
N.. G180

or
M30, Cancel program softkey, Clear control softkey.

Parameters

Bei G182:
X,Y,Z Zylinderebene:2 / Werkzeugachse:3
Zylinderebene:1
Radius Zylinder

Bei Bewegungen:

X,Y,Z,U,V,W	Linearachsenkoordinate
A,B,C	Rotationsachsenkoordinate
F	Vorschub auf der Zylinderfläche

G182 A1 X2 Z3 R..
or (as until now)
G182 A1 X1 Z1 R..

G182 B1 Y2 Z3 R..
or (as until now)
G182 B1 Y1 Z1 R..

G182 C1 Z2 X3 R..
or (as until now)
G182 C1 X1 Z1 R..

G182 C1 Y2 Z3 R..

Specification of the cylinder plane

Notes

The words X,Y,Z,A,B,C must not be programmed without a value.
The configuration for the cylinder interpolation is programmed in block G182:

- standard configuration

rotational axis	A1	B1	C1
cylinder axis	X1	Y1	Z1
tool axis	Y1/Z1	X1/Z1	X1/Y1
cylinder radius	R	R	R

- enhanced configuration (V321)

rotational axis marked 1	A1	B1	C1
cylinder axis marked 2	X2/Y2/Z2	X2/X2/Z2	Z2/X2/Y2
tool axis marked 3	Y3/Z3/X3	X3/Z3/Y3	X3/Y3/Z3
cylinder radius	R	R	R

Machine constants

The machine constants for the axis definitions have to be correct.

MC 102 = 1, MC103 = 88 (X-axis)
MC 107 = 2, MC108 = 89 (Y-axis)
MC 112 = 3, MC113 = 90 (Z-axis)
MC 117 = 4 belongs to axis 1 (4-3), MC118 = 65 (A-axis turning around X-axis)
MC 122 = 6 belongs to axis 3 (6-3), MC123 = 67 (C-axis turning around Z-axis)

Example

The recess on the curved surface of a cylinder (diameter 40 mm) is to be milled using a dual-point endmilling cutter (diameter 9.5 mm). The working depth is to be 4 mm. The horizontal working of the workpiece is to be performed on the rotational axis C, the cylinder axis Z and the tool axis Y.

```
N12340
N1  G18  S1000  T1  M66
N2  G54
N3  G182  Y1 C1  Z1  R20
N4  G0  Y22 C0  Z15  M3
N5  G1 Y16  F200
N6  G43 Z10
N7  G41
N8  G1 C23.84
N9  G3 Z14.963 C55.774  R15
N10  G1 Z38.691 C116.98
N11  G2 Z42 C138.27  R10
N12  G1 C252.101
N13  G2 Z37 C266.425  R5
N14  G1 Z26
N15  G3 Z10 C312.262  R16
```

```
N16  G1 C365
N17  G40
N18  G41 Z20
N19  G1 C312.262
N20  G2 Z26 C295.073  R6
N21  G1 Z37
N22  G3 Z52 C252.101  R15
N23  G1 C138.27
N24  G3 Z45.383 C95.691  R20
N25  G1 Z21.654 C34.484
N26  G2 Z20 C23.84  R5
N27  G1 C0
N28  G40
N29  G180
N30  G0 Y100  M30
```

Notes

Only cartesian coordinates can be used.

The following functions must not be active when G182 is active:
G41-G44, G64, G92/G93 B4=, G141

The following cannot be programmed when G182 is active:
G25/G26, G27/G28, G51-G59, G61/G62, G70/G71, G73, G92/93.

The selected tool radius should only be fractionally smaller than the width of the recess (undercutting !)

Limitation:

Cylinder radius >5mm <500mm

23.62 Graphic window definition G195

Specify the dimensions and length of a 3D graphics window with reference to the zero point W.

N.. G195 X.. Y.. Z.. I.. J.. K.. {B..} {B1=..} {B2=..}

Parameters

X,Y,Z   Anfangskoordinate
Abmessung parallel zu X
Abmessung parallel zu Y
Abmessung parallel zu Z
Rotation Rundachse hor. (3D)
Rotation Rundachse vert. (3D)
Rotation dritte Rundachse (3D)

Example

```
N9000
N1  G17
N2  G195  X-30  Y-30  Z-70  I170  J150  K100      Graphic window definition
N3  G199                                          Start of graphic contour description
:
```

23.63 End graphic model description G196

N.. G196

Example

:	
N2 G195 X... Y... Z... I... J... K...	Graphic window definition
N3 G199 X... Y... Z.. B.. C..	Start of graphic contour description
N4 G198 X.. Y.. Z.. D..	Start of outside contour description
:	
:	
N25 G197 X.. Y.. D..	Start of inside contour description
:	
:	
N35 G196	End of graphic contour description
:	

23.64 Begin inside / outside contour description G197/G198

Define the start point of an internal contour:
N.. G197 X.. Y.. {Z..} D..

Define the start point of an external contour:
N.. G198 X.. Y.. {Z..} D..

Parameters
X,Y,Z Anfangskoordinate
Tiefe der Kontur

Example
See G199

Notes
The start point of the contour is based on the offset in G199.
The contour must be complete.
The internal contour must lie within the external contour.
An internal contour cannot lie within another internal contour.

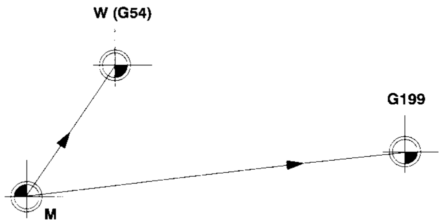
23.65 Begin graphic model description G199

Define the position of a blank contour or a machine part (e.g. chucking equipment) that the tool could collide with. A collision can be detected during the graphical simulation.

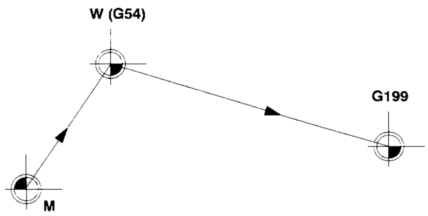
Define a blank contour:
N.. G199 [start coordinates] B1 {C1} {C2}

Define a machine part contour:
N... G199 [start coordinates] B2 {C1} {C2}

Drawing a contour during simulation of the wire plot graph.
N... G199 [initial co-ordinates] B3 {C1} {C2}



Beschreibung bezogen auf M



C2 Beschreibung bezogen auf W

Parameters

X,Y,Z Anfangskoordinate
Model 1=Mat.,2=Masch.,3=Kontur
Nullpunkt 1=Maschine,2=Werkstück

Example

Example

Each chucking tool is described in its own macro. The start point of the chucking tool contour is programmed using two parameters:

E1: X coordinate of the contour start point, in relation to the program zero point
E2: Y coordinate of the contour start point, in relation to the Program zero point

Macro for the left chucking tool:

```
N1991
N1  G92  X=E1  Y=E2
N2  G199  X0  Y0  Z0  B2  C2      Start of graphic contour description
N3  G198  X0  Y0  Z0  D10         Start of outside contour description
N4  G1  X45
N5  Y5
N6  X53
N7  Y25
N8  X45
N9  Y30
N10  X0
N11  Y0
N12  G197  X30  Y15  D-10         Start of inside contour description
N13  G2  I35  J15
N14  G196                         End of graphic contour description
N15  G92  X=-E1  Y=-E2
```

Macro for the right chucking tool (top figure, rotated 180°)

```
N1992
N1  G92  X=E1  Y=E2
N2  G199  X0  Y0  Z0  B2  C2
N3  G198  X0  Y0  Z0  D10
N4  G1  X-45
N5  Y-5
N6  X-53
N7  Y-25
N8  X-45
N9  Y-30
N10  X0
```

```
N11  Y0
N12  G197  X-30  Y-15  D-10      Start of inside contour description
N13  G2  I-35  J-15
N14  G196                        End of graphic contour description
N15  G92  X=-E1  Y=-E2
```

Graphical section of the part program:

```
N199000 (Main program)
N1  G17
N2  G54
N3  S1200  T1  M6
N4  G195  X-20  Y-20  Z-60  I180  J110  K70
N5  G199  X0  Y0  Z0  B1  C2              Start of inside contour description
N6  G198  X0  Y0  D-50                    Start of outside contour description
N7  G1  X70
N8  Y20
N9  X120
N10  Y60
N11  X70
N12  Y80
N13  X0
N14  Y0
N15  G197  X31  Y40  D-20                 Start of inside contour description
N16  G2  I36  J40
N17  G196                                 End of graphic contour description
N18  G22  N=1991  E1=-48  E2=25           Macro call - LH fixture
N19  G22  N=1992  E1=168  E2=55           Macro call - RH fixture
:
N200  M30
```

23.66 Universal pocket milling cycle G200- G208

The universal pocket cycle is an easy and fast way of making CNC-programs for milling pockets of any shape with or without islands.

Program format:

```
N99999
N1  G17
N2  G54
N3        \
:          >                              Normal working
N96       /
N97  G200
N98  G81
N99  G22  N=..                            Rough drill start point
N100 G201 N1=.. N2=..                     Start of pocket description for milling the pocket
N101  G203  N1=..                         Start of pocket contour description
N102    \
:          >                              Pocket contour description
N109    /
N110  G204                                End of pocketncontour description
N111  G205  N1=..                         Start of island contour description
N112    \
:          >                              Contour description island 1
N118    /
N119  G206                                End of island contour description
N120  G205  N1=..                         Start of island contour description
N121    \
:          >                              Contour description island 2
:         /
N130  G206                                End of island contour description
N220  G207 X..  Y..  N=..  N1=..          Call island contour macro
N221  G203 / G205                         Start pocket / island contour description
N222  G208                                Contour description for parallelogramm
N223  G204 / G206                         End of pocket / island contour description
N131  G202                                End of pocket contour cycle
N350  G22  N=..                           Rework the contour
N351  G22  N=..                           Rework island 1
N352  G22  N=..                           Rework island 2
:
N500  M30
```

23.67 Create pocket cycle macro's G200

N.. G200

This function must be programmed before the universal pocket cycle and indicates that:

- the coordinates for the milling paths must be calculated (if they have not already been calculated).
- the milling paths are programmed in a macro generated by the CNC; the number (N1=..) of this macro is specified in a G201 block.
- if necessary (indicated by N2=.. in a G201 block) a second macro to drill the start point will be generated.
- if necessary (indicated in a G203 or G205 block) the macros (N1=..) for reworking the contours will be generated.

All operating conditions, such as processing level, zero point offsets and tool corrections should be specified before the G200 function is executed.

Point definitions (G78), which are used to specify the pocketn contour, should be defined before the G200 block.

A G200 block can be incorporated in a macro; the pocket will, however, only be searched for in macros nested at a lower level.

The CNC expands the macros before the program is executed. Blocks between G200 and G201 are therefore ignored at first. These blocks are processed once the macros have been generated.

All universal pocket cycles programmed between a block G200 and G202 or M30 are calculated simultaneously.

The machining level (G17/G18/G19) has to be defined before G200 or after G202 was programmed.

Note

As of V321 the user can no longer see generated macros in the macro memory. If a macro is to be used in another program, the relevant macro number must first be entered in the macro memory. Only then will the macro be visible in the macro memory and can it be read in/out.

23.68 Start contour pocket cycle G201

Start of the definition of a pocket (possibly including islands). The block contains the technological data required for calculating the milling paths. The milling of the pocket starts from the G201 block.

N... G201 Y... Z... N1=.. N2=.. {B...} {I..} {J..} {K..} {R..} {F..} {F2=..}

Parameters

Y Bearbeitungsaufmaß
Gesamte Taschentiefe

Diese Wörter werden durch die gewählte Bearbeitungsebene bedingt.
B Sicherheitsabstand
WZ-Radius für Berechnung
Schnittbreite des Fräsers in %

Das I-Wort ist vorzeichenlos. Wenn I nicht programmiert ist, wird der unter MC720 gespeicherte Wert verwendet.

J J1:Gleichlauf / J-1:Gegenlauf
Zustelltiefe
Vorschub Taschenfräsen
Eintauchvorschub
Makronummer für Ausfräsen
Startpunkt Makronummer

Notes

The addresses (especially Y and Z) are independent of the active level.

The functions G90, G40 and G63 are activated automatically when the G201 function is executed.

The functions G201/G202, G203/G204 and G205/G206 must all be in the same program/macro.

The only blocks that may appear between G201 and G202 are: G203/G204, G205/G206 and G207.

The only blocks that may appear between G203/G204 and G205/G206: G1, G2/G3, G208, G63/G64, G90, G91.
The movements G1, G2/G3 are limited to the main level. Tool axes and rotary axis coordinates are not permitted.

The program is to be continued from an absolute position after describing the pocket.

E parameters can be used for contour descriptions. Calculations must be performed before G200 is executed.

23.69 End contour pocket cycle G202

Termination of the full pocket description.

N.. G202

Note

The functions G0, G40, G63 and G90 are activated automatically when the G202 function is executed.

With G202 the calculation of universal pocket cycles is stopped. Calculation is continued with the next G200.

23.70 Start pocket contour description G203

N.. G203 X.. Y.. Z.. N1=.. {P..} {B1=..} {B2=..} {L2=..} {P1=..}

Parameters

X Startpunkt in X
Startpunkt in Y
Startpunkt in Z
Makronummer für Nachbearbeitung
Punktedefinitionsnummer
Rotationswinkel Taschenkontur
Startpunkt Polarwinkel
Startpunkt Polarlänge
Punktedefinitionsnummer

Die Werkzeugachsenkoordinaten müssen immer im G203-Satz enthalten sein.

Notes

G1, G63 and G90 are activated automatically when G203 is executed.

The first point of a contour description must be specified in a G203 block. Reworking of the contour also starts at this point.

The bottom of the pocket must lie parallel to the processing level.

The edges of the pocket must be at right angles to the bottom of the pocket.

Two elements of the same pocket must not intersect or touch.

During finish-cutting, the programmer must ensure that a tool diameter is selected that is smaller than the narrowest section in the pocket of the work piece. Contour violations during finishing are not detected by the controller.

23.71 End pocket contour description G204

This function terminates the description of the pocket contour.

N.. G204

23.72 Start island contour description G205

The contour of an island is described in the same way as the contour of a pocket. The description begins with G205 and the absolute start position of the island.

N.. G203 X.. Y.. N1=.. {Z..} {P..} {B1=..} {B2=..} {L2=..} {P1=..}

Parameter

X Startpunkt in X
Startpunkt in Y
Startpunkt in Z
Makronummer für Nachbearbeitung
Punktedefinitionsnummer
Rotationswinkel Inselkontur
Startpunkt Polarwinkel
Startpunkt Polarlänge
Punktedefinitionsnummer

Notes

The CNC assumes that the tops of the island and the pocket are at the same level.

If the island is higher than the top of the pocket, the B word in the G201 block can be used to prevent a collision between the milling tool and the work piece while moving from one starting point to the next.

G205 activates G1, G63 and G90.

The tool axis must not be programmed.

The contour of an island must be complete.
Two islands may not intersect or touch.
Islands must be situated in the pocket and must not intersect or touch the edges.
The sides of an island must be at right angles to the bottom.

23.73 End island contour description G206

The contour description is terminated by G206. The description for pocket contours applies equally to island contours.

N.. G206

23.74 Call island contour macro G207

N... G207 X.. Y.. Z.. N=.. N1=..

There are three possibilities:

1. The same island appears at another place in the same pocket contour.
2. The same island contour appears in another pocket contour.
3. The same island contour appears in another program.

As the island contour is included in a macro, the three options can all be handled in the same way.

Parameters

X Verschiebung in X
Verschiebung in Y
Verschiebung in Z
Makro mit Inselkonturbeschreibung
Makronummer für Nachbearbeitung

Das Makro der Inselkontur lautet:
N9xxx G205 X=X2 Y=Y2 N1=..

```
N..    \
:       > Inselkontur
N..    /
N.. G206
```

N9xxx stellt hier die Makrokennzeichnung dar.
Das Makro wird mit der Funktion G207 aufgerufen.

```
N.. G201
:
N.. G207 N=9xxx
N.. G207 N=9xxx X=(X1-X2) Y=(Y1-Y2)
N.. G202
```

Example

1 : An island whose contours are programmed as a macro
P1 : Starting point of the contour description (G205 block)
2 : Desired position of the island
P2 : Starting point of the moved contour

X..: Distance parallel to the X axis of P1 to P2
Y..: Distance parallel to the Y axis of P1 to P2

Notes

The subprogram called up in the G207 block must not contain any references to G63/G64.
The best procedure is to start an island contour with the coordinates X0,Y0 (zero offset).The starting point can be programmed in the G207 block without calculation.

The identical macro of the island contour will then be as follows:

```
N9xxx G205 X0 Y0 N1=..
N..        \
:           > Island contour with zero offset
N..        /
N.. G206
```

N9xxx represents the macro identification.
The macro is called with the G207 function.

```
N.. G201
:
N.. G207 N=9xxx X=X2 Y=Y2
N.. G207 N=9xxx X=X1 Y=Y1
N.. G202
```

The subprogram for the island contours can be programmed in absolute or incremental dimensions.

23.75 Quadrangle contour description G208

The G208 function enables a regular quadrangle, particularly a rectangle or a parallelogram, to be programmed very easily.

N... G208 X.. Y.. {Z..} {I..} {J..} {R..} {B1=..}

Parameter

X Länge in X
Länge in Y
Länge in Z
Fasenlänge
J1:Gleichlauf / J-1:Gegenlauf
Rundungsradius
Winkel Viereckkontur

Example

G203 X (=X1) Y (=Y1) Z (=Z1) B1= (=A)
G208 X (=X) Y (=Y) B1= (=B)
G204

Note

The bottom of the pocket must always be parallel to the main level.

Example of a pocket contour

Pocket with islands. Rough drilling of the starting point and reworking of the contours are taken into account.

```
N82150
N1 G17
N2 G54
N3 G98 X-10 Y-10 Z10 I320 J320 K-60
N4 G99 X0 Y0 Z0 I300 J300 K-40
N5 F200 S3000 T2 M6
N6 G200
N7 G81 Y2 Z-20 M3                        (Predrill start point)
N8 G22 N=9992
N9 S2500 T3 M6                           (Evacuate pocket)
N10 G201 Y0.1 Z-20 B2 I50 R10 F200 N1=9991 N2=9992 F2=100
N11 G203 X70 Y40 Z0 N1=9993              |
N12 G64                                  |
N13 G1 X260 B1=0 I1=0                    |
N14 G1 I30                               |
N15 G1 X260 Y260 B1=90 I1=0              |(Pocket contour)
N16 G1 I30                               |
N17 G1 X40 Y260 B1=180 I1=0              |
N18 G1 I30                               |
N19 G1 X40 Y70 B1=270                    |
N20 G63                                  |
N21 G204                                 |
N22 G205 X100 Y80 N1=9994
N23 G208 X-30 Y30 J-1                    (Island 1)
N24 G206
N25 G205 X190 Y80 N1=9995
```

```
N26 G91
N27 Y50                                 (Island 2)
N28 X40 Y-50
N29 G90
N30 G206
N31 G205 X150 Y130 N1=9996
N32 G2 I150 J150                        (Island 3)
N33 G206
N34 G205 X110 Y210 N1=9997
N35 G208 X-40 Y40 J-1 B1=135            (Island 4)
N36 G206
N37 G205 X180 Y200 N1=9998
N38 G91
N39 G1 Y30
N40 X20                                 (Island 5)
N41 X30 Y-30
N42 G90
N43 G206
N44 G202
N45 F200 S2200 T4 M6                    |
N46 G22 N=9993                          |
N47 F200 S2500 T5 M6                    |
N48 G22 N=9994                          |(Reworking)
N49 G22 N=9995                          |
N50 G22 N=9996                          |
N51 G22 N=9997                          |
N52 G22 N=9998                          |
N53 G0 Z100 M30
```

24. Special G functions for macros

The range G300 upto G399 (exept G301) is for special functions. These functions are used in the macro development for Cycle Design, Easy Operate and IPLC.

24.1 Programming error messages G300

Programming of error messages when universal programs or macros are executed.

N... G300 D...

Parameters

D	Fehlermeldungs-Nummer

Example

Defining error messages if a programmed angle is not allowed.

N9999 (macro for calculating table rotation)	
: (E4 is input value for angle phi)	
:	
N110 G29 I1 E30 N=180 E30=(E4>360)	If E4 > 360°, jump to N180
N120 G29 I1 E30 N=210 E30=(E4<0)	If E4 < 0°, jump to N210
N150 G29 I1 E30 N=290 E30=1	Jump to N290 (0° <= E4 <= 360°)
:	
N180 G300 D190	Error message (phi >360°): Programmed value > maximum value Program should be ended and a modified E4 must be entered
N190	
N210 G300 D191 (programmed value < minimum value)	
:	
N290	Normal program

Notes and usage

This function may only be used in programs and macros.

Only error messages from the existing P-error list may be used. (See error lists P, O and F in chapter: Miscellaneous).

24.2 Error message in read-in program or macro G301

Error message in read-in program or macro.

N... G301 (O... wrong original block)

Parameters

keine

Example

Correct program stored on hard disk.
Program was made with MC84 = 0.

```
N9999 (program ...)
N1 G17
N2 G57
N3 T1 M6
N4 F200 S1000 M3
:
N99 M30
```

Faulty program in RAM.
Extended zero offset is active (MC84 > 0)

```
N9999 (ERR*) (program ...)
N1 G17
N2 G301 (O138  G57)     G301 indicates that the block is incorrect. G57 should be G54 I3.
N3 T1 M6
N4 F200 S1000 M3
:
N99 M30
```

This faulty program can be executed. It will stop at the G301 block. Error P33 (other text in defined block) appears. Change this block and restart the program.

Notes and usage

G301 is generated when a reading error is found when a program or macro is read-in. The function can only be used in faulty programs and macros.

The function cannot be entered in MDI.

The error messages relate to the existing O-errors. (See error lists P, O and F in chapter Miscellaneous).

24.3 Query active technology - G319

Query active F (advance), S (speed in rpm) or T (tool number).

N... G319 I1=.. E...

Parameters

I1=	1=Vorschub, 2=Drehzahl, 3=WZ-Nr

Which function:
I1=1 Advance (F)
I1=2 speed in rpm (S)
I1=3 tool number (T)

Example

Selection of the active advance and storing the value in E parameter 10.

N... G319 I1=1 E10 (I1=1 query advance value, E10 then contains the value)

Notes

This function is only allowed to be used within the programs and the macros.

24.4 Query tool table G321

Query values extracted from the tool table.

N... G321 T.. I1=.. E...

Parameters

T	Werkzeugnummer
	WZ-Adresse (1=L .. 28=Q5)
	E-Parameter

Assignment:

I1=1	L	tool length
I1=2	R	tool radius
I1=3	C	tool corner radius
I1=4	L4=	length oversize
I1=5	R4=	radius oversize
I1=6	G	graphics
I1=7	Q3=	tool type
I1=8	Q4=	number of tool cogs
I1=9	I2=	direction of cut
I1=10	A1=	entry angle
I1=11	S	tool size
I1=12	E	tool status
I1=13	M	tool parking time
I1=14	M1=	residual tool parking time
I1=15	M2=	tool parking time monitoring
I1=16	B	tool breakage tolerance
I1=17	B1=	tool breakage monitoring
I1=18	L1=	first supplementary tool length
I1=19	R1=	first supplementary tool radius
I1=20	C1=	first supplementary tool corner radius
I1=21	L2=	second supplementary tool length
I1=22	R2=	second supplementary tool radius
I1=23	C2=	second supplementary tool corner radius
I1=24	L5=	wear tolerance length
I1=25	R5=	wear tolerance radius
I1=26	L6=	measurement offset length
I1=27	R6=	measurement offset radius
I1=28	Q5=	tool breakage monitoring cycle (0-9999)
I1=28	Q5=	WZ-Bruchübewachungs Zyklus (0-9999)

Example

Program sentences for querying the tool table.

N30 G321 T10 I1=1 E1 (Read command, T (tool number), I1=1 information about the tool address, E1 is the E parameter, L (tool length) is set in E parameter 1)

N40 G321 T10 I1=2 E10 (R (tool radius) is set in E parameter 10)
N50 G321 T10 I1=3 E20 (C (tool corner radius) is set in E parameter 20. If C has no value, E20=-999999999 is set)

N60 G321 T10 I1=4 E2 (L4 (length oversize) is set in E parameter 2)
N70 G321 T10 I1=5 E11 (R4 (radius oversize) is set in E parameter 11)
N80 E3=E1+E2 (the correct tool length (E3) is L+L4 (E1+E2))

N90 E12=E10+E11 (the correct tool radius (E12) is R+R4 (E10+E11))

Notes

This function is only allowed to be used within programs and macros.

The tool number (T) must be known. The position (P) in the tool table cannot be queried.

If the E parameter contains the number -999999999, the address in the tool table is empty.

24.5 Calling machine constant values G322

Readout of a machine constant value and storing this value in the relevant E-parameter.

N... G322 E.. N1=...

Parameters

N1=	Maschinenkonstantennummer
E-Parameter	

Example

Universal program blocks which can be used for either type of zero point table.

N40 E5= E6=	
N50 G322 N1=84 C10	Machine constant 84 is defined in E10
N60 G29 E1 N=90 E1=E10>0	Compare if MC84 > 0. If so, jump to N90
N70 G150 **N1=57** X7=E5 Z7=E6	Changing the zero offset table ZO.ZO
N80 G29 E1 N=100 E1=1	Jump to N100
N90 G150 **N1=54.03** X7=E5 Z7=E6	Changing the zero offset table ZE.ZE
N100 ..	

Notes and usage

This function may only be used in programs and macros.

If invisible addresses are called in the machine constant table, the E-parameter is not changed.

24.6 Query current modal G function G324 (from V400)

Query current modal G function and stores with this value in the E parameters pre-programmed for this purpose.

N... G324 I1=.. E...

Parameters

I1=	G-Gruppe (1 - 27)
E-Parameter	

Assignment:

I1=	G function
1	G0,Gg1,G2,G3,G6,G9
2	G17,G18,G19
3	G40,G41,G42,G43,G44,G141
4	G53,G54,G54_I,G55,G56,G57,G58,G59
5	G64,G63
6	off,G81,G83,G84,G85,G86,G87,G88,G89,G98
7	G70,G71
8	G90,G91
10	G94,G95
13	G72,G73
14	G66,G67
15	off,G39
16	G51,G52
17	G196,G199
19	G27,G28
20	G25,G26,G26_S,G26_F_S
21	off,G9
22	G202,G201
24	G180,G182,G180_XZC
27	off,G7
28	off,G8

Example

Selection of the G function (I1=2) and storage of the value in E parameter 10.
N... G324 I1=2 E10

I1=2 query Group 2 G function
E10 holds the result

E10 =17	G17 is active
E10 =18	G18 is active
E10 =19	G19 is active

Notes

This function is only allowed to be used within programs and macros.

If the group or the G function does not exist, the E parameters are unchanged.

24.7 Query current modal M function G325 (from V400)

Query current modal M function and store this value in the E parameter pre-programmed for this purpose.

N... G325 I1=.. E...

Parameters

I1= M-Gruppe (1 - 10)
E-Parameter

Assignment:

I1=	M function
1	M5,M3,M4,M13,M14,M19,No_dir
2	M40,M41,M42,M43,M44,No_range
3	M9,M7,M8
4	off,M17,M18,M16
5	off,M10,M11
6	off,M22,M23
7	off,M32,M33
8	off,M55
9	off,M51,M52
10	off,M53,M54

Example

Selection of the M function (I1=1) and storage of its value in the E parameter 10.
N... G325 I1=1 E10

I1=1 Query Group 1 function
E10 holds the result
E10 =5 M5 is active
E10 =3 M3 is active
E10 =4 M4 is active

Notes

This function is only allowed to be used within programs and macros.

If the group or the M function does not exist, the E parameter is unchanged.
(off equates to 0).

24.8 Calling up the current axis position values G326

Calling up a current axis position value and storing this value in the E-parameter provided for this purpose.

N... G326 {X7=..} {Y7=..} {Z7=..} {A7=..} {B7=..} {C7=..}

Parameters

Y7= E-Parameter für Meßwert in Y
= E-Parameter für Meßwert in Z
= E-Parameter für Meßwert in X
= E-Parameter für Meßwert in A
= E-Parameter für Meßwert in B
= E-Parameter für Meßwert in C

Examples

Calling up current axis position values of X,Y and Z and storing the values in the E-parameters 20, 21 and 22.

N... G326 X7=20 Y7=21 Z7=22

Continuation of program after universal pocket milling cycle.

N30 G202	End of pocket milling cycle
N40 G326 X7=20 Y7=21	Unknown current end position of X and Y
N50 G29 E1 N=90 E1=E20>100	When current X-position >100, jump to N90
N60 G29 E1 N=90 E1=E20<-100	When current X-position <-100, jump to N90
N70 G0 X-110	G0 movement to X-110, when the current X-position is between 100 and -100. In this way one can, for instance, bypass an obstacle
N80 G0 Y 100	Further evasive movement
N90 ..	

Notes and use

This function is only to be used within programs and macros.

If the axis does not exist, the E-parameter is not changed.

With graphic simulation the X-,Y- and Z-axes are called up correctly. The rotary axes remain zero.

24.9 Scan current operating mode G327 (from V410)

To scan the current operating mode and store this value in the E parameter provided.

Format

N... G327 I1=.. E...

Parameters

I1= Lese Betriebsart
E-Parameter

Notes and use

This function must only be used within programs and macros.

Arrangement of group

Group

I1=	Operating mode	
1	Easy Operate	0 = not active, 1=active
2	Single record	0 = not active, 1=active
3	Graphic	0 = not active, 1=active
4	Test run	0 = not active, 1=active
5	Search	0 = not active, 1=active
6	Demo	0 = not active, 1=active

Examples

Fetch operating mode (I1=1) and store the value in E parameter 10.

N... G327 I1=1 E10

Explanation
I1=1 : Check whether Easy Operate is active.
E10 contains the result: 0= not active, 1=active.

24.10 Write to the tool table G331 (from V400)

Write from values in the tool table.

N... G331 T.. I1=.. E...

Parameters

T	Werkzeugnummer
	WZ-Adresse (1=L .. 28=Q5)
	E-Parameter

Assignment

I1=1	L	tool length
I1=2	R	tool radius
I1=3	C	tool corner radius
I1=4	L4=	length oversize
I1=5	R4=	radius oversize
I1=6	G	graphics
I1=7	Q3=	tool type
I1=8	Q4=	number of tool cogs
I1=9	I2=	direction of cut
I1=10	A1=	entry angle
I1=11	S	tool size
I1=12	E	tool status
I1=13	M	tool parking time
I1=14	M1=	residual tool parking time
I1=15	M2=	tool parking time monitoring
I1=16	B	tool breakage tolerance
I1=17	B1=	tool breakage monitoring
I1=18	L2=	first supplementary tool length
I1=19	R2=	first supplementary tool radius
I1=20	C2=	first supplementary tool corner radius
I1=21	L3=	second supplementary tool length
I1=22	R3=	second supplementary tool radius
I1=23	C3=	second supplementary tool corner radius
I1=24	L5=	wear tolerance length
I1=25	R5=	wear tolerance radius
I1=26	L6=	measurement offset length
I1=27	R6=	measurement offset radius
I1=28	Q5=	tool breakage monitoring cycle (0-9999)

The tool commentary cannot be changed.

Example

N10 E5=100 (tool length)
N11 E6=10 (tool radius)
N12 E7=-999999999 (tool corner radius)
N13 E8=0 (length oversize)
N14 E9=0 (radius oversize)

N..
N20 G331 T10 I1=1 E5
N21 G331 T10 I1=2 E6
N22 G331 T10 I1=3 E7
N23 G331 T10 I1=4 E8
N23 G331 T10 I1=5 E9

N30 T10 M67
N..
N40 G321 T10 I1=4 E8
N41 E8=0.3
N42 G331 T10 I1=4 E8
N43 T10 M67

Explanation:

N10 : L (tool length) is set in E parameter 5
N11 : R (tool radius) is set in E parameter 6
N12 : C (tool corner radius is set in E parameter 7 (if C has no value, E7= must be set to -999999999)
N13 : L4 (length oversize) is set in E parameter 8
N14 : R4 (radius oversize) is set in E parameter 9

N20 : L (tool length) writing of E parameter 5 in the tool table
N21 : R (tool radius) writing of the E parameter 6 in the tool table
N22 : C (tool corner radius) writing of the E parameter 7 in the tool table
N23 : L4 (length oversize) writing of the E parameter 8 in the tool table
N24 : R4 (radius oversize) writing of the E Parameter 9 in the tool table

N30 : the tool must be activated once more with the changed information

N40 : L4 (length oversize) E parameter 8 is set to 0.3
N41 : L4 (length oversize) writing of E parameter 8 in the tool table

N50 : the tool must be activated once more with the changed information

Notes

This function is only allowed to be used within programs and macros.

The tool number (T) must be known. The position (P) in the tool table cannot be changed.

If the E parameter contains the value -999999999, the address in the tool table becomes empty.

The changed tool information must be activated again following the writing. (T.. M67)

24.11 Writing to the machine settings store G332 (from V400)

Writing of a machine setting value.

N... G332 E.. N1=...

Parameters

N1=	Maschinenkonstantennummer
E-Parameter	

Example

Writing of several machine setting values.

N30 E5=... (measured X value, e.g using calibration)	E5 contains the measured X value
N40 E6=... (measured Y value)	
N50 E7=... (measured Z value)	
N60 G332 N1=3155 E5	Changing of the machine setting value 3155 using E parameter 5
N70 G332 N1=3255 E6	
N80 G332 N1=3355 E7	
N90 ..	

Notes

This function is only allowed to be used within programs and macros.

Only machine settings which can be changed by the operator (OPER_MC) can be entered. If the machine settings are not allowed to be changed, Error P195 (Only OPER_MCs) is generated.

If the value of the machine settings is not allowed, error message P190 (Programmed value > highest value) is generated.

25. G-functions produced by cycle design

25.1 Cycle Design

Cycle Design allows the user to define his own G functions and integrate them in the control. These G functions can be programmed within part programs using graphics support.

Note

Refer as well to your Programming manual.

25.2 Laser: Calibration G600 (from V410)

To determine the position of the laser measuring instrument and store this position value in the machine constants provided.

Format

N... G600 {X... Y... Z...} {S...}

Parameters

X	Position measuring instrument
Y	Position measuring instrument
Z	Position measuring instrument
S	Speed (recommended value S3000)

Associated functions

G601, G602, G603, G604

Notes and use

The machine and MillPlus must be prepared by the machine manufacturer for the measuring instrument. If not all the G functions described here are available on your machine, consult your machine handbook.

Example

Calibrating laser measuring instrument and storing the position value in the E parameter.

N... G600 X300 Y500 Z600 S3000

Note

Refer as well to chapter Calibration of the laser measurement system.

25.3 Laser: Measure tool length G601 (from V410)

To measure the length of centric tools.

Format

N... G601 {S...}

Parameter

S	Speed (recommended value S3000)

Folgende Adressen vom Werkzeugspeicher werden verwendet:

L	Werkzeuglänge
L4=	Aufmaß Länge
L5=	Längentoleranz
E	Werkzeugstatus

Associated functions

G600, G602, G603, G604

Notes and use

The machine and MillPlus must be prepared by the machine manufacturer for the measuring instrument. If not all the G functions described here are available on your machine, consult your machine handbook.

Note

Refer as well to chapter Measurement of the length of concentric tools.

25.4 Laser: Measureng length and radius G602 (from V410)

To measure the length and radius of acentric tools with laser measuring instrument

Format

N... G602 {S...}

Parameters

S	Speed (recommended value S3000)

Folgende Adressen vom Werkzeugspeicher werden verwendet:

L	Werkzeuglänge
L4=	Aufmaß Länge
L5=	Längentoleranz
R	Werkzeugradius
R4=	Aufmaß Radius
R5=	Radiustoleranz
L6=	Position oberhalb der Werkzeugspitze für Rundlaufkontrolle.
R6=	Radius-Position für Längenvermessung.
Q4=	Anzahl der Zähne
E	Werkzeugstatus

Associated functions

G600, G601, G603, G604

Notes and use

The machine and MillPlus must be prepared by the machine manufacturer for the measuring instrument. If not all the G functions described here are available on your machine, consult your machine handbook.

Note

Refer as well to chapter Measurement of the length and radius of eccentric tools.

25.5 Check of individual edge G603 (from V410)

To monitor the lower part (inspection height) of the tool with a laser measuring instrument.

Format

N... G603 {I1=...} {F2=...}

Parameters

I1=	Inspection height
F2=	Inspection feed

Folgende Adressen vom Werkzeugspeicher werden verwendet:

L	Werkzeuglänge
L4=	Aufmaß Länge
R	Werkzeugradius
R4=	Aufmaß Radius
R5=	Radiustoleranz
L6=	Position oberhalb der Werkzeugspitze für Rundlaufkontrolle.
Q4=	Anzahl der Zähne
E	Werkzeugstatus

Associated functions

G600, G601, G602, G604

Notes and use

The machine and MillPlus must be prepared by the machine manufacturer for the measuring instrument. If not all the G functions described here are available on your machine, consult your machine handbook.

Note

Refer as well to chapter Individual cutting control.

25.6 Laser: Tool breakage control G604 (from V410)

Tool breakage check

Format

N... G604 {S...}

Parameters

S	Speed (recommended value S3000)

Folgende Adressen vom Werkzeugspeicher werden verwendet:

L	Werkzeuglänge
L4=	Aufmaß Länge
R	Werkzeugradius
R4=	Aufmaß Radius
B	Bruchtoleranz in mm.
R6=	Radius-Position für Bruchkontrolle.
E	Werkzeugstatus

Associated functions

G600, G601, G602, G603

Notes and use

The machine and MillPlus must be prepared by the machine manufacturer for the measuring instrument. If not all the G functions described here are available on your machine, consult your machine handbook.

Note

Refer as well to chapter .

25.7 TT120/TT130: Calibration G606 (from V410)

To determine the position of the measuring instrument and store this position value in the machine constants provided.

Format

N... G606 {X... Y... Z...}

Parameter

X	Position of measuring instrument
Y	Position of measuring instrument
Z	Position of measuring instrument

Associated functions

G607, G608, G609

Notes and use

The machine and MillPlus must be prepared by the machine manufacturer for the measuring instrument. If not all the G functions described here are available on your machine, consult your machine handbook.

Note

Refer as well to chapter Calibration of the TT120/TT130.

25.8 TT120/TT130: Measure tool length G607 (from V410)

To measure the length of tools.

Format

N... G607 {I1=...} {I2=...}

Parameter

I1=	safety distance
I2=	0=complete 1=per tooth

Folgende Adressen vom Werkzeugspeicher werden verwendet:

L	Werkzeuglänge
L4=	Aufmaß Länge
R	Werkzeugradius
R4=	Aufmaß Radius
R6=	Meßversatz Radius
E	Werkzeugstatus

Associated functions

G606, G608, G609

Notes and use

The machine and MillPlus must be prepared by the machine manufacturer for the measuring instrument. If not all the G functions described here are available on your machine, consult your machine handbook.

Note

Refer as well to chapter Tool length measurement……

25.9 TT120/TT130: Measuring tool radius G608 (from V410)

To measure the tool radius.

Format

N... G608 {I1=...} {I2=...}

Parameter

I1=	safety distance
I2=	0=complete, 1=per tooth

Folgende Adressen vom Werkzeugspeicher werden verwendet:

L	Werkzeuglänge
L4=	Aufmaß Länge
R	Werkzeugradius
R4=	Aufmaß Radius
E	Werkzeugstatus

Associated functions

G606, G607, G609

Notes and use

The machine and MillPlus must be prepared by the machine manufacturer for the measuring instrument. If not all the G functions described here are available on your machine, consult your machine handbook.

Note

Refer as well to chapter Measurement of tool radius

25.10 TT120/TT130: Measure length and radius G609 (from V410)

To measure tool length and radius.

Format

N... G609 {I1=...} {I2=...}

Parameter

I1=	Safety distance
I2=	0=complete, 1=per tooth

Folgende Adressen vom Werkzeugspeicher werden verwendet:

L	Werkzeuglänge
L4=	Aufmaß Länge
R	Werkzeugradius
R4=	Aufmaß Radius
R5=	Radiustoleranz
E	Werkzeugstatus

Associated functions

G606, G607, G608

Notes and use

The machine and MillPlus must be prepared by the machine manufacturer for the measuring instrument. If not all the G functions described here are available on your machine, consult your machine handbook.

Note

Refer as well to chapter Measure the tool completely

26. Liste der G-Funktionen und M-Funktionen

26.1 G-Funktionen

G..	Beschreibung	Modal
G0	Eilgang	*
G1	Linearinterpolation	*
G2 **G3**	Kreis im Uhrzeigersinn Kreis im Gegenuhrzeigersinn	*
G4	Verweilzeit	-
G6	Splineinterpolation	*
G7	Bearbeitungsebene schwenken	
G8	Werkzeugrichtung schwenken	
G9	Definier Polposition	*
G11	Polarkoordinate,Eckenrundung,Fase	-
G14	Wiederholfunktion	
G17 **G18** **G19**	Bearbeitungsebene XY, Werkzeug Z Bearbeitungsebene XZ, Werkzeug Y Bearbeitungsebene YZ, Werkzeug X	*
G22 **G23**	Makroaufruf Hauptprogrammaufruf	-
G25 **G26**	Vorschuboverride wirksam Vorschuboverride nicht wirksam	*
G27 **G28**	Positionierfunktionen löschen Positionierfunktionen	*
G29	Bedingter Sprungbefehl	-
G39	Werkzeug-Aufmaß aktivieren	*
G40 **G41** **G42** **G43** **G44**	Keine Werkzeugradiuskorrektur Werkzeugradiuskorrektur, links Werkzeugradiuskorrektur, rechts WZ-Radiuskorrektur bis Endpunkt WZ-Radiuskorrektur über Endpunkt	*
G45 **G46**	Messen eines Punktes	-

G..	Beschreibung	Modal
G46 + **M26** **G49** **G50**	Messen eines Vollkreises Messtaster kalibrrieren Vergleich der Toleranzwerte Verrechnung der Meßwerte	
G51 **G52**	Aufheben G52 Achsenverschiebung Aktivieren G52 Achsenverschiebung	*
G53 **G54** **G55** **G56** **G57** **G58** **G59** **G54 I1** .. **G54** **I99**	Aufheben der NPV (G54-59) NP-Verschiebung aktivieren NP-Verschiebung aktivieren NP-Verschiebung aktivieren NP-Verschiebung aktivieren NP-Verschiebung aktivieren NP-Verschiebung aktivieren NP-Verschiebung aktivieren	*
G61 **G62**	Tangentiales Anfahren Tangentiales Wegfahren	-
G63 **G64**	Aufheben der Geometrieberechnung Geometrieberechnung aktivieren	*
G70 **G71**	Maßeinheit: Inch Maßeinheit: Metrisch	*
G72 **G73**	Spiegeln und Maßfaktor aufheben Spiegeln und Maßfaktor aktivieren	*
G74	Absolutposition	-
G77	Lochkreiszyklus	-
G78	Punktedefinition	-
G79	Zyklusaufruf	-
G81 **G83** **G84** **G85** **G86** **G87** **G88** **G89**	Bohrzyklus Tieflochbohrzyklus Gewindebohrzyklus Reibzyklus Ausdrehzyklus Rechtecktaschenfräszyklus Nutenfräszyklus Kreistaschenfräszyklus	*
G90 **G91**	Absolutprogrammierung Inkrementalprogrammierung	*
G92 **G93**	NP-Verschiebung inkr./Rotation NP-Verschiebung abs./Rotation	*

G..	Beschreibung	Modal
G94 **G95**	Vorschub in mm/min (Inch/min) Vorschub in mm/U (Inch/U)	*
G98 **G99**	Grafikfensterdefinition Grafik: Materialdefinition	-
G141	3D-Werkzeugkorrektur	*
G145 **G148** **G149** **G150**	Lineare Meßbewegung Meßtasterstatus abfragen Werkzeug- oder NPV-Werte abfragen Ändern Werkzeug- oder NPV-Werte	-
G180 **G182**	Zylinderinterpolation aufheben Zylinderinterpolation aktivieren	*
G195 **G196** **G197** **G198** **G199**	Grafikfensterdefinition Grafikkonturbeschreibungsende Anfang Innenkonturbeschreibung Anfang Außenkonturbeschreibung Anfang Grafikkonturbeschreibung	-
G200 **G201** **G202** **G203** **G204** **G205** **G206** **G207** **G208**	Taschenfräszyklenmakros erzeugen Konturtaschenfräszyklusanfang Konturtaschenfräszyklusende Konturtaschenbeschreibungsanfang Konturtaschenbeschreibungsende Inselkonturbeschreibungsanfang Inselkonturbeschreibungsende Aufruf Inselkonturmakro Viereckkonturbeschreibung	*

26.2 Liste der G-Funktionen für Makros und IPLC

G..	Beschreibung	Modal
G300 **G301** **G319** **G321** **G322** **G324** **G325** **G326** **G327** **G331** **G332**	Program error call Program halt Read actual technology data Read tool data Read machine constant memory Read G-group Read M-group Read actual position Abfragen aktuelle Betriebsart Schreiben in die Werkzeugtabelle Schreiben in den Maschinenkonstantenspeicher	-

26.3 Liste der G-Funktionen Zyklen Design

G..	Beschreibung	Modal
G600	Lasersystem: Kalibrieren	

G..	Beschreibung	Modal
G601	Lasersystem: Länge vermessen (zentrischen Werkzeugen)	-
G602	Lasersystem: Länge und Radius (azentrischer Werkzeuge) vermessen	
G603	Lasersystem: Einzelschneidenkontrolle	
G604	Lasersystem: Werkzeugbruchkontrolle	
G606	TT130: Kalibrierung	
G607	TT130: Länge vermessen	
G608	TT130: Radius vermessen	
G609	TT130: Werkzeug Länge und Radius vermessen	

26.4 Basis M-Funktionen

M..	Früh	Spät	Beschreibung	Modal mit:
M0 **M1** **M30**	 X 	X X	Programm-Halt Wahlweiser Halt Programmende.	- - -
M3 **M4** **M5** **M19**	X X 	 X X	Spindel EIN Rechtslauf Spindel EIN Linkslauf Spindel STOP Spindel STOP in bestimmter Winkellage.	M4,M5,M14,M19 M3,M5,M13,M19 M3,M4,M13,M14 M3,M4,M13,M14
M6 **M66**	X X		Automatischer Werkzeugwechsel ausführen Manueller Werkzeugwechsel	- -
M7 **M8** **M9**	X X 	 X	Kühlmittel Nr. 2 einschalten Kühlmittel Nr. 1 einschalten Kühlmittel ausschalten	M9 M9 M7,M8,M13,M14
M13 **M14**	X X		Spindel EIN, Rechtslauf und Kühlmittel EIN Spindel EIN, Linkslauf und Kühlmittel EIN	M9 M9
M25 **M26** **M27** **M28** **M24** **M29**	X X X X		Zum Aktivieren der WKZ-Messung Meßtaster kalibrieren Meßtaster aktivieren Meßtaster ausschalten Tastsystem aktivieren Blasluft beim Meßtaster einschalten	- - M28 M27
M41 **M42** **M43** **M44**	X X X X		Auswahl Getriebestufe Spindelantrieb.	M42,M43,M44 M41,M43,M44 M41,M42,M44 M41,M42,M43
M67	X		Werkzeugkorrektur aktivieren	-

26.5 Maschinenabhängige M-Funktionen

M..	Früh	Spät	Beschreibung	Modal mit:
M10 **M11** **M22** **M23** **M32** **M33**	 x x x	x x x 	Klemmung der 4.Achse ZU AUF Klemmung der 5.Achse ZU AUF Klemmung der 6.Achse ZU AUF	
M16 **M18**	 x	x 	Werkstückreinigung AUS Werkstückreinigung EIN	
M20	x		Freibelegbarer NC-Ausgang	
M46	x		Automatischer Werkzeugwechsel (ohne Rückzug der nicht am Werkzeugwechsel beteiligten Achsen)	
M53/M54	x		Schwenkfräskopf für horizontale/vertikale Bearbeitung	
M55	x		Gesteuerten NC-Fräskopf in 0-Gradstellung richten und fixieren	
M56 **M57** **M58**			1. Fahrbereich (Einschaltstellung) für X-Achse freigeben (Modal) 2. Fahrbereich für X-Achse freigeben (Modal) 3. Fahrbereich für X-Achse freigeben (Modal)	
M60/M61 /M62	-		Palettenwechsel-Befehle	
M68			Werkzeugmagazin im Arbeitsraum beladen/entladen	
M70 **M71**	x 	 x	Späneförderer EIN Späneförderer AUS	
 M74 **M75** **M76** **M77**	 - - - -		Rettungsfunktionen: Paletten-Rundspeicher Palettenwechsler Schwenkfräskopf Werkzeugwechsler	
M80-M89	-		Reserviert für Software-Option	

27. Technological commands

27.1 Feed rate

Feed rate F.. [mm/min | inch/min]
N.. F100

Constant feed rate:

F1=0 Feed rate relative to equidistant. (Starting position)
N.. F.. F1=0

F1=1 Feed rate relative to workpiece contour. The feed is reduced in the case of inside radii.
N.. F.. F1=1

F1=2 Feed rate relative to workpiece contour. The feed is reduced in the case of inside radii and increased for outside radii.
N.. F.. F1=2

F1=3 Feed rate relative to workpiece contour. The feed is increased in the case of outside radii.
N.. F.. F1=3

F2=... Retract feed at G85, infeed at G86/G89, G201 or measuring feed at G145.

F3=... Feed for (negative) infeed movement (insertion).

F4=... Feed for plane movement.

F5=... Feed unit for rotating axes
F5=0 degrees/min (default)
F5=1 mm/min or inches/min

F6=... Local feed within a block

Tool axis:	axis perpendicular to plane of operation (G17, G18, ...).
radial milling direction:	milling in the plane of operation
axial milling direction:	milling in direction of tool axis (only in infeed direction)

Modal parameters F, F1=

27.2 Spindle speed

Spindle speed S.. [rpm]

S parameters are modal.

N.. S600

27.3 Tool number

Tool number T.. [Format 8.2] (255 tools max.)	N.. T1 M..
Original tool (T1-T99999999)	N.. T1
Replacement tool (Tx.01-Tx.99)	N.. T1.01
Activation:	
Automatic tool change	N.. T.. M6
Manual tool change	N.. T.. M66
Activate tool data	N.. T.. M67
First additional tool offset	N.. T.. T2=1 M6/M66/M67
Second additional tool offset	N.. T.. T2=2 M6/M66/M67
Tool life T3=..[0-9999,9min]	N.. T.. T3=x M6/M66
Cutting force control T1=..[1..99]	N.. T.. T1=x M6/M66
Deactivate (T1=0 or T1= not programmed)	N.. T1=0
Modal parameters T, T1=, T2=	

28. E Parameters and arithmetic functions

28.1 E parameters

Parameter E..	N.. E..
Format:	
Integer	E1=20
Fixed-point number	E1=200.105
Floating point number (exponent value: -99 - +99)	E1=1.905e5

Change unit of measurement G70 <--> G71:
All values are converted. In this case information such as spindle speed, feed rate, etc., should not be defined as parameters.

E parameters are modal.

Note

The address 'E' (parameter) must be entered into the program as an upper case character.

28.2 Arithmetic functions

Standard arithmetic functions (Blanks not permitted in functions)	E1=E2 E1=E2+E3 E1=E2-E3 E1=E2*E3 E1=E2:E3
Exponentiation	E1=E2^2 E1=(-3)^E3
Reciprocal values	E1=E2^-2(E1=1:E2^2)
Square root (value must be positive)	E1=sqrt(E2)
Absolute values	E1=abs(E2)
Integers	E1=int(E2)
Angle definition Format: Degree/Minutes/Seconds (cannot be entered directly) Input formats	44° 12' 33.5":
Decimal format	E1=44.209303
Angular conversion (gives an angle of)	E1=44+12:60+33.5:3600 E1=44.209303
Circle constants 'pi' or π (3.14)	E1=(E2*pi):2

Radian format	E1=44+12:60+33.5:3600 E2=((E1:360)*2*pi)rad
Trigonometric functions	sin(E..) cos(E..) tan(E..) asin(E..) acos(E..) atan(E..)
Comparison functions (Condition satisfied --> E..=1) (Condition not satisfied --> E..=0)	E1=E2=E3 --> E1=1 E1=E2<>E3 --> E1=1 E1=E2>E3 --> E1=1 E1=E2>=E3 --> E1=1 E1=E2<E3 --> E1=1 E1=E2<=E3 --> E1=1

Evaluation priority of arithmetic expressions and comparison functions

1. sin, cos, tan, asin, acos, atan, sqrt, abs, int
2. Exponentiation (^), reciprocal values (^-1)
3. Multiplication (*), division (:)
4. Addition (+), subtraction (-)
5. Relational expressions (=, <>, >, >=, <, <=)

If a block contains operations of the same priority, they are executed in sequence from the start of the block to the end.

28.3 Expanded calculation operations

28.3.1 E parameters

Format:

Arc sine E1=asin(E2,E3)
Arc cosine E1=acos(E2,E3)
Arc tangent E1=atan(E2,E3)
Whole number conversion with large value E1=ceil(E2)
Whole number conversion with small value E1=floor(E2)
Rounding E1=round(E2,n) (n is no. of decimal places)
Remainder of division E1=mod(E2,E3)
Sign E1=sign(E2)

Remark: The integer function is changed with the floor function in V420 and higher.

28.3.2 Whole numbers

When using the integer function, the numerical value is rounded, i.e. all figures after the decimal point are ignored.
E1=int(E2)

Example: E2=8.9 results in 8, E2=-8.9 results in –8

28.3.3 Whole numbers with largest value

When using the integer function with the largest value, the numerical value is rounded according to the largest argument.
E1=ceil(E2)

Example: E2=8.9 results in 9, E2=-8.9 results in –8

28.3.4 Whole numbers with smallest value

When using the integer function with the smallest value, the numerical value is rounded according to the smallest argument.
E1=floor(E2)

Example: E2=8.9 results in 8, E2=-8.9 results in –9

28.3.5 Rounding

When the rounding function is used, the numerical value is rounded according to the number of decimal places.
E1=round(E2,n) (n is number of decimal places)
Remark: If the number of decimal places is not entered, zero is assumed.

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

28.3.6 Remainder of division

When the remainder function is used, the remainder is returned by the argument.
E1 =mod(E2,E3)
Remarks:
-E1=E2-int(E2:E3)*E3
- If E3 is 0, E2 is returned.
- If E3 is not entered, 1 is assumed.
- The sign is the same as the sign of E1.

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

28.3.7 Sign

When the sign function is used, the sign is returned.
E1 =sign(E2)

Example: E2=8.9 results in 1, E2=0 results in 0, E2=-8.9 results in -1
Also possible (V429 and higher):
E1=asin(E3,E4) E1=acos(E3,E4) E1=atan(E3,E4) where E2=E3:E4
Remark: - abs(E2) must be less than or equal to 1 for acos and asin.
- the angle created lies between 0° and +360°

28.3.8 Variable parameter no.:

E(value or expression)=<value or expression>

Examples:
E(1)=
E(1.2e1)
E(E1)=
E(E1+E2)=
E(sin(45)*100)=

29. Miscellaneous

29.1 Operator machine constants

Refer to the documentation provided by the machine builder.

For user

For maintenance/customer service personnel only

29.2 Machine settings monitoring file

In Edit-MC the machine settings which also exist in the monitoring file are displayed with a lock indicator. These machine settings then cannot be edited.
Release for editing purposes is achieved by means of a password.

Machine settings which exist in the monitoring file can only be overwritten if the password has been entered. In this way, the unintentional changing of machine settings is precluded.

Note

The machine settings 250 to 316 inclusive are used for selection of the available options.

29.2.1 List of user machine constants

20 Axes orientation (0=0,1=-90,2=180,3=90) O
21 Spindle power display (0=off, 1=on) O
22 Display G181 (0=fictive, 1=real) O
24 Screensaver time out (0=off,1-255[min]) O
80 Selection demo mode (0=off,1=on,2=IPLC) O
93 BTR memory size (4-1024)[kB] O
251 Technology entry (0=off, >0 = on) O
252 DNC remote function (0=off, >0 =on) O
254 Tool measurement entry (0=off,1=on) O
255 Int.act.contour prog. (0=off, >0 = on) O
262 BTR function (0=off, >0 = on) O
262 BTR function (0=off, >0 = on) O
263 3D tool function (0=off,1=on) O
264 Cylinder interpolator (0=off,1=on) O
265 G6 (spline) function (0=off,1=on) O
266 Universal pocket cycle (0=off, >0 = on) O
271 Erase graphics (0=off, >0 = on) O
272 Synchrone graphics (0=off, >0 = on) O
292 Memory MEX 1 (0=off,??????=on) O
293 Memory MEX 2 (0=off,??????=on) O
294 Memory MEX 3 (0=off,??????=on) O
295 Memory MEX 4 (0=off,??????=on) O
296 Memory MEX 5 (0=off,??????=on) O
297 Memory MEX 6 (0=off,??????=on) O
350 Probe position 1st axis negative [µm] O
351 Probe position 1st axis positive [µm] O
352 Probe position 2nd axis negative [µm] O
353 Probe position 2nd axis positive [µm] O
354 Probe position 3rd axis negative [µm] O
355 Probe position 3rd axis positive [µm] O

714 Scaling mode (0+2=factor,1+3=%,2+3=3D) O
715 Decimal point scaling (0-6) O
772 DIO: line syntax check (0=off,1=on) O
773 DIO: block numbers > 9000 (0=off,1=on) O
774 Tool in (0,1=clear,2=protect,3=replace) O
782 DNC remote directory (0=no, 1=yes) O
783 DNC disk format function (0=no, 1=yes) O
792 IPC remote directory (0=no, 1=yes) O
793 IPC disk format function (0=no, 1=yes) O
795 IPC %-protocol in file (0=no, 1=yes) O
799 MPC %-protocol in file (0=no, 1=yes) O
847 Width fixed measuring probe [µm] O
848 Radius calibration ring [µm] O
901 Dev1: baudrate (110-57600) O
903 Dev1: number of stopbits (1 or 2) O
904 Dev1: leader/trailer length (0-120) O
905 Dev1:data carrier (0=ASCII,1=ISO,2=EIA) O
906 Dev1:auto code recognition (0=off 1=on) O
907 Dev1: flowcontrol (0=RTS,1=RTS-f,2=XON) O
908 Dev1: check DTR (0=no, 1=yes) O
911 Dev2: baudrate (110-57600) O
913 Dev2: number of stopbits (1 or 2) O
914 Dev2: leader/trailer length (0-120) O
915 Dev2:data carrier (0=ASCII,1=ISO,2=EIA) O
916 Dev2:auto code recognition (0=off 1=on) O
917 Dev2: flowcontrol (0=RTS,1=RTS-f,2=XON) O
918 Dev2: check DTR (0=no, 1=yes) O
921 Dev3: baudrate (110-57600) O
923 Dev3: number of stopbits (1 or 2) O
924 Dev3: leader/trailer length (0-120) O
925 Dev3:data carrier (0=ASCII,1=ISO,2=EIA) O
926 Dev3:auto code recognition (0=off 1=on) O
927 Dev3: flowcontrol (0=RTS,1=RTS-f,2=XON) O
928 Dev3: check DTR (0=no, 1=yes) O
931 LSV/2 baudrate (110-57600) O
932 LSV/2 characterset (0=ASCII,1=ISO) O
933 LSV/2 time out period (0-128)[s O
934 LSV/2 nr.of repeats (0=no limit,1-12) O
935 LSV/2 delay time (0-128)[ms] O
936 LSV/2 check DTR (0=no, 1=yes) O

2455 Position fixed measuring probe 1 O
2456 Position fixed measuring probe 2 O
2457 Position calibration ring O
2655 Position fixed measuring probe 1 O
2656 Position fixed measuring probe 2 O
2657 Position calibration ring O
2855 Position fixed measuring probe 1 O
2856 Position fixed measuring probe 2 O
2857 Position calibration ring O
2955 Position fixed measuring probe 1 O
2956 Position fixed measuring probe 2 O
2957 Position calibration ring O
3055 Position fixed measuring probe 1 O
3056 Position fixed measuring probe 2 O
3057 Position calibration ring O
3155 Position fixed measuring probe 1 O
3156 Position fixed measuring probe 2 O
3157 Position calibration ring O
3255 Position fixed measuring probe 1 O
3256 Position fixed measuring probe 2 O
3257 Position calibration ring O
3355 Position fixed measuring probe 1 O
3356 Position fixed measuring probe 2 O
3357 Position calibration ring O
3455 Position fixed measuring probe 1 O
3456 Position fixed measuring probe 2 O
3457 Position calibration ring O
3555 Position fixed measuring probe 1 O
3556 Position fixed measuring probe 2 O
3557 Position calibration ring O
3655 Position fixed measuring probe 1 O
3656 Position fixed measuring probe 2 O
3657 Position calibration ring O
3755 Position fixed measuring probe 1 O
3756 Position fixed measuring probe 2 O
3757 Position calibration ring O
3855 Position fixed measuring probe 1 O
3856 Position fixed measuring probe 2 O
3857 Position calibration ring O
3955 Position fixed measuring probe 1 O
3956 Position fixed measuring probe 2 O
3957 Position calibration ring O
4055 Position fixed measuring probe 1 O
4056 Position fixed measuring probe 2 O
4057 Position calibration ring O
4155 Position fixed measuring probe 1 O
4156 Position fixed measuring probe 2 O
4157 Position calibration ring O
4255 Position fixed measuring probe 1 O
4256 Position fixed measuring probe 2 O
4257 Position calibration ring O

29.3 Connecting cable for data interfaces

Client must ensure that an external interface cable is being used which is shielded on either side.

If a T-switch is being used, the signal ground and shield must not be connected. Mechanical switch-over is only permitted to signal lines.

Should any problems be encountered with the data interface, check for the following:
Is a shielded data cable being used?
Does the length of the data line not exceed 15 metres?
Is the machine connected to the machine socket?

29.4 Configuring the Ethernet interface

Note

The MillPlus should be configured by a network specialist.

The MillPlus is fitted with an Ethernet interface to allow the control to be integrated into your network as a client. The MillPlus transfers data across the Ethernet interface using the TCP/IP protocol (Transmission Control Protocol/Internet Protocol) and the NFS (Network File System). TCP/IP and NFS are widespread in UNIX systems, so you should normally be able to integrate MillPlus into the UNIX world without having to use additional software.
The PC world with its Microsoft operating systems also uses TCP/IP for networking, but not NFS. You will therefore need to install some additional software to enable MillPlus to be integrated into a PC network.
NFS Client in the CNC has been tested with the following network software:

Operating system	Network software
Windows NT 4.0	Diskshare NFS server for Windows NT, version 03.02.00.07 (Intergraph, web site: www.intergraph.com).
	Maestro NFS server for Windows NT, version 6.10 (Hummingbird Communications, web site: http:\\www.hummingbird.com). e-mail: support@hummingbird.com
Windows 95	Solstice NFS server, a component from the Solstice Network Client for Windows package, version 3.1 (Sun Microsystems, web site: www.sun.com).
Windows 95/98, NT4.0	Omni-NFS server, (Xlink Technologies Inc., web site: http:\\www.xlink.com).
	CimcoNFS server, (CIMCO Integration, web site: http:\\www.cimco.dk).

29.4.1 Ethernet interface connection options

You can connect the MillPlus Ethernet interface to your network using the RJ45 connector (10BaseT). The connector is galvanically isolated from the control electronics.

RJ45 connector (10BaseT)
Use twisted-pair cables to connect the MillPlus to your network if using the 10BaseT connector.
If using screened cables, the maximum cable run between MillPlus and a node is 400m.

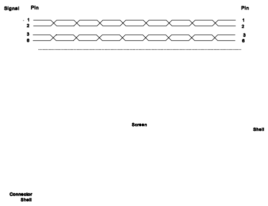
Note

If you connect the MillPlus directly to a PC, crossover cables must be used.

29.4.2 Connecting cable for Ethernet interface

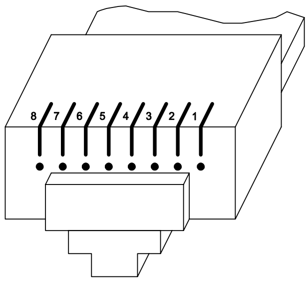
Ethernet interface RJ45 socket

Maximum screened cable run	:400 m
Maximum transmission rate	:200 kBaud to 1 MBaud



Pin	Signal description
1	TX+ Transmit Data
2	TX– Transmit Data
3	REC+ Receive Data
4	spare –
5	spare –
6	REC– Receive Data
7	spare –
8	spare –

Front view of connector



The interface complies with the safe mains isolation requirements of IEC 742 EN 50 178.

29.4.3 Configure MillPlus Ethernet interface (file tcpip.cfg)

Note

The MillPlus should be configured by a network specialist.

Setting up machine constants:

Mc311=0	DNC Plus	(0=off, on=??????)
Mc313=Password	NFS Server	(0=off, on=??????)
??????=Password		

The data connection can be configured using the tcpip.cfg file. The tcpip.cfg file must always be on the C:\ hard disk. A maximum of one local, four hardware, one service, ten NFS servers and ten DNC servers can be defined and managed. The language is always English.

The tcpip.cfg file can be modified from the "HEIDENHAIN NUMERIC Service Menu". The Service menu can be called up while the CNC system is being initialised by pressing the S key on the ASCII keyboard. Select the tcpip.cfg editor using "TCP/IP configuration". A line should have no more than 128 characters. No distinction is made between upper and lower case characters. A comment line is indicated by a semicolon ‘;’. Configuration sections can be repeated. A section is defined by a name in square brackets. ‘[Name]’

Hardware section

This is indicated by the section name [Hardware] and contains the network device parameters. The configuration file may have a number of hardware sections containing settings for several network devices. The 'local' section determines which network device is to be used.

Parameter		Meaning
Type	= <device name>	Name of the network device, e.g. SMC, NE2000, i8255x or AT-lantic
i0	= <irq number>	The parameters i0 to i3 control the allocation of the four interrupt outputs of the network device to the IRQ lines of the CPU. This is determined by the CNC hardware. See "Sample tcpip.cfg file".
i1	= <irq number>	
i2	= <irq number>	
i3	= <irq number>	
Irq	= <irq number>	Defines which IRQ the driver software uses. This number must be one of the numbers defined through i0 to i3.
Iobase	= <iobase address>	Setting of the I/O base address of the network device.

Local section

[local] contains the local parameters for the TCP/IP data communications protocol. There may only be one local section.

Parameter		Meaning
Type	= <device name>	Defines the type of network device in the CNC. The name must correspond to the device name specified in one of hardware sections under Type_Parameter.
Connector	= 10baseT | 10base2	Defines which connection is to be used, 10BaseT (RJ45) or 10Base2 (BNC).
HostName	= < network name>	The name that the MillPlus uses to log on to the network. Network name: must contain no more than 17 letters. If you do not enter a name, the MillPlus will use the Null authentication and not the normal Unix authentication; the parameters UserId, GroupID, DirCreateMode and FileCreateMode will be ignored.
IpAddress	= <IP address>	Address that your network administrator must assign for the MillPlus. Input: four integer values separated by decimal points (0 to 255). Ask your network administrator for the number, e.g. 192.168.0.17
SubnetMask	= <IP adress mask>	The subnet mask used to save on addresses within your network. This defines how many bits of the 32 bit Internet address are to be used for the Subnet-ID and how many for the station ident number e.g. 255.255.255.0 defines 24 bits for the subnet number and 8 bits for the station ident number. Again, ask your network administrator what value to enter.
DefaultRouter	= < Router addr>	Internet address of your default router. Only to be specified if your neetwork consists of various subnets. Input: four integer values separated by decimal points (0 to 255). Ask your network administrator for the number. Enter 0.0.0.0 if your network does not have a router.
Protocol	= rfc | ieee	Definition of the transmission protocol. rfc: Ethernet protocol according to RFC 894 ieee: IEEE 802.2/802.3 protocol according to RFC 1042 Default value is 'rfc'.
Timezone	= <time zone>	The time parameter of files addressed by NFS. Shown in UTC (Universal Time Coding), commonly known as GMT (Greenwich Mean Time). The Timezone parameter indicates the difference between local time and UTC, e.g. in Frankfurt the local time is UTC+1 (hours), in other words Timezone = -1. Default value is -1.
DncPort	= <port number>	Defines the port number for the DNC service in both the Mill Plus CNC and the DNC service of a remote system. Default port number = 19000
SummerTime	= y | n	The SummerTime parameter determines whether the system is to switch automatically from summer to winter time and vice versa. Default value is y.

NFS server section

[NFS server] marks the start of the NFS server section. This section contains the remote parameter values for the NFS server. The configuration file may have a number of remote sections containing settings for several NFS servers.

Parameter	Meaning
IpAddress = <IP address>	Defines the IP address of your server. Input: four integer values separated by decimal points. Ask your network administrator for the number, e.g. 192.168.0.1
DeviceName = <server name>	Name of the NFS server as shown in the MillPlus file administration, e.g. Server_NT1.
RootPath = <Path name>	Directory on the NFS server to which you want to link the MillPlus. The MillPlus is only able to access this directory and its sub-directories. Watch out for upper/lower case when typing the path name.
TimeOut = <Timeout in ms>	Time in ms that the MillPlus allows to elapse before repeating a Remote Procedure Call that the server did not respond to. Input range: 0 to 100 000. The default value '0' corresponds to a timeout of 700 ms. You should only use higher values if the MillPlus has to communicate with the Server via a number of routers, e.g. for Intergraph and Hummingbird Servers, 1000 ms is sufficient; for Sun's Solstice Server, 5000 ms is necessary. Ask your network administrator for the value.
rwtimeOut = 30	Timeout before retrying a read/write operation on NFS files (the time is doubled on each retry of the same record until the timeout value is reached).
ReadSize = <packet size>	Packet size in bytes when receiving data. Input range: 512 to 4096. Input 0: MillPlus uses the optimum packet size as reported by the server. Default value is 1300.
WriteSize = <packet size>	Packet size in bytes when sending data. Input range: 512 to 4096. Input 0: MillPlus uses the optimum packet size as reported by the server. Default value is 1300
HardMount = y | n	Specifies whether the MillPlus should repeat the Remote Procedure Call until the NFS server responds. y: always repeat n: do not repeat Do not use y if there is no active server on the network.
AutoMount = y | n	Specifies whether MillPlus should be automatically mounted on the network when it is switched on. y: automount n: no automount
UseUnixId = y | n	Use 'Unix style' authentication for NFS. y: Unix authentication, uses Userid, GroupId, DirCreateMode and FileCreateMode n: no authentication. Userid, GroupId, DirCreateMode and FileCreateMode are not used. Default value is y.
UserId = <user Id>	User identification (Unix style): used by NFS to identify the user (the CNC) on the server, e.g. 100. Ask your network administrator for the value.
GroupId = <group Id>	Specifies which group_identification (Unix style) you use to access files on the network, e.g. 100. Ask your network administrator for the value.
DirCreateMode = <mode>	Specifies the access rights to directories on the NFS server. Enter value in binary format. Example: 111101000 0: access not permitted 1: access permitted Default value is 0777 (octal).

CaseSensitive	= y | n	Uses or ignores the difference between capitals and small letters when comparing directory or file names during directory searching. Defaults to ‘y’. y: Case sensitive searches. E.g. 1234.pm is different from 1234.PM n: Not case sensitive searches. E.g. 1234.pm is equal to 1234.PM
DncPort	= <port number>	Defines the port number for the DNC service in both the Mill Plus CNC and the DNC service of a remote system. Default port number = 19000
FileCreateMode	= <mode>	Specifies the access rights to files on the NFS server. Enter value in binary format. Example: 111101000 0: access not permitted 1: access permitted Default value is 0777 (octal).

```
1 1 1 1 0 1 0 0 0          = 0750 (Oktalzahl)
| | | | | | | | |
| | | | | | | | └──────────── Alle anderen Benutzer: Suchen
| | | | | | | └────────────── Alle anderen Benutzer: Schreiben
| | | | | | └──────────────── Alle anderen Benutzer: Lesen
| | | | | └────────────────── Arbeitsgruppe:    Suchen
| | | | └──────────────────── Arbeitsgruppe:    Schreiben
| | | └────────────────────── Arbeitsgruppe:    Lesen
| | └──────────────────────── Benutzer:         Suchen
| └────────────────────────── Benutzer:         Schreiben
└──────────────────────────── Benutzer:         Lesen
```

DncServer

[DncServer] indicates a DNC remote server section. It contains the parameter settings for a remote DNC server. One or more DNC remote server sections can be present in the configuration file to define one or more DNC servers. The remote section contains the following parameters:

Parameter		Meaning
IpAddress	= <IP address>	Defines the IP address of your server. Input: four integer values separated by decimal points. Ask your network administrator for the address, e.g. 192.168.0.1
DeviceName	= <server name>	Name of the DNC server as shown in the MillPlus file management, e.g. DMG_Service_1.
TimeOut	= <Timeout in sec.>	Defines the connection timeout in s for the connection between local DNC client and remote DNC server. When the remote DNC server is on the local network set TimeOut to zero. Use non-zero values when the remote DNC server is reached through an external connection such as an ISDN router.

Service

[Service] indicates a DNC remote server section. It contains the parameter settings for a remote DNC server. One or more DNC remote server sections can be present in the configuration file to define one or more DNC servers. The remote section contains the following parameters:

Parameter	Meaning
IpAddress = <IP address>	Specifies the IP address of your server. Input: four integer values separated by decimal points. Ask your network administrator for the address, e.g. 192.168.254.3
serverName = <server name>	Name of the DNC server as shown in the MillPlus file management, e.g. DMG_Service_1.
port = <port number>	Default = 19001
repeatTime = <Time in sec.>	Default = 10 Sec.
idleTimeout = <Time in min.>	Default = 15 Min.
request = @<File name> or <Ascii string>	e.g. @c:\OEM\request.txt.

Sample tcpip.cfg file

```
; TCP/IP configuration file
; More sections of [remote] are allowed   --> more NFS servers to choose
; More sections of [hardware] are allowed --> actually used hw is defined in [local] section
; The keywords with an ';' placed in front can be omitted. The value shown is the default
; value
;
;[hardware]                    ; LE412 HARDWARE
;type      = smc               ; this hw is an smc network device
;irq       = 9                 ; irq used by network device driver
;i0        = 9                 ; hardware connections of network device to irq's
;i1        = 3
;i2        = 10
;i3        = 11
;iobase    = 0x300             ; io base address of network device
;
;[hardware]                    ; LE422 HARDWARE
;type      = i8255x            ; this hw is an i8255x network device
;irq       = 10                ; irq used by network device driver
;iobase    = 0xE400            ; io base address of network device
;
[hardware]                     ; VMEBUS HARDWARE
type      = at-lantic          ; this hw is a ne2000 compatible network device
                               ; note: the VMEbus at/lantic is used in ne2000 compatible mode
irq       = 5                  ; irq used by network device driver
i0        = 3                  ; hardware connections of network device to irq's
i1        = 5
i2        = 9
i3        = 15
iobase    = 0x300 0x240        ; io base address of network device
;
[hardware]                     ; dos_shape_pc

type      = ne2000             ; this hw is a ne2000 compatible network device
                               ; note: the VMEbus at/lantic is used in ne2000 compatible mode
irq       = 5                  ; irq used by network device driver
iobase    = 0x300              ; io base address of network device
;
[local]                        ; configuration of CNC
type      = ne2000             ; the type of network device used:
```

```
                                        ; must match a [hardware] type
connector       = 10base2               ; 10baseT: RJ45 (twisted pair), 10base2: bnc (coax)
hostName        = MillPlusshape         ; CNC network name, maximum of 17 characters
ipAddress       = 170.4.100.16  ; internet address of the CNC ==> ask your network
subnetMask      = 255.255.0.0  ;subnet mask of network      ==> administrator for values
defaultRouter   = 0.0.0.0          ;internet address of default router, 0.0.0.0: no router
                          ;                   ==> ask your network
                          ;                      administrator for value
;protocol       = rfc          ; Link layer protocol used rfc: Ethernet, ieee: IEEE 802
;timezone       = -1           ; + 1 hour of gmt :gmt + tz == local-> gmt=local - tz!!
;summerTime     = y            ; use automatic summertime correction (daylight saving)
port            = 19000        ; portnumber DNC service
;
[nfsServer]                   ; configuration of a remote server.
                          ; more than one remote sections allowed
ipAddress       = 170.4.100.140 ; internet address of the server ==> ask your network
                          ;                        administrator for value
deviceName      = Intergraph    ; Server name used inside CNC
rootPath        = c:\temp       ; server directory to be mounted as network drive on CNC
                          ; This must be a shared directory on the NFS server
timeOut         = 50000        ; units in milliseconds for timeout in server connection
                          ; 0..100 000, 0: timeout set to 700 ms
;rwtimeOut      = 30           ; timeout used for retry at read/write of NFS-files
                          ; (time is doubled for each retry of same packet until timeOut)
;readSize       = 1300         ; packet size for data reception: 512 to 4096, or 0 = use
                          ; server reported packet size
;writeSize      = 1300         ; packet size for data transmission
;hardMount      = n            ; yes/no continue mouting until succesfull
                          ; don't use 'y' if you're uncertain server is running
autoMount       = n            ; yes/no automatically mount when CNC initialises
;useUnixId      = y            ; use UserId/groupId to identify to the server
userId          = 100          ; Unix style user id for Authentication ==> ask your network
groupId         = 100          ; Unix style group id               ==> administrator
;dirCreateMode   = 0777          ; Unix style access right for dir-create: Octal number
;fileCreateMode  = 0777          ; Unix style access rights for file-create: Octal number
;
[nfsServer]                   ; configuration of a remote server.
                          ; more than one remote sections allowed
ipAddress       = 170.4.100.171 ; internet address of the server ==> ask your network
                          ;                        administrator for value
deviceName      = Hummingbird   ; Server name used inside CNC
rootPath        = c:\NFS_DATA   ; server directory to be mounted as network drive on CNC
                          ; This must be a shared directory on the NFS server
timeOut         = 1000         ; units in milliseconds for timeout in server connection
                          ; 0..100 000, 0: timeout set to 700 ms
;rwtimeOut      = 30           ; timeout used for retry at read/write of NFS-files
                          ; (time is doubled for each retry of same packet until timeOut)
;readSize       = 1300         ; packet size for data reception: 512 to 4096, or 0 = use
                          ; server reported packet size
;writeSize      = 1300         ; packet size for data transmission
;hardMount      = n            ; yes/no continue mouting until succesfull
                          ; don't use 'y' if you're uncertain server is running
autoMount       = n            ; yes/no automatically mount when CNC initialises
;useUnixId      = y            ; use UserId/groupId to identify to the server
userId          = 100          ; Unix style user id for Authentication ==> ask your network
groupId         = 100          ; Unix style group id               ==> administrator
;dirCreateMode   = 0777          ; Unix style access right for dir-create: Octal number
;fileCreateMode  = 0777          ; Unix style access rights for file-create: Octal number
;
;
[NFSserver]                   ; configuration of a remote server.
                          ; more than one remote sections allowed
ipAddress       = 170.4.100.194 ; internet address of the server ==> ask your network
                          ;                        administrator for value
deviceName      = Solstice      ; Server name used inside CNC
rootPath        = C:\solstice   ; server directory to be mounted as network drive on CNC
                          ; This must be a shared directory on the NFS server
timeOut         = 6000         ; units in milliseconds for timeout in server connection
                          ; 0..100 000, 0: timeout set to 700 ms
rwtimeOut       = 600          ; timeout used for retry at read/write of NFS-files
                          ; (time is doubled for each retry of same packet until timeOut)
;readSize       = 1300         ; packet size for data reception: 512 to 4096, or 0 = use
                          ; server reported packet size
;writeSize      = 1300         ; packet size for data transmission
```

```
;hardMount       = n            ; yes/no continue mouting until succesfull
                                ; don't use 'y' if you're uncertain server is running
autoMount        = n            ; yes/no automatically mount when CNC initialises
;useUnixId       = y            ; use UserId/groupId to identify to the server
userId           = 100          ; Unix style user id for Authentication ==> ask your network
groupId          = 100          ; Unix style group id              ==> administrator
;dirCreateMode   = 0777         ; Unix style access right for dir-create: Octal number
;fileCreateMode  = 0777         ; Unix style access rights for file-create: Octal number
;
[NFSserver]                     ; configuration of a remote server.
                                ; more than one remote sections allowed
ipAddress        = 170.4.100.143 ; internet address of the server ==> ask your network
                                ;                       administrator for value
deviceName       = pmeSolstice  ; Server name used inside CNC
rootPath         = d:\solstice  ; server directory to be mounted as network drive on CNC
                                ; This must be a shared directory on the NFS server
timeOut          = 5000         ; units in milliseconds for timeout in server connection
                                ; 0..100 000, 0: timeout set to 700 ms
rwtimeOut        = 100          ; timeout used for retry at read/write of NFS-files
                                ; (time is doubled for each retry of same packet until timeOut)
;readSize        = 1300         ; packet size for data reception: 512 to 4096, or 0 = use
                                ; server reported packet size
;writeSize       = 1300         ; packet size for data transmission
;hardMount       = n            ; yes/no continue mouting until succesfull
                                ; don't use 'y' if you're uncertain server is running
autoMount        = n            ; yes/no automatically mount when CNC initialises
;useUnixId       = y            ; use UserId/groupId to identify to the server
userId           = 100          ; Unix style user id for Authentication ==> ask your network
groupId          = 100          ; Unix style group id              ==> administrator
;dirCreateMode   = 0777         ; Unix style access right for dir-create: Octal number
;fileCreateMode  = 0777         ; Unix style access rights for file-create: Octal number
;
[dncServer]
serverName       = Teleservice  ; alias name for this server (PME-pc)
ipAddress        = 170.4.100.143 ; its ip address
;timeOut         = 1000         ; timeout in connection
;port            = 19000        ; port number for dnc services

[Service]                       ; (MAHO) service centre
serverName       = "Maho Service"; alias name for this service
ipAddress        = 170.4.100.140 ; its ip address
request          = "here I am"  ; @fileName/tekst to identify yourself
;IdleTimeOut     = 15           ; disconnect after .. minutes
;port            = 19001        ; port number for service
;repeatTime      = 10           ; repeat time in seconds to connect
;
; end of file
```

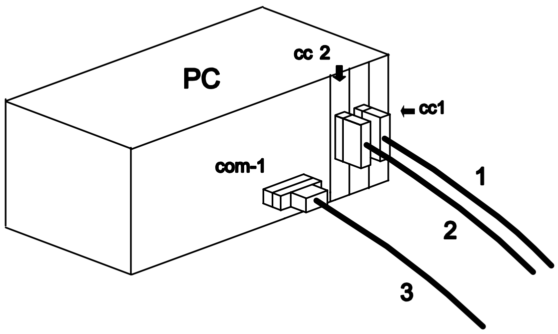
29.5 Digitizing

The purpose of digitization is to make the machining program for complex products in a relatively simple fashion.

To this end, a measuring probe scans an area to be defined in which the product to be reproduced is located. This measuring probe is linked to an external PC and transmits the coordinates of all contact points with the product to the PC. The Renishaw Trace software package is run on the PC, which shows every contact point on the screen, thus scanning the shape.
The second digitization phase is making the machining program. The technological information required for machining is added to the scanned shape. The pattern is then transformed into a control program by a machine-dependent postprocessor.

29.5.1 Installation

The basic hardware configuration for the Trace program is shown below.



1 PL79 cable to measuring probe
2 PL84 cable to measurement system
3 Cable from PC to CNC (refer to Installation Manual V320 Part 1,
Chapter "Connection information" cable 11a or 11b)

The machine should also be set to the Digitizing mode. For this purpose, set the CNC to Device 3, which enables you to use the Xon/Xoff protocol. The following machine constant settings are also required:

-MC10 (Number of Axes)	= 3
-MC303 (Digitizing mode; 0=off, ???????=on)	= ???????
-MC775 (Pipelined Digitizing; 0=off, 1=on)	= 1
-MC920 (Channel; 0=none, 1=RS232C, 2=RS422)	= 1
-MC921 (Baudrate; 110-38400)	= 38400
-MC923 (Number of stopbits; 1 or 2)	= 2
-MC924 (Leader/Trailer length; 0-120)	= 120
-MC925 (Data Carrier; 0=ASCII, 1=ISO, 2=EIA)	= 0
-MC926 (Auto Code Recognition; 0=off, 1=on)	= 1
-MC927 (Flowcontrol; 0=RTS, 1=RTS-F, 2=Xon)	= 2
-MC928 (Check DTR; 0=no, 1=yes)	= 0

If an older Trace version is used, set MC921 to 19200.

To enable communication between CNC and PC, the CNC should be in the Digitizing mode.

A holder should be inserted into the spindle before the spindle can be fitted with the measuring probe. The measuring probe is then mounted in the holder. We recommend that you should not mount the measuring probe entirely in the holder, but to retain some clearance. Refer to the Trace instructions for more details of the mounting procedure.

As a next step, the three links between PC and CNC should be made. Two of these links go from CNC to PC. For this reason, two cards (CC1 and CC2) have to be inserted into the PC. The procedure for introducing these cards is set out in the Trace instructions.

The first link goes from the CNC to the CC1 card of the PC via the PL79 cable. As soon as the measuring probe reaches a contact point with the product, the measuring probe transmits a signal to this card.

The second link goes from the machine to the CC2 card via the PL84 cable. As soon as the CC1 card is informed of a contact point being reached, the CC2 card reads out the relevant coordinates. The standard measures (rulers) of the X, Y and Z axes are the input to this link. The method of linking to the rulers depends on the specific machine.

The third link goes from PC to CNC. Being a serial link, it is led via the COM1 interface. Where appropriate, the COM2 connection may also be used; the relevant software settings have to be made for this purpose. If a serial dongle is used, it must be placed between the serial cable and the COM interface. The PC transmits commands to the CNC via this link, for instance the command to proceed when a contact point has been reached.

Note

Refer to the Renishaw Trace documentation and the chapter on activate/execute Program.