MANUAL

D - 22 / 23

(VERSION 3.02 / 08.01.97)

CE

EN 50082-1 EN 50082-2
VDE 0843-2 VDE 0843-3
EVDE 0843-4 EVDE 0843-5
IEC 801-1 bis IEC 801-5

Dipl. - Ing. ENGELHARDT GmbH
Heinrich-Hertz-Str. 9
76646 Bruchsal
Tel.: 07251 / 7218-0
Fax.: 07251 / 7218-99

Our most actual control manual can be downloaded from internet under
http//www.engelhardtgmbh.de/c88.pdf

email: mail@engelhardtgmbh.de **web: www.engelhardtgmbh.de**

Änderungsstand: 08/2003 W. Schäffner

1.1 LIST OF G-FUNCTIONS

G00 RAPID TRAVERSE

G01 LINEAR INTERPOLATION

G02 CIRCLE CW

G03 CIRCLE CCW

G04 DWELL TIME

G05 CIRCLE WITH RADIUS CW

G06 CIRCLE WITH RADIUS CCW

G09 SKIP REST OF TRAVEL

G10 CORNER ROUNDING

G11 ADDITIONAL FUNCTIONS F,S,T

G13 ADDITIONAL M-FUNCTIONS

G20 JUMP TO PROGRAM

G22 CALL PROGRAM

G23 CALL PROGR. WITH CONDITION

G33 THREAD G88 RELIEVE

G36 TOOL CHANGE

G40 RADIUS COORECTION OFF

G41 RADIUS CORRECTION LEFT

G42 RADIUS CORRECTION RIGHT

G53 DISPLACEMENT OFF

G54 DISPLACEMENT ON

G55 DISPLACEMENT II

G67/68 SOFTWARE LIMIT SWITCH -/+

G74 GO TO REFERENCE SWITCH

G75 SCALE FACTOR ON

G76 SCALE FACTOR OFF

G82 DEEP DRILLING

G83 CUT OFF

G84 TAPER CUTTING HORIZONTAL

G85 TAPER CUTTING VERTICAL

G86 CONTUR TURNING

G87 THREAD CYCLE

G88 RELIEVE GROOVE

G90 ABSOLUTE INPUT

G91 INCREMENTAL INPUT

G92 SET ACTUAL VALUE

G94 FEEDRATE IN MM/MIN

G95 FEEDRATE IN MICROM./ROT

G96 CONSTANT CUTTING SPEED

G97 CONSTANT SPINDLE SPEED

1.2 LIST OF M-FUNCTIONS

M00 PROGRAMMED STOP

M03 SPINDLE ON CW

M04 SPINDLE ON CCW

M05 SPINDLE STOP

M08 COOLING ON

M09 COOLING OFF

M10 CLAMPING ON

M11 CLAMPING OFF

M19 WAITING FOR AXES TO STOP

M21 BLOCK UPDATING OFF

M22 BLOCK UPDATING OFF IN PROGRAM CALLS

M23 FEEDRATE POTENTIOMETER OFF

M24 TESTRUN WITHOUT G04 AND M-FUNCTIONS

M25 TESTRUN WTH RAPID TRAVERSE

M26 KEYBOARD OFF

M27 WAIT FOR „IN POSITION“

M28 ACTUAL VALUE DISPLAY OFF

M1xx HANDLING OF I/O CARD 1

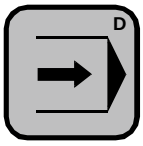
M2xx HANDLING OF I/O CARD 2

2. OPERATING MODES

After switching on, the CNC displays the menue of the operating modes. The following 9 modes can be selected:

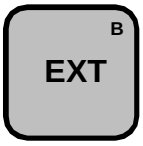
MANUAL MODE

TEACH IN



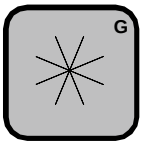
AUTOMATIC MODE

MANUAL INPUT (MDI)

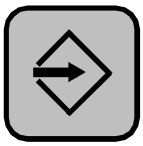


EXTERNAL DATA

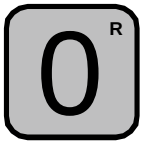
CLEAR MEMORY



CRAPHICS



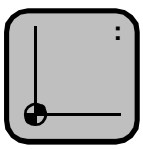
INPUT MODE



SWITCH OFF CNC

These operating modes can be selected by pressing the corresponding key.
Pushing the key [H] displays a list of implemented G-functions in the CNC.
In the MENU, each key can call a programm with the number P98XX. If for example the Programm P9801 is present in memory, it can be invoked by pushing the key [1].

Further functions key´s:

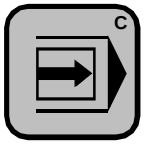


HOME POSITION

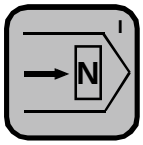
SPINDLE ON CW

SPINDLE ON CCW

RESET

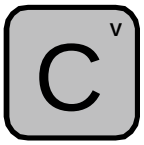


SINGLE STEPP



SEARCH BLOCK

DELATE BLOCK



CLEAR

2.1 MANUAL MODE

After pushing [hand] , the desired moving direction can be selected. Then, after pushing [START] the CNC moves the selected axes in the desired direction. The moving speed is controlled by the feedrate potentiometer. The actual value counter monitors the movement.
The key [STOP] immediately stops the movement, the direction keys are cleared.

Alternatively, push [START] and then for example [+X] . The axis starts moving as long as [+X] is pushed. This jogging mode can be leaved by pushing [STOP] .

2.1.1 Clearing the actual value counter

Cearing the actual value counter is done as follows:
1. Select the axis to be cleared.
2. Clear the counter by pushing [C] (see PO N905x)
If a G54 or a tool is active, the actual counter will not display 0 but the programed displacement of G54 or the tool offset.

2.1.2 Exiting the manual mode

To exit the manual mode, the key [MENU] can be selected. The modes [hand] or [TEACH IN] can also be selected.

2.1.3 Hand wheel

The hand wheel can be activated by pressing [+X] . By turning the handwheel, the activated axis is moved forwards and backwards. By pushing [+X] several times, the distance which will be executed with each tick of the handwheel can be changed, starting with 0.001, 0.010, 0.100. This value is displayed on the screen under the item H, f.e.
H...0,100.

2.1.4 Home Position (P 9974 must be present in memory)

A home position can be memorized by pushing [home] and then [arrow] . The actual position is put into memory. The axes then can be moved to the home position by pushing [home] and then [home] one more time.

2.1.5 „Spindle“, „Cooling“, S, T

Spindle on / off, cooling on / off, spindle speed and tool number can be selected by the apropriate keys.

2.2 MANUAL INPUT

This operating mode allows to input and execute single G-functions and parametrical functions. Also cycles like G87 and programm calls can be executed.

The modal G-functions as well as the actual values will be displayed. The G-function to be executed always refers to these modal G-functions.

Pushing positions the cursor to the next data input area, where new values can be input.

The key START executes the programmed block. The execution of the block can be interrupted with STOP . A new G-function can be entered.

2.2.1 Tool change (P9936 must be present in memory)

When entering the manual input, the block

G36 F..... S..... T.... M...

can be input. With move to T and enter 2 f.a. Then push START the tool T2 will be activated and the tool offset of T2 wil be taken from the tooltable P9900.

2.2.2 Jogging mode

1. Switch to incremental mode with G91.
2. Select G00 and input a distance in X or Z.
3. By pushing START the block will be executed.
   By pushing START one more time, the block will be executed again.

2.2.3 Move to a position

1. Switch to absolute mode with G90.
2. Select G00 and input a position in X or Z.
3. Move to the desired position with START

2.3 TEACH IN

The CNC requests the user to input a program number after TEACH IN , which can be acknowledged by →. Then the TEACH IN mode is invoked. TEACH IN allways will be done in G90.
A complete block input buffer for G01 will be displayed by pushing 1 and then →.
With → single adresses can be selected and updated.

Programming will be done as follows:

Select the moving direction and push START. Move the axes under control of the feedrate potentiometer to the desired position and push STOP. The actual position will be dis-played in the block buffer. The block will be stored in memory by pushing the key .

By selecting the G-address, also an other function can be chosen: f.e. G90 or G05. After a movement, the X and Z addresses will be updated. With G05 the R-address can be completed with → the appropriate radius and the block can be memorized.

2.4 AUTOMATIC MODE

When invoking the automatic mode (⇉), the CNC proposes a program number. The proposed program number will be the one processed at last. Another program number can be input if desired.

The program will be executed by pushing (START). If the program number is acknowledged by (→) the CNC proposes a block number. A different block number can be entered if desired.

To acknowledge the block number, push (→). Then the first blocks of the program will be displayed in the lower part of the screen.

With (START) the automatic program execution is activated. Single stepping can be activated by pushing (⇉) and than (START).
At the end of each block, the CNC stops awaiting a new (START) Pushing (START) again disables single steppingfunction.

The automatic mode is exited with (MENU).

If a limit switch is detected during program execution, the CNC will stop all axes immediately and an error message will be displayed.

The programmed speed F can be controlled with the feedrate potentiometer, as far as this feature has not been disabled by programming M23.

M21 will stop screen updating in the (⇉) mode, allowing a faster program execution between blocks.

M22 same function as M21, but effective only for programm calls. The execution of the main program will be displayed.

M24 suppresses the execution of all following M-functions (except M20 - M28) as well as G04 (dwell time). This allows a program test without machine functions.

2.4.1 Autostart P9999

After switching on the CNC, a check for the presence of P9999 in memory is done. If it is present, it will be executed immedietly.
This program allows customizing the CNC to different needs of the user. For example if the actual display should not be set to zero but should contain the actual value of the time before the CNC was switched off, the following program will be used:

P9999
N10 G92 X#102 Z#103

After switching on, P9999 will not be executed if the key (→) is pressed and hold down.

2.5 EXTERNAL DATA

The operating mode [EXT] permits input or output of programs to the serial interface.

By pushing [1] a program number is proposed which can be changed to a different value, depending of the program number which is to be output to the serial interface.

The data output is done in a formated form including control codes for the printer, So that the program will be printed in a good readable form on PC compatible printer with IBM emulation.

By pushing [2] the same is done, however the output is not formated in order to reduce the program to its minimal length.

With the key [3] data from an external PC can be input into the CNC.

In all cases, data is transmited with 9600 baud, XON-XOFF protocoll. The last transmited charachter always is a „%“ followed by „CR“ (0x0dH).
The data format itself can be analyzed by entering a little program on the CNC, transmit it to a PC and edit it on the PC.

We offer a service program for a PC, allowing to receive, memorize, edit and send back CNC programs. Optional programs for translating CNC programs f.e. HPGL to CNC, are also offered.

2.6 INPUT MODE

The input mode allows to input and edit programs. After selection of this mode, a program number will be proposed. For selection of another program number, push C and enter the desired program number.

If a program with this number already exists in memory, the last blocks of this program will be displayed by pushing →. By pushing ↓ the first blocks of the program will be displayed.

If the entered program number doesn't exist, N001 will be proposed as first block number. Acknowledgement will be made with →. The cursor moves to „G..". After entering a G-function and then →, the words corresponding to the G-function are displayed.

When all necessary words of a block have been entered, the block can be stored by pushing . The block number is incremented automatically. An error message will be displayed when trying to memorize an uncomplete block.

Alter block:
If a block already stored must be altered, it can be put into the editing buffer by typing the block number and then pushing „SEARCH BLOCK" →N. With → the cursor can be moved to the word to be corrected. Then the block must be stored again by pushing .

Delete block:
The block to be deleted must be searched with „SEARCH BLOCK" →N. Then push „DELETE BLOCK" //.

Insert block:
Type in the new block number to be inserted and then →. Select the G-function and complete the block. Push to store the block. It will be inserted automatically and the following block numbers will be incremented.
Note: Blocknumbers in G20, G23 are not changed automatically !

List blocks:
Entering a block number and then ↓ will display the next blocks starting with the entered number.

Program overview:
After selecting the „INPUT MODE“ (⇔), an overview of all programs in memory will be displayed by pushing key (C) and then (→). If a program is marked with „!“, this Programm has a „CHECKSUM ERROR“. In that case clean complete memory including P000 and P9900, and make a new transfer from a PC through the serial interface.

Program duplication:
If the request for the program number is acknowledged with (C) and then (⇔) the CNC will ask for the program number to be duplicated and for the new program number. Also P0000 can be duplicated.

Add a program name:
To an existing programm, a program name can be added. Therefor the program number can be ackowledged with the key (⇔) the program name can be typed in and then the key (⇔) pushed again.

Tool table:
The program number P9900 is reserved for the tool table and machinendata. Up to 99 tools (T001 - T099) can be stored with radius and tool offset. These data will be called up by the T-word and are used by the path and the length compensation algorithms.

Reserved program numbers:

P0000 Machine data.
P8000 Text for customer menues.
P98XX are invoqued by a keystroke in the MENU
P9900 Tool table
P9998 Error handling in the automatic mode.
P9999 Autostart.

2.7 CLEAR MEMORY

This mode allows to delete single programs or to clear the complete memory.

First, the CNC requests the input of a code number. This code number can be defined in the machine data. If the number 0 was selected in the machine data, the input of a code will not be requested. Single programs can be deleted by inputing the program number and then pushing .

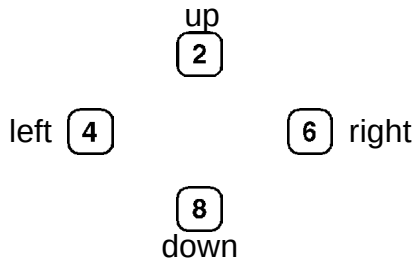
To delete a range of blocks of a program, enter the program number and then .-
The CNC requests the starting and the ending block numbers. All blocks in this range will be deleted.

The whole memory can be cleared by pushing C and then . Here the code defined in the machine is requested, even if the code is 0.
P0000 and P9900 remains in memory. However tey have to be deleted push P000 or P9900 then anser then „CODE“ with the Code (default=0) and then push .

If a program number is entered and the key is pushed, the rest of the memory starting with this program number is cleared.

2.8 GRAPHIC

This mode (✳) allows to display the programmed tool path. The CNC asks for the starting program and block number. The program used at last and the first block within this program will be proposed from the CNC.
Then, the CNC asks for the last block to be displayed. The CNC proposes the same program and it's last block.
Finally the scaling value must be input. Then the display will be cleared and a cross wire, which can be moved on the screen by the following keys, appears:



The cross wire defines the starting point of the program. If the position of the start point on the screen is correct, push (→) to display the contour. With input of (→) the scale can be changed to increase or reduce the picture.

NOTES:

- G04 and all M-functions won't be executed.
- Attention with G20! If a jump is programmed for example in P0001 a jump to P0001 N001, the graphic mode will execute this program continousely. The execution can be interrupted by pushing (MENU) .
- The graphic mode uses an internal actual value counter, which will be cleared by invoking the graphic mode. At the end of the contour, this counter will be displayed.
- First the programmed path is displayed. In a second run, the correct path can be displayed by pushing (START) .

Example

```
001 G11 F1000 S1000
002 G74 X..200,000 Z...10,000
003 G00 X...50,000 Z....0,000
004 G01            Z-..20,00
005 G03 X..150,000 Z-..70,000 I...0,000 J-..50,00
006 G00 X..200,00 Z..10,00
```

2.9 SWITCH OFF CNC

When the CNC is to be switched off, the key O is pressed. Then the complete memory contents are put into a second memory bloc, in order to have all programs duplicated. After that, the CNC can be switched off.
For restoring the saved programs from the second memory bloc, proceed as follows:

Switch off the CNC push the key C and hold it pushed.

Switch on the CNC, continuing pushing the key C until the message

Code oppears, input " 0 " and acknowlage with →.

3 PROGRAMM STRUCTURE

Each programm consists of a program number and up to 999 blocks. Each block has a block number and a G-function. This function tells the CNC what to do in this block, for example a linear interpolation or a movement to the reference point.

3.1 The G-Functions

This part will explain the G-functions and the corresponding block structures.
The CNC can be equipped with less G-functions depending on the control purchased.
A list of the implemented G-functions can be displayed by pushing the key [H] in the MENU of the CNC.

If the G-funktion is only to be executed in the GRAPHIC mode, the key [*] can be pressed when the G-input field is active. The bloc then looks like that: 001*G..

G00 RAPID TRAVERSE

... G00 X.....,... Z.....,...
All two axes can be moved together. The feedrate is determinded by Fmax in the machine data memory.

G01 LINEAR INTERPOLATION

... G01 X.....,... Z.....,...
Up to 2 axes can be moved together. The feedrate (mm/min.) can be programmed before the block with G11.

G02/G03 CIRCLE INTERPOLATION CLOCKWISE / COUNTERCLOCKWISE

... G02 X.....,... Z.....,... I.....,... K.....,...
XZ, are the endpoint of the circle segment, IK are the coordinates of the circle center. The circle center must be programmed relative to the starting point of the circle segment, even if the endpoint is defined in absolute coordinates.

G04 DWELL TIME

... G04 H....,...

Dwell time programming between 0,010 and 9999,990 seconds.

G05/G06 CIRCLE INTERPOLATION WITH INPUT OF RADIUS

... G05 X.....,... Z.....,... R.....,...

Input the desired endpoint in X and Z, the radius in R. The sign of R determines if a small or large circle segment is generated.

G09 SKIP REST OF TRAVEL

... G09 X.....,... Z.....,... M....

The linear interpolation will be done just like a G01 block. However, if the input programmed with M (M161-168, M171-178) becomes active, the interpolation will stop and the next block will be executed.

Possible uses:

- recognization of tool fracture
- digitization of workpieces

G10 CORNER ROUNDING

... G10 X....,... Z....,... X....,... Z....,... R....,...

In G 90, the 1. XZ input field is to be programmed with the first line, in the 2. XZ the second line and in R the radius of the corner between these lines.

```
G00 X01 Z0
G01 X50
G10 X100 Z-10 X100 Z-50 R10
G01 Z-60
```

G11 ADDITIONAL FUNCTIONS F,S,T,M

... G11 F..... S..... T.... M....

These functions allow programming of F, S, T and M. Between 2 movements, no Stop will be generated, the movement will be continous. G94 has the same function but with a stop between the blocs.

G13 ADDITIONAL M-FUNCTION

... G13 M.... M.... M.... M....

G13 allows to program several M-functions in one block.

G20 JUMP TO PROGRAM
... G20 P.... N....
This function executes a jump to the program „P“ and continues with the starting block number „N“. If only N is programmed, the jump will be done into the actual programm. If only P is programmed, the jump will be done to the first block in program P.

G22 CALL PROGRAM WITH REPETITION FACTOR
... G22 P.... N.... W....
The program „P“ will be called, starting with block number „N“. It will be repeated as programmed with „W“. If the program is to be executed only one time, W00 must be programmed. Up to 6 programm calls can be stacked.

Note:
An error „too many calls“ will be reported in the following case:
P0100
N001 G..
.
N010 G22 P0100 N0001 W0001

G23 JUMP/CALL PROGRAM WITH REPETITION FACTOR AND CONDITION
... G23 P.... N.... W.... M....
Program „P“ will be called, if condition M is true. If W is not programmed, a jump to program P will executed. The condition M can be all waits for input f.e. M161. The jump or call only will be executed if input 1 in I/O card 1 is active.

G33 THREAD
... G33 X.....,... Z.....,... K.....,... J.....,...
During thread cutting, the axes X and Z are synchronized with the spindle, to that slight variations in the spindle speed are automatically eliminated.
The ending point of the thread is defined in X, Z. K is the pitch relative to the Z axis, J is the acceleration and deceleration distance.
For the execution of G33, the CNC waits for the reference pulse of the encoder connected to the spindle, then the acceleration distance is performed, then the thread itself, then the deceleration distance. If J is negative, the CNC does not wait for the reference pulse of the encoder.

P33 Example for thread cutting
N1 G11 S100 M3
N2 G00 Z1
N3 G33 Z-10 I1 J1

G36 TOOL CHANGE
... G36 F..... S..... T.... M....
The pogrammed values F,S,T,M are put into the registers #080 - #083 and then program P9936 is called. Here the customer can store his tool change program.

G40 RADIUS CORRECTION OFF (Initial state)
... G40 correction off
This function will reset G41/G42. The next programmed linear interpolation in the XZ plane will be used to exit the tool path.

G41/G42 RADIUS CORRECTION LEFT/RIGHT
For correct use of the path compensation, the following notes must be observed:

- Before using a path compensation, an appropriate tool must be programmed.
- G41 compensates always lefthand, G42 always righthand in the moving direction of the tool.
- The compensation must be programmed one block before the block to be corrected. This block will then be used to enter the compensated path.
- G40 will switch off path compensation. The following movement in XZ is used to exit the compensated path.
- During the compensation absolute or incremental input can be programmed. It is also possible to call subroutines (G22), however the called programm must contain at least one G01 movement.
- If the last program block is reached without encountering G40, the path compensation mode will be exited automatically.
- Jumps with condition (G23) will always be executed in the compensation mode. During G41/42, no parametrical functions should be used.

Example for path compensation:

```
P9900 Tool table
001 X....0,000  Z....0,000  R....5,000

P0001 Test program
001 G92  X150 Z10
002 G42
003 G00  X0 Z0
004 G01  X50
005 G01       Z-50
006 G01  X70
007 G40
008 G00  X150 Z10
```

In the graphic mode, a straight line is used for the programmed path, a dashed line for the corrected path.

G53 DISPLACEMENT OFF (Initial state)

G54 DISPLACEMENT I

... G54 X.....,... Z.....,...

With G90, the values programmed in G54 will be added to all the following movements. With G91, the displacement will only once be added to the first movement in the propriate axis.

Example:

The program P0010 is programmed in absolute coordinates.

```
P0010
001 G90 Absolute input
002 G00 X....0,000 Z....0,000
003 G01 X...20,000 Z....0,000
004 G01 X...20,000 Z...20,000
005 G01 X....0,000 Z....0,000
```

The actual position of the CNC is X,Z. P0010 now should be executed in the position (100,50).

```
.
.
010 G90
011 G54 X..100,000 Z...50,000
012 G22 P0010
```

During the execution of P0010, the programmed values and the absolute coordinates are displayed.

G55 DISPLACEMENT II

As G54, however must be switched off with G55 X0 Z0. This function must not be used together with G86 in the TURNING MODE.

G67/68 SOFTWARE LIMIT SWITCH -/+

... G67 X....,... Z....,...

If these limits are passed, the CNC will stop and display the error message SOFTWARE LIMIT SWITCH.

G74 REFERENCE POINT

... G74 X.....,... Z.....,...

This function moves the axes to the corresponding limit switches while the direction will be determined by the sign of the programmed value. This value will be set into the actual value counter.

Example:
... G74 X....0,000 Z-...1,000

X will be moved to the positive, Y to the negative limit switch.

Note: The axes always must be moved separatly to the reference point.

It is recommented to generate a program P0074, which will always be invoked to move X and Z to their reference point and to put the actual counter to the correct value for X and Z.
The currect value for X would be the diameter on the actual point. The correct value for Z schould be the actual distance to the spindle or to the workpiece.

```
P0074
001 G74 X0
002 G74       Z0
003 G92 X150 Z10
```

After execution of P74, the actual count displays X 150 Z10, which would be the actual diameter in X and the distance of the working piece in Z.

G75 SCALE FACTOR ON (Initial state)

... G75 X.....,... Z.....,...

This modal function allows increase, decrease and reflection of the following programs. All the following movements will be multiplied by the values programmed in X, and Z. A negative value will switch the polarity of the movement.

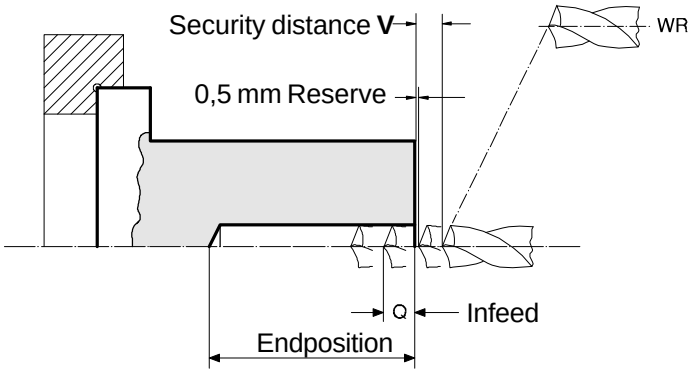
G76 SCALE FACTOR OFF

G82 DEEP DRILLING

... G82 Z.....,... Q.....,... V..... H.....,...

Input data: Z = Endposition (Absolut)
Q = Infeed
V = Security distance
H = Dwell time

Example:
P0001
N001 G11 F1000 S1000 T01
N003 G00 X0 Z1 : move to the security distance in Z
N004 G82 Z-..20,000 Q-...4,000 V....1,000 H....0,100

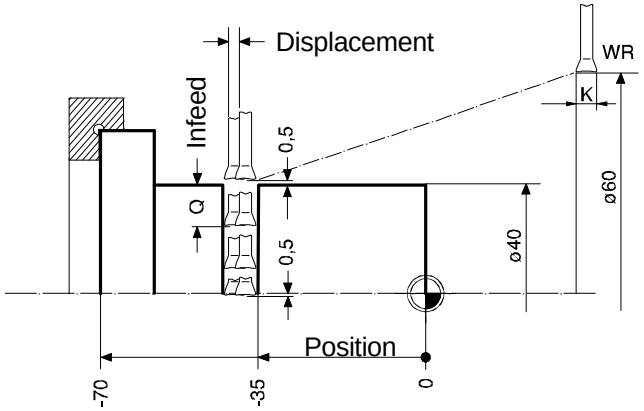


G83 CUT OFF

... G83 X.....,... Z.....,... K.....,... Q.....,...

Input data: X,Z = Starting position
K = Tool dimension Z
Q = Infeed

Example
P0001
N001 G00 X0 Z0
N002 G83 X...10,000 Z...20,000 K....5,000 Q-...3,000



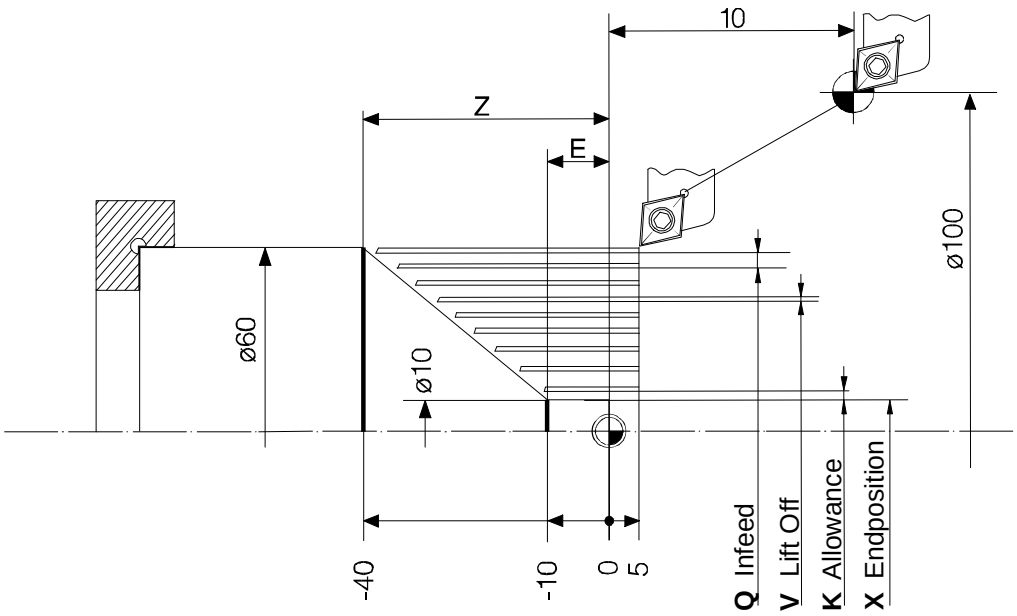
G84 TAPER CUTTING HORIZONTAL

N... G84 X.....,... Z.....,... E.....,... Q.....,... V.....,... K.....,...

Input data: X = Endposition X
Z = Endposition of the large diameter
E = Endposition of the small diameter
Q = Infeed X
V = Lift off
K = Allowance

The allowance K remains at the end of the cycles.
If the V value is negative, only a rough cutting is executed. With a positive V value, also a finishing cut with half of the programmed speed is done.

Example:
N001 G11 F1000 S1000
N002 G92 X100 Z10
N003 G00 X60 Z5
N004 G84 X+..10,000 Z-..40,000 E-..10,000 Q-...4,000 V....1,000 K....1,000



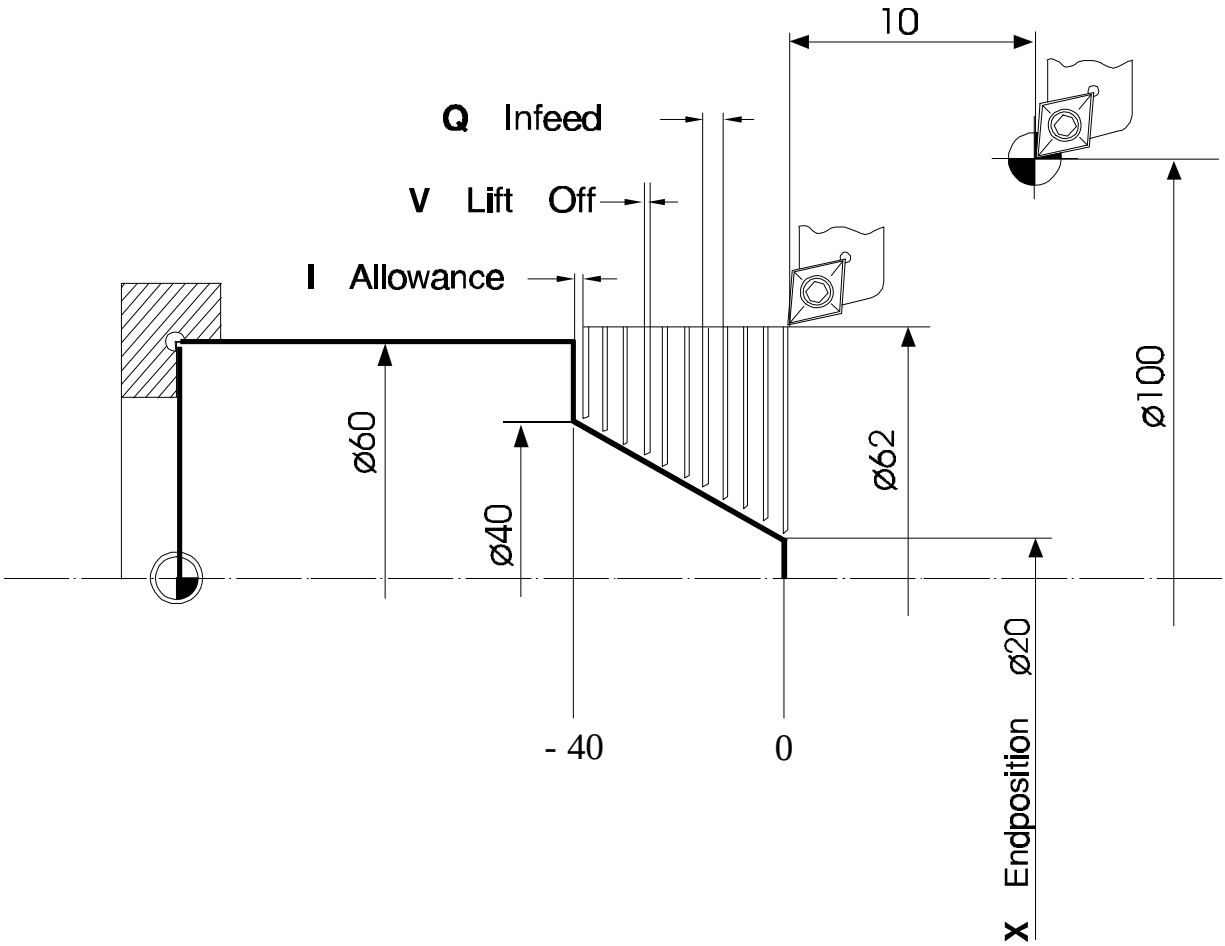
G85 TAPER CUTTING VERTICAL

N... G85 X.....,... Z.....,... E.....,... Q.....,... V.....,... I.....,...

Input data: X = Endposition X
Z = Endposition Z at the small diameter
E = Endposition X at the large diameter
Q = Infeed X
V = Lift off
I = Allowance

The allowance I remains at the end of the cycles. E must be greater than X.

Exampel:
N001 G90
N002 G11 F1000
N003 G92 X100 Z10
N004 G00 X62 Z0
N005 G85 X...20,000 Z-..40,000 E...40,000 Q-...2,000 V....1,000 I....1,000



G86 CONTOUR TURNING

N... G86 X.....,... Z.....,... I.....,... K.....,... P.....,... V.....,...

Input data: X = Allowance, should be greater than the diameter of the tool
Z = Allowance, should be greater than the radius of the tool
I, K = Infeed X or Z
P = Programnumber (<8000) which defines the contour
V = 1,000

Example:

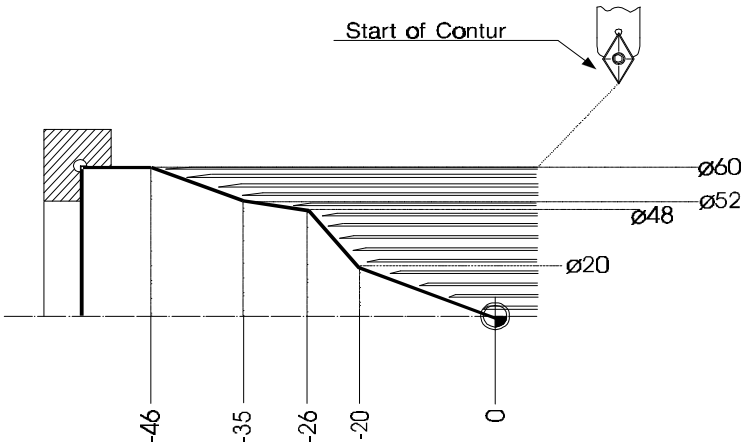
```
P0086
N001 G11 F1000  T1                          ;P9900  T1  R0,4
N002 G90
N003 G92        X..100,000      Z ..10,000
N004 G00        X...60,000      Z ....5,000
N005 G86        X.....1,000     Z.....0,500  I....0,000  K-...2,000   V1,000   P186
```

Z..... greater than radius of T1
X..... greater than diameter of T1

```
P186
N001 G90                                        >  Starting blocs
N002 G42                                        >

N010 G00        X....0,000      Z .....0,000    >
N011 G01        X..20,000       Z-.. 20,000     >
N012 G01        X .48,000       Z-...26,000     >  Contur definition starts
N013 G01        X..52,000       Z-...35,000     >  with N10, both X and Z
N014 G01        X..60,000       Z-...46,000     >  must be programmed.

N015 G40                                        >  Ending blocs
N016 G00                        Z ....0,000     >  containing necessarily G40 !!!
```



Contur definition must start with bloc number N10 ! Allowed are G01, G02, G03! The maximum diameter of the contur must be smaller or equal to the starting diameter of the cycle.

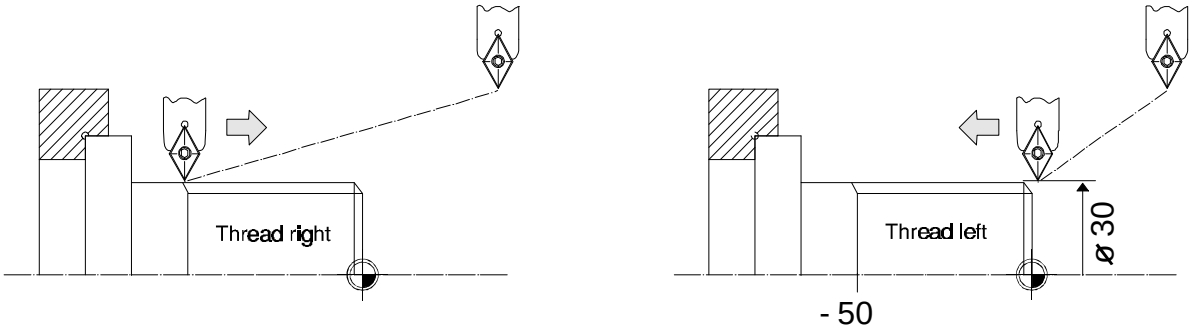
G87 THREAD CYCLE

N... G87 Z.....,... K.....,... I.....,... Q.....,... E.....,... J.....,...

Eingabe: Z = Endpoint
K = Pitch
I = Depth of thread
Q = Infeed X
E = Angle
J = Acceleration and deceleration distance

Example:

```
N001 G11 S100 M03
N002 G00 X50  Z1
N003 G87 Z-..50,000 K....1,000 I-...1,000 Q-...0,300 E...60,000 J....1,000
```



Inner thread is not possible with G87. However the following program helps:

```
P10                                ;Depth of thread is 10 ∗1mm
N01  G22  N10  W9
N10  G91
N11  G33  Z -100  I....      J....
N12  G00  X -10
N13  G00  Z  100
N14  G00  X  10
N15  G00  X  1
```

G88 RELIEF GROOVE

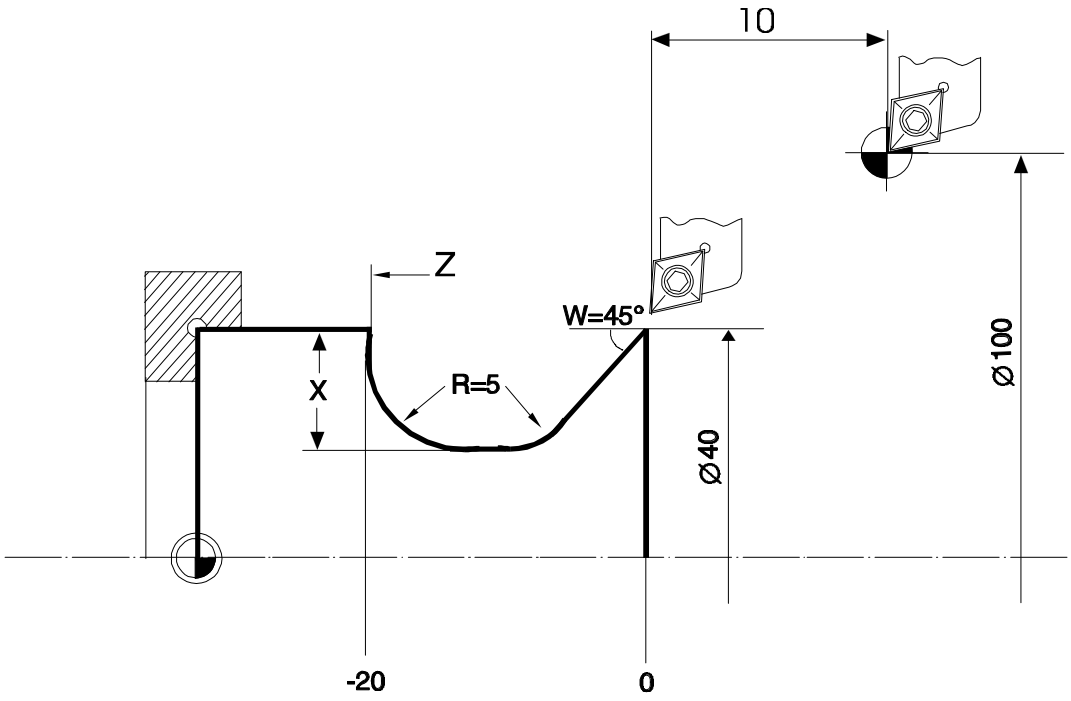
N... G88 X.....,... Z.....,... R.....,... W.....,...

Input data: X = Depth of relief
Z = Lenght of relief
W= Angle
R= Radius

Example:

N001 G00 X..40,000 Z.....0,000

N010 G88 X-...5,000 Z-..20,000 R....5,000 W...45,000



G90 ABSOLUTE INPUT

N... G90 absolute input

This function switches from incremental to absolute input. All the following inputs will be interpreted as absolute values. All X values are programmed in diameter.

G91 INCREMENTAL INPUT

N... G91 incremental input

This function switches from absolute to incremental input. All the following inputs will be interpreted as incremental values. All values are programmed in radius.

G92 ZERO OFFSET
N... G92 X.....,... Z.....,...
The programmed values are taken into the actual value counter. If G54 or a tool is active, these values are added to the actual counter.

G94 FEEDRATE IN MM/MIN
N... G94 F..... S..... T.... M....
The feedrate can be programed in mm / min. G94 cancels G95.

G95 FEEDRATE IN MICROMETER / ROTATION
N... G95 F..... S..... T.... M....
The calculation of the feedrate in mm/min is done by the following formula, by using the last programed spindle speed S or the actual measured spindle speed.

F(mm/min) = F(µ/R) x S(R/min) / 1000

F can be calculated from the measured or programmed spindle speed, machine datum N905X

G96 CONSTANT CUTTING SPEED
N... G96 V..... S..... T....

With G96, the cutting speed is hold constant. The CNC calculates the appropriate spindle speed for each diameter. The cutting speed in m/min is programed under the adress V.
With each change in diameter the spindle speed is calculated and updated. The calculated speed could be to high , so a maximum spindle speed is input at the adres S.When G95 is programed in addition to G96, also the feedrate F is updated synchronously to the variation of the spindle speed.

$$S\ (U/min) = \frac{V\ (m/min) \times 1000}{X\ (mm) \times p}$$

where X is the actual diameter, that is to say the value in the X display.

```
P0001 Test G96
N001      G94
N002      G11 F..500
N003      G96 V..100 S..2000
N004      G90
N005      G92 X..50,000
N006      G01 X...0,000
N007      G01 X..50,000
```

G97 CONSTANT SPINDLE SPEED
N... G97 F.... S.... T.. M..

The spindle speed is programed in the rotations per min, which is the initial state when switching the CNC on.

3.2 THE M-FUNCTIONS

The M-functions have the following meaning:

M00	programmed stop
M01	programmed stop with acoustic signal
M02	program end
M03	spindle on clockwise
M04	spindle on counterclockwise
M05	spindle stop
M08	cooling on
M09	cooling off
M10	clamping on
M11	clamping off
M15	acoustic signal
M16	wait for „Input 1“ to go active
M17	wait for „Input 1“ to go inactive
M18	wait until no more key is pressed

Use of the I/O cards 1 and 2

M0140	set all outputs on I/O card 1
M0240	set all outputs on I/O card 2
M0141 - M0148	set output 1 - 8 on I/O card 1
M0241 - M0248	set output 1 - 8 on I/O card 2
M0150	reset outputs 1-8 on I/O card 1
M0250	reset outputs 1-8 on I/O card 2
M0151 - M0158	reset output 1 - 8 on I/O card 1
M0251 - M0258	reset output 1 - 8 on I/O card 2
M0160	wait for all inputs to go active on I/O card 1
M0260	wait for all inputs to go active on I/O card 2
M0161 - M0168	wait for input 1 - 8 to go active on I/O card 1
M0261 - M0268	wait for input 1 - 8 to go active on I/O card 2
M0170	wait for all inputs to go inactive on I/O card 1
M0270	wait for all inputs to go inactive on I/O card 2
M0171 - M0178	wait for input 1 - 8 to go inactive on I/O card 1
M0271 - M0278	wait for input 1 - 8 to go inactive on I/O card 2
M0180	invert all outputs on I/O card 1
M0280	invert all outputs on I/O card 2
M0181 - M0188	invert output 1 - 8 on I/O card 1
M0281 - M0288	invert output 1 - 8 on I/O card 2

The waiting functions M16, M0x60 - M0x68 and M0x70 - M0x78 can be skipped with START. This however can be inhibited by M2347.

Special M - Functions

M2241	(M21)	block updating off in the automatic mode.
M2242	(M22)	block updating off in program calls
M2243	(M23)	feedrate potentiometer off
M2244	(M24)	test run without G04 and M-functions
M2245	(M25)	test run with rapid traverse
M2246	(M26)	keyboard off
M2247	(M27)	wait for IN POSITION (see also N803)
M2248	(M28)	actual value display off. Display remains active during SINGLE STEP

M2251 - M2258 will reset the preceeding functions.

M2343	(M33)	feedrate potentiometer off with G00
M2344	(M34)	by pressing „MENU“ go to P9999
M2347	(M37)	the „wait for input“ cannot be left by pressing „START“ or “MENU“

M2351 - M2358 will reset the preceeding functions.

3.3 The F function (feedrate)

The feedrate will be programmed with the F-function. Possible values are 1 to 999999 mm/min. The CNC will only move at values below or equal to Fmax defined in the machine data.

Example:
... G11 F1000
... G01 X..100,000 Z..100,000
The X and Z axes will not move with 1000 mm/min. each, but only with 1000:1,4=714 mm/min. Because both axes are moving, the resulting feedrate will be 1000 mm/min.

3.4 The S-function (spindle speed)

The spindle speed will be programmed with the S-function.Possible values are 1 to 60000 in r.p.m. The CNC will only accept values below or equal to Smax defined in the machine data memory.

The output SPEED of connector X2 (option) gives a voltage between 0V (= S0000) and 10V (= SMAX) and proportional to the programmed spindle speed S. To update this output, program G11 S..... M03.

3.5 The T-function (tool call)

With the T-function up to 99 tools (T01 - T99) can be programmed. These tools will be defined in P9900 with N0001 to N0099. Invoking G41, G42 the data of the just activated tool will be read out of the tool table P9900. If another tool is to be used, programming can be made with the T-function.

The desired tool must be called before programming a path or length compensation with G11 T..... The T-function automatically activates the tool length compensation, which can be switched off with T00.

5. PARAMETRICAL FUNCTIONS

The parametrical functions are an essential extension of the possibilities of a CNC. The user can develop cycles or make calculations within his program.

5.1 Linear interpolation with parameters

Select linear interpolation (G01) and push (→). The input buffer for X is active now. Push (↓) and enter a 3-digit number representing a parameter register.

N001 G01 X.....#004 Z...10,000

The actual contents of parameter register #004 will be taken as endpoint for X and the value 10,000 will be used for Z.

All addresses can be programmed in this way.

5.2 Calculation with parameters

100 parameter registers (000-099) are available to the user. They can be manipulated by mathematical functions. To select these functions (f.e. addition), push (↓) while the G-address is active. The input line now looks as follows:

N002 ↓..

Now the code for addition (01) can be typed in. After using (→) the following display will appear:

N002 ↓01 #... = #... + @.....,...

Now one can define f.e.:

N0002 ↓01 #001 = #002 + @....#003

This means that the new value in parameter register #001 is the result from the addition of the values of parameter #002 plus #003.
The inputfield @....,... also can be programmed directly.

N0003 ↓01 #001 = #002 + @....3,000

The new value of #001 is the result from the addition of the value in #002 and the value 3,000.

5.3 INDIRECT PROGRAMMING

Also indirect programming is possible:

N0004 ↓01 #001 = #002 + @....#210

The new value is calculated from the contents of #002 and the contents of the register whose adress is defined in #010. #200 to #255 allow indirect programming with registers #000 to #055.
Or: ↓94 #210 means that the text, whose number is in #010, will be displayed.

5.4 Reserved parameters

The parameter registers #040-#099 can be changed by the cycles. If no cycles are used, they are available to the user.
A cycle G36,G84 - G89 will load #080 to #089 with the programmed values. #090 will be loaded with the byte defining which axes have been programmed in the bloc. The cycles G81 - G82 will load #070 to #079. #100 will be decremented all 10 ms to zero.
Registers # 102, # 103, # 104 contain the “Home Position“.

5.5 Parametrical functions

102 #103,

↓00 #... = @.....,...	assign value
↓01 #... = #... + @.....,...	*addition
↓02 #... = #... - @.....,...	*substraction
↓03 #... = #... * @.....,...	*multiplication
↓04 #... = #... / @.....,...	*division
↓10 #... = COPY #...	copy contents
↓11 #001 = ATN #002	arcustangens of (#02)/(#03)
↓12 #001 = PYTH #002	#002 = SQRT ((#02)² + (#03)²)
↓13 #... = CPL #...	calculate complement
↓14 #... = ABS #...	calculate absolute value
↓15 #... = SQRT #...	calculate the root
↓16 #... = SIN #...	calculate sine (gives sine x1000)
↓17 #... = COS #...	calculate cosine (value x1000)
↓18 #... = AND #...	*logical AND function
↓19 #... = DIV2 #...	division by 2
↓20 #... = OR #...	*logical OR function
↓50 (JUMP ZER TO) N...	jump if result zero
↓51 (JUMP POS TO) N...	jump if result positive
↓52 (JUMP NEG TO) N...	jump if result negative
↓53 (JUMP TO) N...	jump without condition
↓54 (JUMP NZ TO) N...	jump if result not zero
↓55 (JUMP DEC TO) N...	decrement register #099 and jump if #099 not equal to 0

Functions marked with * will influence the result register used for jumps with condition.

5.6 PARAMETRICAL SPECIAL FUNCTIONS

The programmable values can be between 0 and 255!

↓80		Input of text. For a space, press „±“. Shift and then → deletes the last character. The last character always should be a letter, a number or a space. If the last character is a „=“, an input field will be opened in the automatic or graphic mode. A value can be input which will be transfered into a parametrical register by pressing → or „START“. The number of this register is the same as the block number in which the function ↓ was programmed.
↓81	#010	Display or print the the text, which is stored in the program P8000 with the block number N0010.
	#200	The value in the register #000 - #055 defines the text number to be displayed.
	#255	In addition the content of register #000 defines, how and where the text will be output.
	#000 = 0,000	Text output to display.
	#000 = 0,001 - 10,217	Content of register #000 defines the screen position of the text.
	#000 = 16,384-	as before, however an eventual request to input a 32,767 value will be ignored
	#000 = 100,000	Output to printer.
	#000 = 150,000	Output to serial interface
↓82	#000	Subprogram call of the CNC operating system. #040 contains the address, #041,42,43,44 will be stored to HL,DE,BC,A.
↓83	#...	Like â81, but the textes will not be selected from P8000 but from the actual program in progress.
↓83	#... #A #B	If „=“ is present at the end of the text, an input field of the length A with B digits after the decimal point is opened. A can be 1 to 9, B can be 0 to 5, however a must be at least B+2! If a sign has to be displayed, b must be 16 to 21.

↓84 #A #B #C #D #E Read/Write

A = 0 = Memory access
16 = I/O access
128 = DILAG access

B = 1 = Read
2 = Write
(5 = Read 4 Bytes from DILAG)
(6 = Write 4 Bytes to DILAG)
(8 = Reset Dilag)

C = parametrical register, where to read or from where to write.

D = parametrical register with the memeory adress, from where to read or where top write.
If A = 16, then D is directly the I/O adress for read/write. (The I/O cards 1-8 have the adress 64 - 71)
If A = 128, then D is directly the DILAG register for read/write.

E = number of values to transmit.

↓86 #A#B#C monitoring system on.

A= 72-73 lag errorr X-Z

B= 76-77 given value X-Z

C= 0

To display the given value of X and the corresponding lagerror program ↓86 #72 #76 #0 #0.
This function alters registers #40 #49.

↓87 #010 get a block from memory: Program the desired program number in #010, the block number in #011. The block is put into #012 - #016.

↓88 #010 as ↓87, but 0#012 - #016 are put into memory.

↓89 #A #B keyboard scan. B=0: The code of the depressed key is put into register A. If no key is pressed, the ZERO flag will be set. B=1: The code of the last depressed key is put into register A.

↓ 90	#000	insert blank line.
↓ 91	#000	clear screen.
	#001	clear a part of the screen.
	#A #B	The starting point A,B and
	#C #D	the length/height in C,D. (values between 0 and 255)
	#002	invert part of the screen.
	#255	set all display points.
↓ 92	#A #B	display registers #A to #A+4. (B=0) With B=1, the output will be on the printer.
↓ 94	#...	display intern text.
↓ 95	#...	display error message and stop automatic or graphic mode, f.e.
	#058	displays the message „check block“.
↓ 96	#000/01	save/restore state of G90/91, G94/95 and M21-M28
	#002/03	as #001/002 however used in G81 - G83.
	#004 #A	put actual T,S,F,R to register #A - #A+3.
	#005 #A	store the actual value of interpolator to #A - #A+7.
	#006 #A	store the actual value of DILAG to #A - #A+7.
	#007 #A	store 8 analog inputs on interpolator to #A - #A+7.
↓ 98	#A #B	draw line. Programm the starting point in #A,#B
	#C #D	the ending point in #C,#D. at the upper left is position 0,0, at the lower right position 255,255.

Hint:
Parametrical functions are executed on the fly during a movement. If the movement should have finished before the execution of the parametrical function, a block G13 M.... must be programmed after the movement.

6. MACHINE DATA

The machine data allow an easy adaptation of the CNC to different mechanics. The machine data memory is addressed by P0000 starting with N699.
The data concerning one axis can be programmed for each axis separatly. An interpolation will be done with for example the lowest feedrate F of all participating axes.

The value in brackets is the default. Only if a different value for an axis is needed, it must be programmed in P0000.

N699XZ CORRECTION REFERENCE POSITION (0)
This value will be moved after the referencepulse with the feedrate programmed in N902A.

N700XZ F MAX (1000)
Maximal feedrate in millimeter/min. The maximal interpolationfrequency of the CNC is 30KHz with steppingmotors and 600 KHz with servomotors. The resulting frequency with given FMAX and STEPS/MM is calculated as follows:

$$f(Hz) = \frac{FMAX}{60} \times (STEPS/MM) \qquad FMAX = 60 \times f(Hz) / (STEPS/MM)$$

N701XZ F START (100) **N702XZ F STOP** (100)
Start and Stop frequency in mm/min of an interpolation. The smallest programmable value is 1.

N703XZ B START (500) **N704XZ B STOP** (500)
Acceleration and deceleration in multiples of 10 mm/sec².

N705XZ Z OFF REF (200)
After a reference movement, this feedrate is used for moving the axis off the limit switch.

N706XZ STEPS per .... (200)
N707XZ .... MM (1)
These 2 parameters define both together for each axis the resoltion of the system.
The CNC needs the following information for stepping motors:
How many steps (N706) give a movement of how many millimeter (N707)?
Example: A stepping motor for the X axis makes 1000 steps per rotation and is connected to a 5 mm spindle. The resulting values are:

706 X...1000
707 X.....5

For 120KHz stepping system (highresolution steppingdrive), the value N 706XZ must be divided by 4.

N709XZ LIMIT SWITCH DEBOUNCE TIME (10)
During this time in ms at least the limit switch signal must be stable.

N710XZ F REFERENCE (500)
Feedrate in mm/min for G74.

N711XZ WAY OFF FROM LIMIT SWITCH (1000)
In G74, this value in µm will be moved off the limit switch with the feedrate programmed in N705.

N712XZ MAXIMAL WAY OFF (50000)
If the limit switch is not desactivated in between this value in µm while moving away from the limit switch, the CNC stops and displays an error message.

N713XZ MAXIMAL DISTANCE FOR STOP (0)

N714XZ BACKSLASH in µm (0)

N722XZ FEEDRATE OF BACKSLASH COMPENSATION in µm/min (0)
With a value of 0, the feedrate of the compensation is the value from N701

N790XZ AXISDEFINITION (771)
This value is composed for each axis separatly from the sum of the following options:

01: „Main axis“. A main axis will influence the feedrate during an interpolation. In most cases X,Y,Z will be main axes.
02: „Linear axis“ with + and - limit switches which always will be active. A rotary axis will react to the limit switch only during G74.

Conclusion:

Value	**Function**
01	Main axis
02	Linear axis
16	Servomotor instead of stepping motor is connected
32	Search of referencepulse. With G74 and after moving to and then off the limit switch, the distance programmed in N711 will be moved in the same direction with the feedrate N705. Then the axis moves on with the speed F REFPULSE (N902A) until the reference pulse of the transducer is detected. Here the internal counter are zeroed.
256	Limitswitch + connected
512	Limitswitch - connected
1024	Limitswitch + is normally open
2048	Limitswitch - is normally open
4096	Invert moving direction
———	
SUM	

The input for each axis can be calculated by adding the values of the desired functions.

Default for X,Z = 771 (1+2+256+512), other axes = 770 (2+256+512).

The machinedata N800-N813 are necessary for Servomotors.
N790 must be programmed with 16 (Servomotor).

N800XZ P-FACTOR (20)
The output voltage to the servoamplifier is proportional to the lag error.
The maximal output voltage of +/- 10V will be reached with a lag error.

$$\text{of f.e.} \quad \frac{\text{32000 (constant)}}{\text{20 (P-factor)}} = 1600 \text{ increments.}$$

This value of 20 works with most applications.

N803XZ IN POSITION F (10)
When M27 is active, the CNC waits at the end of a movement, until the lag error has become smaller than the value IN POSITION.

N804XZ LAGMAX F (1600)
If the lagerror becomes graeter than this value, the CNC stops and displays an error message.

N812XZ ZERO OFFSET (0)
When an axis stands still and the lag error cannot be corrected to 0 with the trimmer “Offset“ of the servoamplifier, then an offset can be defined here for all axes together. A value of 35 will give +15mV, a value of 65550 gives -15mV.

N813X SERVO ON (0)
Activates the SERVO ON output of connetor X11

value	activated axis
1	**X**
2	**Z**
3	**X+Z**

N813Z FATAL LAGERROR (32000)
If the lagerror of one axis exceeds this value, the servoamplifiers are disabled. This value should always be greater than N804XYZU.

N900A SPINDLEAXIS (0)

0: no spindle axis; 1-4: axis 1-4 is spindle axis

M03 or M04 activates the spindleaxis, it can be programmed with G11 S..... .
M03/M04 takes the spindle out of the position control and lets it run with the programmed speed S.
M05 takes the spindle again in the position control.
It now can be positioned with f.e. G00. In order to reference the spindle with G74, in P0 N790 the option 64 must be activated.
After M05 the spindle must be referenced.
Spindleaxis positioncontrol only possible with D-23.

N900X CODE (0)
When going to INPUT MODE, EXTERNAL DATA, TEACH IN and CLEAR MEMORY, the CNC requests a usercode, which can be determined here. A value of 0 disables this request.

N901X S MAX (3000)
Maximal spindle speed in r.p.m. The optional S-output generates a voltage between 0 and 10V, according to S0000 to S3000. The programmed value must be a rounded up multiple of 250.

N901Y BAUDRATE (9600)
Defines the baudrate of the serial interface on the CPU.

N902X RESERVED CONTROLCODES I (0)
The date N902X is composed from the following numbers:
64: default: XON-XOFF protocoll in EXTERNAL DATA for serial data input. Only with baudrate=9600.

N902Y MISCELLANEOUS CONTROLCODES II (128)

04: In the manual mode, only Jogmode activ.
08: An active G54 or tool will not influence the actual value counter.
16: No delay with direct change from M03 to M04.
64: Generate „PARITY EVEN“ for serial output in EXTERNAL DATA.
128: M03 sets the output SPINDLE ON, M04 sets SPINDEL R/L.
512: Tool is behind Z-axis.

N902Z LANGUAGE (0)

0: German
1: English
2: French
3: Dutch
4: Italian
5: Spanish
6: Portugese
7: Swedish

N902U,V INITIAL VALUE M23xx, M22xx (0)
M2341 corresponds to the value 1, M2342 ¸ 2, M2343 ¸ 4, M2344 ¸ 8. With the value 256, no initialisation will be done.

N902A F REFPULSE (20)
Feedrate in mm/min. for search of referencepulse on transducer, if N790 is programmed with 32.

N903XYZUVABC I/O-INITIAL VALUES (0)
These values will be put to the ouputs of the I/O cards 1-2, when switching on or when changing to MENU.
If a value of 250 is programmed, the corresponding I/O card will not be initialized.

N904A G-Function for MANUAL INPUT and **TEACH IN** (0)

N904V
16: hand wheel external
2048: Anzeige des programmierten S-Wertes kommt aus #105.

N905X
32: In the normal mode, only one keystroke is necessary for C and ↵
128: In G95 the speed of the axes depends of the measured spindle speed and not of the programmed spindle speed.
256: In G95 the feedrate potentiometer is disabled.
512: External interrupt will be generated by activating input 8 on X1 (signal IN 8) and P 9998 will be invoked.

N906A **Lubrification intervall in minutes** (0)

N906Z ACTUAL VALUE DISPLAY UPDATE-TIME (0)

N921XYZU SCREEN ADAPTION
X 640 pixel in X * 2
Y 240 pixel in Z
Z 114 mm in X
U 85 mm in Z

N923U Teiler für externes Handrad
Mit 4112 wird der Teiler auf 4 eingestellt

N925X SPINDLE PULSES per rotation of the spindle

7. GENERAL INFORMATIONS

Display version of the CNC program
In the „MENU“ push the key and hold it down until an error message is displayed. At the same time the implemented version is displayed.

Program archiving
Programs are valuable and cost a lot of time to generate them. That is why that at least 2 copies of each programm should exist outside the CNC, one copy from the preceeding day and one of the day before

Initialization of the CNC !!! All programms and machinedata are deleted !!!
Switch the CNC on or activate RESET, hold the key for 3 seconds and release it. The message „CODE“ appears on the screen. Then push the key 0 followed by which initialises the CNC. All programms and machinedata are deleted.

Code override
If you programm a code number in P0 N900 and you forget this number, you have the possibility to clear the code number by starting P9990 in the AUTOMATIC mode.

Keyboard simulation with V24 interface

The CNC keyboard can be simulated on an external computer using the Serial Interface.

External computer	Generated function	External computer	Generated function
@	+X	O	CLEAR MEMORY
A	-X	P	SEARCH bloc
B	+Y	Q	DELETE bloc
C	-Y	R	EXTERNAL DATA
D	+Z	S	
E	-Z	T	SPINDLE
F	SINGLE bloc	U	COOLING
G	START	H	STOP
I	MANUAL	<	MENU
J	GRAPHIC	=	**->**
K	AUTOMATIC	>	CLEAR
L	MANUAL INPUT	**.**	**.**
M	TEACH IN	0-9	0-9
N	REFERENCE POINT $		

- ENTER, ? INPUT MODE or STORE into memory
- / returns given position, status, operating mode, error status. The status corresponds to the output of I/O card 4.
- ! returns Actual Value from the DILAG cards. This is the Actual Value of the axes XYZU and VABC with a time jitter of aproxymatly 500 ns.
- & returns given position in HEX format.
- „ returns the numbers of the programs in memory.
- (state of the inputs and outputs of the i/o cards 1 - 8.
-) returns the following values: POT%, programmed F, actual F, T
- * returns Software Version of the CNC

Programs also can be sent to the CNC over the Serial Interface (X6) by Remote Control.This can be done for example by sending the following data to the CNC:
R7>=
P0001 CR
N1 G0X55 CR
% CR

- R Switch to EXTERNAL DATA
- 7 Select input through Serial Interface
- > Clear input field
- = 
- Then follows the program to be transmitted.

These functions can be simulated and tesed with each TERMINAL program like Hyper-terminal or of course our DIENSTPROGRAM.

The following BASIC program allows to send data from the keyboard of a PC to the CNC:

```
10   CLS:OPEN „com1:9600,n,8,1,RS,CS,DS,CD“ AS #1
20   REM Keyboard scan.
30   A$=INKEY$:IF A$=““THEN 30
40   REM Wait until CNC is ready for receiving data.
50   IF (INP(&H3FD)AND 64)=0 THEN 50
60   IF (INP(&H3FE)AND 16)=0 THEN 60
70   PRINT #1, A$
80  GOTO 30
```

If COM2 is to be used, correct line 10 (COM2), line 50 (&H2FD) and line 60 (&H2FE).

7.1 PUTTING INTO OPERATION

The CNC needs a power supply of 230V. For a quick first test, the CNC is connected to the mains and switched on. The CNC emits a short beep and after 5 seconds it should display a DOWNLOAD message and then go into the main MENU. Now you can go into the INPUT MODE and enter a short programm which can be displayed in the GRAPHIC MODE.

7.2 STARTING HINT

At the first start the CNC should be initialized.

7.3 SERVOMOTORS

Note: **- Use shielded cable. Connect shield to the case of the CNC to the ground connection.**
- Use transducer with TTL output!

If the servodrivers were purchased from our company, you must only connect the servomotors to the corresponding outputs MOTOR X, MOTOR Z. The pin connections can be found on page 9/10.

After that the connector X11-x of the CNC is wired to X10 on the servo driver unit.

If the servo driver was purchased from an other company, the connector X11-x (page 9/10) must be wired, first only the X-motor. For that pins 11, 12, 9, 10 must be used.

The servoamplifier must have a DIFFERENTIAL-input.
With +/- 10V at input, the speed of the servomotor must be adjustable within the range used later for moving the axis. The maximal speed is not necessarily 3000 rot/min, but can be much lower f.e. 600 rot/min. With a spindle pitch of 5mm this will result in a feed-rate of 3000 mm/min!
Each servoamplifier has 2 pins, which, when shorted, activate the amplifier.
It is important to get to know, which of these 2 connectors is the positive one!

After these preparations, the servoamplifier can be wired to pins 11, 12 and 9, 10, as described on page 9/10.

In the machine data P0 the following blocs must be inserted:

N790 X19: Servomode on, limit switches disabled.
N813 X1: X-axis activated

After that you change to the MANUAL MODE, select X+, and START. By opening the feedratepotentiometer for a short time, a small lag distance is generated in the DILAG resulting in a small output voltage going to the servoamplifier. The servomotors starts moving with the corresponding speed. Becaus there is no feedback through the encoder, the lag distance will stay constant and the servomotor als will move at constant speed. The lag distance can be displayed by pushing the key „2“.
It can vary between +/- 1600 increments. At higher values an error message „LAG ERROR“ will be displayed.

If this test is finished successfully, the motor can be fixed to the mechanical axis. Then the encoder is wired according X11, page 9/10. If the encoder is wired correctly, the X-axis can be moved in the MANUAL MODE under control of the feedratepotentiometer.

As next the steps/mm are adjusted with the machinedata N706 and N707 (page 6/1). After that N700 - N704 can be selected with the help of the following program:

```
P1    N1 G00 X100
      N2 G04 H1
      N3 G00 X-100
      N4 G04 H1
      N5 G20 P1
```

This program can be started in the AUTOMATIC MODE. One can observe the effect of the different machine data on the run of the axis. The goal of this adjustment is to get a smooth movement of the axis at all speeds adjustable with the feedratepotentiometer.

If the feedratepotentiometer is fully opened, the lag distance should display between 1000 and 1400 increments (to be viewed by pushing the key „2“), with a feedrate of 0, the lag distance also should be 0. If not, it can be adjusted with the OFFSET trimmer of the servoamplifier until the lag distance oscillates slowly between 0 and 1.

7.4 STEPPING MOTORS

If the stepper drivers were purchased from our company, you must only connect the stepping motors to the corresponding outputs MOTOR X, MOTOR Z. The pin connections can be found on page 9/4, X3-I -SM SIGNAL. We deliver 3-phase stepping motors. After that the connector X3-I of the CNC is wired on the stepper driver unit.

If the stepper driver was purchased from an other company, the the connector X3 (page 9/4) must be wired, first only the X-motor. For that pins 1, 2, 6 must be used.

In P0 the following bloc is inserted: N790 X3.
Then the stepping mode is activated, the limit switches are disabled. You go into MANUAL MODE, select X+ or X- and then START. The X-axis can be moved under control of the feedrate potentiometer.

HINT: to execute this test, you only have to program P0 with N790 X3, nothing else!

As next, the steps/mm are adjusted with N706 and N707 (page 6/1). After that N700 - N704 can be selected with the help of the following programm:

```
P1   N1 G00 X100
     N2 G00 X-100
     N3 G20 P1
```

This program can be started in the AUTOMATIC MODE. You can observe the effect of the different machine data on the run of the axis. The goal of this adjustment is to get a troublefree movement of the stepping motor without loss of steps at all speed adjustable with the feedrate potentiometer.

7.5 LIMIT SWITCHES

The limit switches will be connected to X9 for a servo system (page 9/7) or X1 for a stepping system (page 9/9). As always, first only the X-axis will be wired. An external 24V- supply will be used for powering the limit switches. The limit switch which will be activated when the axis is moving in the postive direction, will be connected to X9/8 and X9/21and the other one to X9/4 and X9/17. At this moment you should be aware if you are connecting closers or openers as limit switches. A closer is switch outputting 24V when activated. An opener outputs 0V when activated. Normally openers are used because they simulate an activation if a wire has broken.
If one does not know if the switch is opener or closer, the output can be measured between X9/8 and X9/21 (f.e. for a servo system) with a voltmeter. If the voltage is +24V and the switch is not activated, there is an opener.

According to the above result, the machinedata N790 must be corrected. Add 256 for the X+ limitswitch and 512 for X- limitswitch. With a steepingmotor system we have a total of 771, with servo-motors 878. If closers were used we will have to add 1024 plus 2048. See also page 6/2.

If the x axis is moved slowly in the MANUAL MODE to a limitswitch, it will stop immediatly when the limit switch is reached and an error message will be displayed. The axis can only move back in the opposite direction

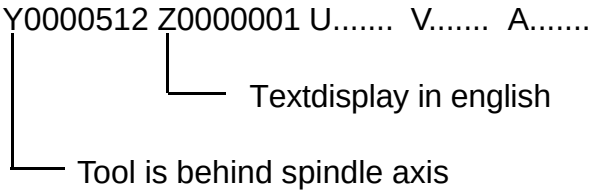
Explanation of machine data P0000 of a sample lathe.

P0000 MACHINEDATA STEPPING MOTORS

```
N700    X0004000 Z0006000    ;Maximal feedrate for X and Z
N701    X0000050 Z0000050    ;Starting feedrate
N702    X0000050 Z0000050    ;Stopping feedrate
N703    X0000100 Z0000100    ;acceleration of movement
N704    X0000100 Z0000100    ;deceleration of movement

N706    X0000500 Z0001250    ;in X there is 500 steps for 2 mm,
N707    X0000002 Z0000006    ;in Z there is 1250 steps for 6 mm

N710    X0004000 Z0004000    ;feedrate for G74
N790    X0000771 Z0000771    ;001: main axis
                             ;002: linear axis
                             ;256: limit switch at the positive end of axis present
                             ;512: limit switch at the negative end of axis present
```




```
N903    X0000256 Y0000256 Z..... U.....  V.....  A.....
          ;IO1 and IO2 are not reset when changing to MENU

N904    X..... Y..... Z..... U.....  V.....  A0000036
          ;G36 is presented in MANUAL INPUT

N905    X0000128 Y..... Z..... U.....  V.....  A.....
          ;when G95 is active, the feedrate is proportional to speed of the spindle
          and not to the programmed spindlespeed.

N921    X0000640 Y0000240 Z0000114 U0000085  V.....  A.....
          ;resolution of LC-display

N925    X0001024 Y0000001 Z..... U.....  V.....  A.....
          ;1024 pulses for 1 rotation of spindlemotor
```

P0074 Reference Point

P0074 is used to move the axes to the reference point when the CNC is switched on. In N003 the actual diameter, in N012 the distance to the spindle should be input.

```
N001↓80 Enter the actual diameter
N002↓80 in N003 X
N003 G74 X+.670,200 Z.....
N004 G13 M0246 M4050 M0256 M.... M....
N010↓80 Enter the actual distance to the
N011↓80 spindle in bloc N012 Z,,,
N012 G74 X..... Z+.215,100
N013 G13 M0247 M4050 M0257 M.... M....
```

P9900 TOOL TABLE

P9900 is the tooltable. It must be present in memory when the tool is invoked with G11 T... or with G36 T.... T001 is the reference tool, it must be the longest tool of all tools physically used. It should have the length of 0 in X and Z.
The first pair of X,Z of the remaining tools must contain the difference to the tool T001. The second pair of X,Z are used to correct the tool when the lenght changes during work. In fact both pairs of X,Z are added together in order to generate the actual toollengthcompensation.

```
T001   X+...0,000 Z+...0,000 X+...0,000  Z+...0,000  R+..0,000  O000000
T002   X+...2,000 Z+...0,000 X+...0,000  Z....0,000  R+...0,000  O000000
T003   X+...3,000 Z+...0,000 X+...0,000  Z....0,000  R+...0,000  O000000
T004   X+...4,000 Z+...0,000 X+...0,000  Z....0,000  R+...0,000  O000000
T005   X+...5,000 Z+...0,000 X+...0,000  Z....0,000  R+...0,000  O000000
T006   X+...6,000 Z+...0,000 X+...0,000  Z....0,000  R+...0,000  O000000
```

P9936 Tool Change

P9936 is sample tool change programm which can be adapted to the needs of the user. When the user programms G36 F100 S2 T3 M3 he wants in this case to have a feedrate of 100, the spindle gear number 2, the tool number 3 and the spindle on clockwise. When this G36 is executed, P9936 is invoked and the values of F,S,T,M are put into the CNC registers #80, #81 #82, #83.
#90 contains a value indicating wether F or S or T or M have been programmed in G36.

```
N001 ↓00    #071 =      @+...0,180     ;S-gear 1 from 0 to 180 rpm
N002 ↓00    #072 =      @+...0,500     ;S-gear 2 from 181 to 500 rpm
N003 ↓00    #073 =      @+...1,000     ;S-gear 3 from 501 to 1000 rpm
N004 ↓00    #074 =      @+...1,800     ;S-gear 4 from 1001 to 1800 rpm

;test if F has been programmed
N010 ↓00    #092 =      @+...0,128
N011 ↓18    #092 = AND   #090
N012 ↓50   (JUMP ZER TO) N0020       ;F was not programmed
N013 G11 F..#080 S..... T....  M.... ;use the programmed F for the next movement

;test if S has been programmed
N020 ↓00    #092 =      @+...0,064
N021 ↓18    #092 = AND   #090
N022 ↓54   (JUMP NZ  TO) N0800       ;S was programmed

;test if T has been programmed
N030 ↓00    #092 =      @+...0,032
N031 ↓18    #092 = AND   #090
N032 ↓54   (JUMP NZ  TO) N0060       ;T was programmed

;test if M has been programmed
N040 ↓00    #092 =      @+...0,016
N041 ↓18    #092 = AND   #090        M was not programmed
N042 ↓50   (JUMP ZER TO) N0990       ;end of P9936

;If M03 or M04 was programmed, execute the function with G11 and
;then wait in N050 for the input 5 of IO2. This input should be active
;when the spindle motor is working.

N043 G11 F..... S..... T....  M#083
N044 ↓02    #082 = #083 - @+...0,003
N045 ↓50   (JUMP ZER TO) N0050
N046 ↓02    #082 = #083 - @+...0,004
N047 ↓54   (JUMP NZ  TO) N0990
N050 G13 M0265 M.... M.... M.... M....
N051 ↓53   (JUMP    TO) N0990
```

```
;T was programmed, make tool change!
N060 ↓04    #080 = #082 / @+..10,000
N061 ↓54   (JUMP NZ  TO) N0065
N062 G22 P....   N0200   W....    CALL PROGRAM
N063 G11 F..... S..... T#082   M....
N064 ↓53   (JUMP     TO) N0040
N065 ↓10    #089 = COPY   #082
N066 ↓10    #082 = COPY   #080
N067 G22 P....   N0200   W....    CALL PROGRAM
N068 G11 F..... S..... T#089   M....
N069 ↓53   (JUMP     TO) N0040

N100 ↓02    #080 = #081 - @+...0,001
N101 ↓54   (JUMP NZ  TO) N0110
N102 G13 M0251 M0252 M0253 M0254 M....
N103 G13 M0241 M0243 M0261 M0263 M....
N104 ↓53   (JUMP     TO) N0190

N110 ↓02    #080 = #081 - @+...0,002
N111 ↓54   (JUMP NZ  TO) N0120
N112 G13 M0251 M0252 M0253 M0254 M....
N113 G13 M0242 M0243 M0262 M0263 M....
N114 ↓53   (JUMP     TO) N0190

N120 ↓02    #080 = #081 - @+...0,003
N121 ↓54   (JUMP NZ  TO) N0130
N122 G13 M0251 M0252 M0253 M0254 M....
N123 G13 M0241 M0244 M0261 M0264 M....
N124 ↓53   (JUMP     TO) N0190

N130 ↓02    #080 = #081 - @+...0,004
N131 ↓54   (JUMP NZ  TO) N0140
N132 G13 M0251 M0252 M0253 M0254 M....
N133 G13 M0242 M0244 M0262 M0264 M....
N134 ↓53   (JUMP     TO) N0190

N140 ↓02    #080 = #081 - @+...0,000
N142 ↓54   (JUMP NZ  TO) N0150
N143 G13 M0251 M0252 M0253 M0254 M....
N144 ↓53   (JUMP     TO) N0190
```

```
N150 J51   (JUMP POS TO) N0190
N153 J96    #004  #085  #...  #...  #...  #...  #...
N154 J01    #081 = #086 + @+...0,001
N155 J02    #080 = #081 - @+...0,005
N156 J52   (JUMP NEG TO) N0159
N157 J00    #081 =       @+...0,001
N159 J53   (JUMP    TO) N0100
N190 G11 F..... S..#081 T....  M....
N199 J53   (JUMP    TO) N0030

N200 J02    #081 = #082 - @+...0,001
N201 J54   (JUMP NZ  TO) N0220

;change to tool #1, jump to N490 if tool #1 already present
N202 G23 P....  N0490  W....  M0161
N203 G13 M0141   M0161    M0151 M.... M....
                                  toolchanger off
                          , wait until tool #1 is present,
                 ;Toolchanger forward

N204 J53   (JUMP    TO) N0480

N220 J02    #081 = #082 - @+...0,002
N221 J54   (JUMP NZ  TO) N0240
;change to tool #3
N222 G23 P....  N0490  W....  M0162
N223 G13 M0141 M0162 M0151 M.... M....
N224 J53   (JUMP    TO) N0480
N240 J02    #081 = #082 - @+...0,003
N241 J54   (JUMP NZ  TO) N0260
;change to tool #3
N242 G23 P....  N0490  W....  M0163
N243 G13 M0141 M0163 M0151 M.... M....
N244 J53   (JUMP    TO) N0480

N260 J02    #081 = #082 - @+...0,004
N261 J54   (JUMP NZ  TO) N0280
;change to tool #4
N262 G23 P....  N0490  W....  M0164
N263 G13 M0141 M0164 M0151 M.... M
N264 J53   (JUMP    TO) N0480
```

```
N280 ↓02    #081 = #082 - @+...0,005
N281 ↓54   (JUMP NZ  TO) N0300
;change to tool #5
N282 G23 P....   N0490   W....   M0165
N283 G13 M0141 M0165 M0151 M.... M....
N284 ↓53   (JUMP    TO) N0480

N300 ↓02    #081 = #082 - @+...0,006
N301 ↓54   (JUMP NZ  TO) N0320
;change to tool #6
N302 G23 P....   N0490   W....   M0166
N303 G13 M0141 M0166 M0151 M.... M....
N304 ↓53   (JUMP    TO) N0480

N320 G22 P....   N0900   W....    CALL PROGRAM
N321 ↓10    #082 = COPY   #081
N322 ↓53   (JUMP    TO) N0200

;toolchanger backwards, wait for lock-in, wait 0,5 seconds, toolchanger off
N480 G13 M0142 M0167 M.... M.... M....
N481 G04 H+...0,500 DWELL
N482 G13 M0152 M.... M.... M.... M....

;End of toolchange
N490 ↓53   (JUMP    TO) N0990

 ;S was programmed
N800 G13 M9000 M.... M.... M.... M....
N801 ↓02    #080 = #081 - @+...0,004
N802 ↓51   (JUMP POS TO) N0810
N803 G22 P....   N0100   W....    CALL PROGRAM
N804 G13 M9255 M.... M.... M.... M....
N805 ↓53   (JUMP    TO) N0030
N810 ↓02    #080 = #081 - @.....#071
N811 ↓51   (JUMP POS TO) N0820
N812 ↓03    #080 = #081 * @+...0,255
N813 ↓04    #080 = #080 / @.....#071
N814 ↓00    #081 =       @+...0,001
N815 ↓53   (JUMP    TO) N0890
N820 ↓02    #080 = #081 - @.....#072
N821 ↓51   (JUMP POS TO) N0840
N822 ↓03    #080 = #081 * @+...0,255
N823 ↓04    #080 = #080 / @.....#072
N824 ↓00    #081 =       @+...0,002
N825 ↓53   (JUMP    TO) N0890
```

```
N840 ↓02   #080 = #081 - @.....#073
N841 ↓51   (JUMP POS TO) N0860
N842 ↓03   #080 = #081 * @+...0,255
N843 ↓04   #080 = #080 / @.....#073
N844 ↓00   #081 =      @+...0,003
N845 ↓53   (JUMP    TO) N0890
N860 ↓02   #080 = #081 - @.....#074
N861 ↓52   (JUMP NEG TO) N0863
N862 ↓00   #081 =      @.....#074
N863 ↓03   #080 = #081 * @+...0,255
N864 ↓04   #080 = #080 / @.....#074
N865 ↓00   #081 =      @+...0,004
N890 ↓01   #079 = #080 + @+...9,000
N891 G22 P....  N0100  W....  CALL PROGRAM
N892 G13 M#079 M.... M.... M.... M....
N893 ↓53   (JUMP    TO) N0030
```

The routine starting with N900 looks what tool is active at the moment and returns the active tool number in #080. In #081, the next available toolnumber is returned. When input 5 and 6 of IO1 are both active (=connected to 24V), then #081 can contain the values 0,001 to 0,004, else 0,001 to 0,006.

```
N900 ↓00    #080 =      @+...0,000
N901 ↓00    #081 =      @+...0,001
N910 G23 P....  N0914  W....  M0171 ;jump to N914 if input 1 not active
N911 ↓00    #080 =      @+...0,001
N912 ↓00    #081 =      @+...0,002
N913 ↓53  (JUMP    TO) N0950
N914 G23 P....  N0918  W....  M0172
N915 ↓00    #080 =      @+...0,002
N916 ↓00    #081 =      @+...0,003
N917 ↓53  (JUMP    TO) N0950
N918 G23 P....  N0922  W....  M0173
N919 ↓00    #080 =      @+...0,003
N920 ↓00    #081 =      @+...0,004
N921 ↓53  (JUMP    TO) N0950
N922 G23 P....  N0930  W....  M0174
N923 ↓00    #080 =      @+...0,004
N924 ↓00    #081 =      @+...0,005
N925 ↓00    #081 =      @+...0,001
N926 G22 P....  N0960  W....   CALL PROGRAM
N927 ↓50  (JUMP ZER TO) N0950
N928 ↓00    #081 =      @+...0,005
N929 ↓53  (JUMP    TO) N0950
N930 G22 P....  N0960  W....   CALL PROGRAM
N931 ↓50  (JUMP ZER TO) N0990
N940 G23 P....  N0944  W....  M0175
N941 ↓00    #080 =      @+...0,005
N942 ↓00    #081 =      @+...0,006
N943 ↓53  (JUMP    TO) N0950
N944 G23 P....  N0950  W....  M0176
N945 ↓00    #080 =      @+...0,006
N946 ↓00    #081 =      @+...0,001
N950 G11 F..... S..... T#080  M....
N953 ↓53  (JUMP    TO) N0990
N960 ↓84    #016  #001  #088  #064  #001  #...  #...
N961 ↓00    #089 =      @+...0,048
N962 ↓18    #088 = AND    #089
N990 ↓80 END
```

P9974 Home Position

P9974 is called when in the MANUAL MODE is pushed. P9974 is programmed so that a „HOME POSITION“ can be memorized and moved to from any point.

```
N001 ↓80 xx set „HOME“ POSITION
     N002 ↓80 xx move „HOME“ POSITION

     ;Display text N001 and  N002 on the screen
     N050 ↓00    #000 =       @+...1,254
     N051 ↓83    #001  #...  #...  #...  #...  #...  #...
     N052 ↓00    #040 =       @+..41,013
     N053 ↓00    #043 =       @+..49,372
     N054 ↓00    #044 =       @+...0,027
     N055 ↓82    #000  #...  #...  #...  #...  #...  #...
     N060 ↓00    #000 =       @+...7,910
     N061 ↓83    #002  #...  #...  #...  #...  #...  #...
     N062 ↓00    #040 =       @+..41,013
     N063 ↓00    #043 =       @+..56,028
     N064 ↓00    #044 =       @+...0,086
     N065 ↓82    #000  #...  #...  #...  #...  #...  #...
     ;wait for a key to be pushed
     N102 ↓89    #080  #...  #...  #...  #...  #...  #...
     N103 ↓50   (JUMP ZER TO) N0102

     N104 ↓02    #081 = #080 - @+...0,030
     N105 ↓50   (JUMP ZER TO) N0700
     N106 ↓02    #081 = #080 - @+...0,015
     N107 ↓54   (JUMP NZ  TO) N0900

     ;key „INPUT“  was pushed
     N108 ↓96    #005  #082  #001  #...  #...  #...  #...
     N120 ↓00    #081 =       @+..29,184
     N123 ↓84    #000  #002  #082  #081  #002  #000  #000
     N124 ↓00    #081 =       @+..29,188
     N125 ↓84    #000  #002  #083  #081  #002  #000  #000
     N130 ↓53   (JUMP    TO) N0900

     ;Key „REFERENZ PUNKT“ was pushed
     N700 G90 ABSOLUTE INPUT
     N710 ↓00    #085 =       @+..29,184
     N711 ↓84    #000  #001  #086  #085  #002  #...  #...
     N713 ↓01    #086 = #086 + @.....#086
     N715 G00 X.....#086 Z.....#087 RAPID TRAVERSE
     N716 ↓53   (JUMP    TO) N0900
```

```
;wait for all axes to have stopped
N900 G13 M0019 M.... M.... M.... M....
;switch back to MANUAL MODE
N901 ↓00    #040 =      @+..42,339
N902 ↓00    #044 =      @+...0,025
N903 ↓82    #000  #...  #...  #...  #...  #...  #...
```

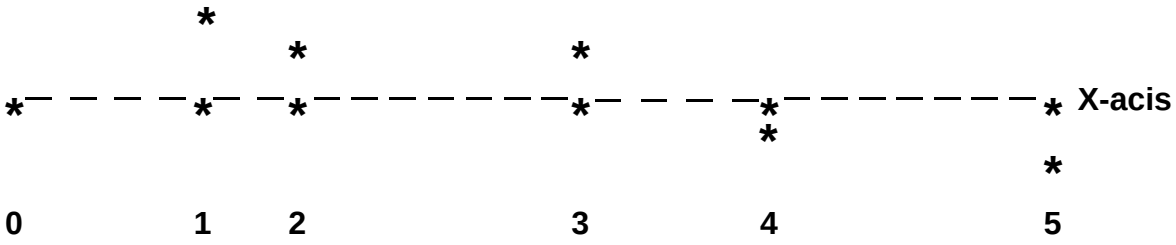
P9999 Autostart

P9999 is executed when the CNC is switched on.

```
N002 [01    #102 = #102 + @.....#102
;#102 and #103 contain the actual value of X and Z from before the
;CNC was switched off. #102 must be doubled because of diameter
;programming of x-axis.
N004 G92 X.....#102 Z.....#103 SET ACTUAL VALUE
;P9936 N900 looks what tool is active.
N005 G20 P9936   N0900   JUMP PROGRAM
```

Ballscrew errorcompensation

This compensation is used to correct errors in the ballscrew. The axis is f.e. moved to positiv limit switch with G74, the actual value is set to 0.
This is the beginning of a compenstion table with the actual value equal to 0 and the measured value also 0. Now the axis must be measured and the points where the pitch changes are noticed. After that one has f.e. the following diagram:



These points then are memorized in the machine data P0 starting with N100 :

```
P0
N100  X  ...0,000    R ....0,000          ;Anfang der Tabelle
N101  X-120,000      R-123,000            ;1. point,    error   -3mm
N102  X-180,000      R-182,000            ;2. point,    error   -2mm
N103  X-370,000      R-372,000            ;3. point,    error   -2mm
N104  X-460,000      R-459,000            ;4. point,    error   +1mm
N105  X-570,000      R-568,000            ;5. point,    error   +2mm
```

Accordingly, the tables for Z start at N200. For activating the table, the limit switches wich were used for generating the table, must be moved to with G74 everytime the CNC is switched on. After that the compensation is active. When generating the table the first time, the bloc N699 in P0 must **NOT** be programmed. The compensation always starts after moving off the limit switch or after the reference pulse of the encoder.
There are 32 table points allowed for each axis.

ATTENTION:

- When connecting the CNC, the national security requirements must be fullfilled.

- Especially, the CNC must be switched off in an emergency situation.

- The power supply must be 230V +/- 5%.

-The CNC - housing must not be totally closed. Air circulation must be available.

! ! ! ! ! ! ! ! ! ! ! ! ! !

-To the adherence to the EMV - we offer metallized plug housings orcomplete sets of cables with metallized plug housings to regulationsfor all our NCC-cControls.
If the cable connections are by the usermade, likewise metallized plug housings must be used and keptfollowing regulations:

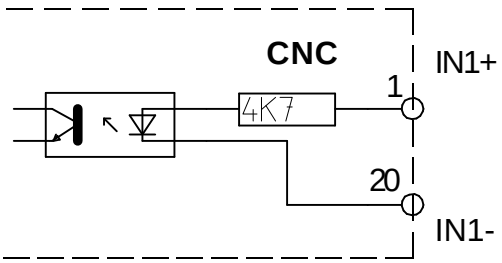
- All connections to the CNC must be shielded, the shield must be firmly connected to the metallic case of the connector.

X1 Input

SUB-D 37pol female

pin	signal	pin	signal	M-function	
1	IN1+	20	IN1-	M161	(TW01)
2	IN2+	21	IN2-	M162	(TW02)
3	IN3+	22	IN3-	M163	(TW03)
4	IN4+	23	IN4-	M164	(TW04)
5	IN5+	24	IN5-	M165	(TW05)
6	IN6+	25	IN6-	M166	(TW06)
7	IN7+	26	IN7-	M167	(TFIN)
8	IN8+	27	IN8-	M168	External interrupt (N905X)
9	IN9+	28	IN9-	M261	(S10A)
10	IN10+	29	IN10-	M262	(S20A)
11	IN11+	30	IN11-	M263	(S30A)
12	IN12+	31	IN12-	M264	(S40A)
13	IN13+	32	IN13-	M265	(MFIN)
14	IN14+	33	IN14-	M266	
15	IN15+	34	IN15-	M267	
16	IN16+	35	IN16-	M268	
17		36			
18		37			
19					

Internal diagram of **inputs IN1 to IN16,** **Input voltage 20-30 V**



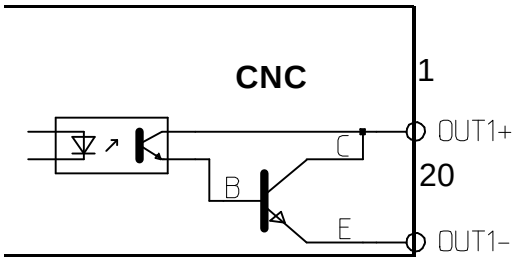
X2 Output

SUB-D 37pol male

pin	signal	pin	signal	M-function	* (f.e.)
1	OUT1+	20	OUT1-	M141	*(TDZ Toolchanger forward)
2	OUT2+	21	OUT2-	M142	*(TDZ Toolchanger backward)
3	OUT3+	22	OUT3-	M143	
4	OUT4+	23	OUT4-	M144	
5	OUT5+	24	OUT5-	M145	
6	OUT6+	25	OUT6-	M146	
7	OUT7+	26	OUT7-	M147	
8	OUT8+	27	OUT8-	M148	(Lubrication pulse) N906A
9	OUT9+	28	OUT9-	M241	*(S10)
10	OUT10+	29	OUT10-	M242	*(S20)
11	OUT11+	30	OUT11-	M243	*(S30)
12	OUT12+	31	OUT12-	M244	*(S40)
13	M03+	32	M03-	M03	
14	M04+	33	M04-	M04	
15	M05+	34	M05-	M05	
16	M08+	35	M08-	M08	
17	M10+	36	M10-	M10	
18	SPEED+	37	SPEED-		(0-10V corresponding to programmed spindle speed S in SM version)
19	---------				

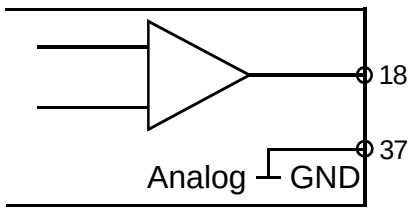
*<u>Example for a toolchanger</u>

Internal diagram of outputs:



Outputs 30V / 0,5A max.

Example for speed output



X3-I SM - SIGNAL (output voltage 5V TTL, for stepping system)

pin	signal	pin	signal	SUB-D 15pol. female
1	GND	9	TAKT Y	
2	R/L X	10	TAKT Z	
3	R/L Y	11	TAKT U	
4	R/L Z	12		
5	R/L U	13		
6		14		
7		15		
8	TAKT X			

X3-II SM - SIGNAL (output voltage 5V TTL, for stepping system)

pin	signal	pin	signal	SUB-D 15pol. female
1	GND	9	TAKT A	
2	R/L V	10	TAKT B	
3	R/L A	11	TAKT C	
4	R/L B	12		
5	R/L C	13		
6		14		
7		15		
8	TAKT V			

X4 EXT SYNC FOR G33 (for stepping system)

pin	signal	pin	signal	SUB-D 9pol. female
1	+5V INTERNAL	6	UA2- (B*)	**max. 60kHz**
2	0V INTERNAL	7	UA0+ (C)	
3	UA1+ (A)	8	UA0- (C*)	
4	UA2+ (B)	9		
5	UA1- (A*)			

In P0 N900A, this axis must be programmed as spindle axis with the value 3.

The encodersignals Ua0, Ua1, Ua2, Ua0*, Ua1*, Ua2* are used for synchronizing the axes with the spindle, so that threading (G33) becomes possible.

For testing G33, the following program is used:

```
N1  G11     S200     M03        ; spindle on
N2  G91
N3  G33     Z -20    K 1  J 1   ; wait for the reference pulse of the encoder
                                  and then make a thread of 20mm
N4  G00     Z 20                ; back to the starting point
```

X5 External hand wheel (Option)

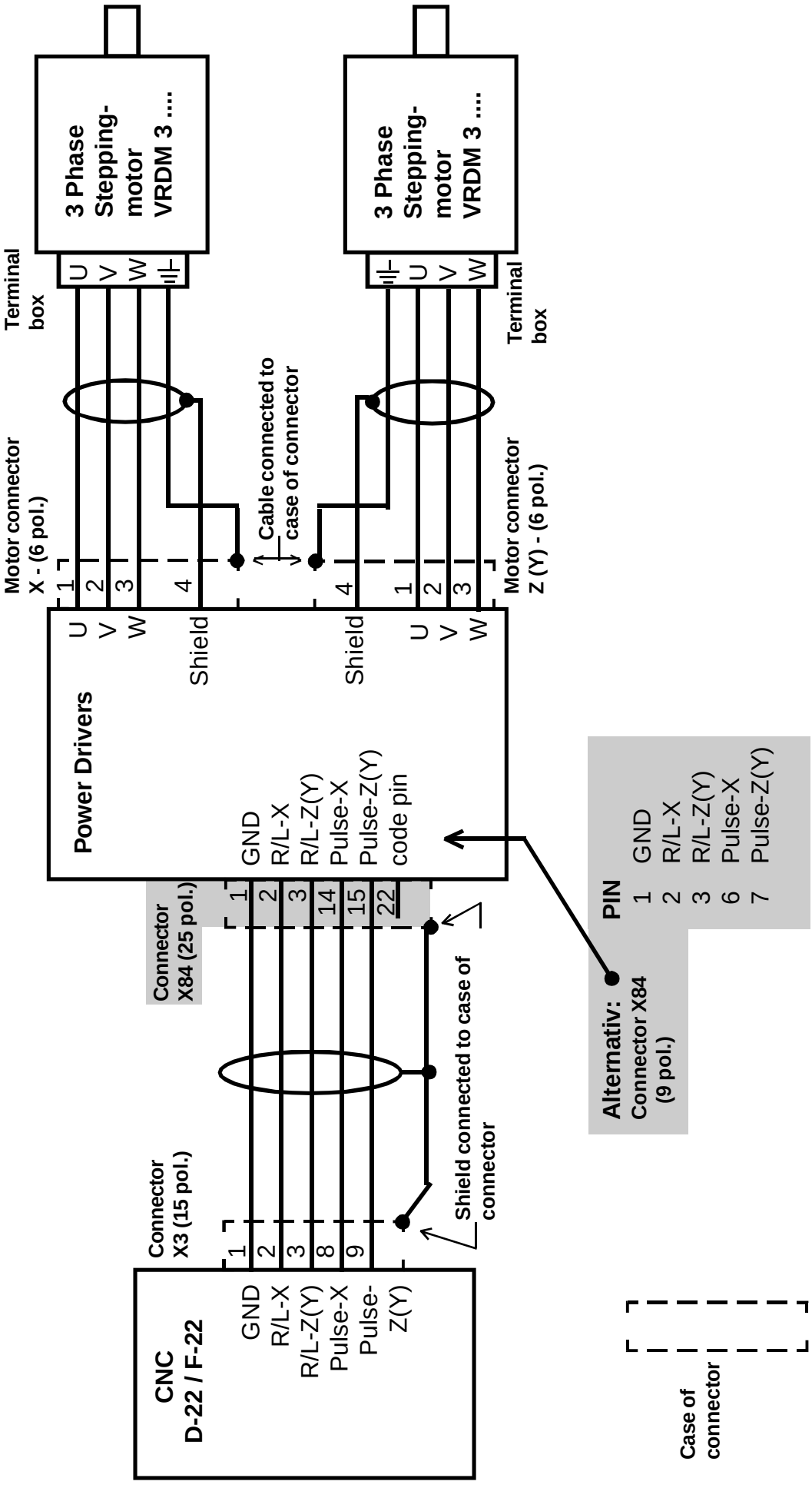
Pin	Signal	Pin	Signal
1	+ Vcc	6	B*
2	GND	7	nc
3	A	8	nc
4	B	9	code pin
5	A*		

SUB-D 9pol. female

X6 V24

CNC			PC (9 pol.)		PC (25pol.)
SUB-D 9 pol male					
pin	**signal**		**pin**	**signal**	
3	TxD	———	2	RxD	3
2	RxD	———	3	TxD	2
5	GND	———	5	GND	7
7	RTS	———	8	CTS	4
8	CTS	———	7	RTS	5

Dataformat:
8 databits, no parity, 1 startbit, 1 stopbit, 9600 baud.
For Xon - Xoff protocol must be P0 N902 X programmed with 64.
Without 64, hardwarehandshake RTS / CTS is aktivated.

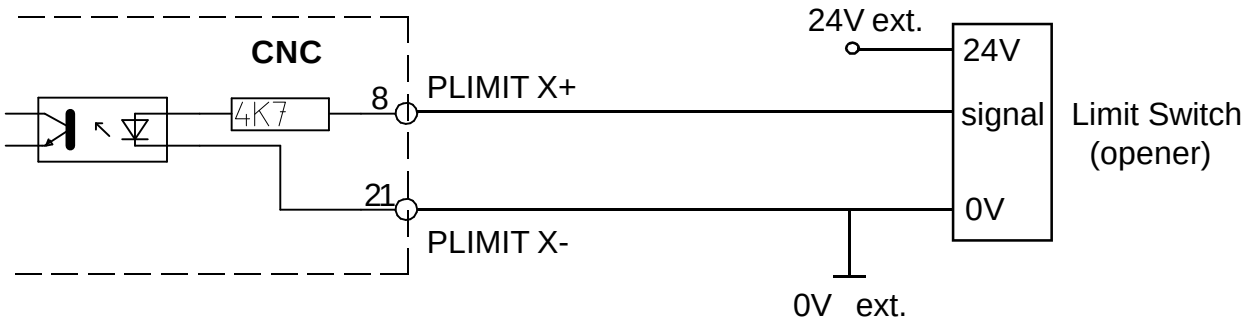

3 Phase Stepping-motor VRDM 3 ....
Terminal box
U
V
W
Motor connector X - (6 pol.)
1
2
3
4
Shield
Cable connected to case of connector
Motor connector Z (Y) - (6 pol.)
Power Drivers
GND
R/L-X
R/L-Z(Y)
Pulse-X
Pulse-Z(Y)
code pin
Connector X84 (25 pol.)
1
2
3
14
15
22
Connector X3 (15 pol.)
8
9
Shield connected to case of connector
CNC D-22 / F-22
Pulse-Z(Y)
Alternativ:
Connector X84 (9 pol.)
PIN
6
7
Case of connector

X9 Limit Switch

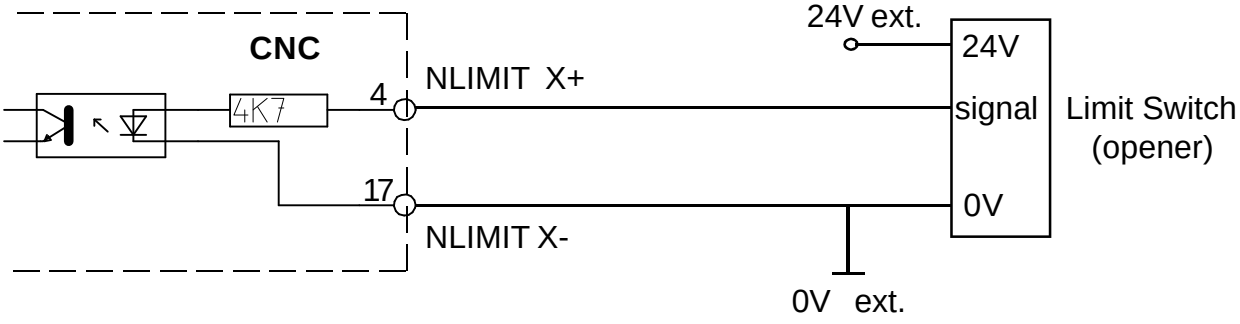
pin	signal	pin	signal
1		14	
2		15	
3	NLIMIT Z +	16	NLIMIT Z -
4	NLIMIT X +	17	NLIMIT X -
5		18	
6		19	
7	PLIMIT Z +	20	PLIMIT Z -
8	PLIMIT X +	21	PLIMIT X -
9		22	
10		23	
11		24	
12		25	
13			

SUB-D 25 pol. female

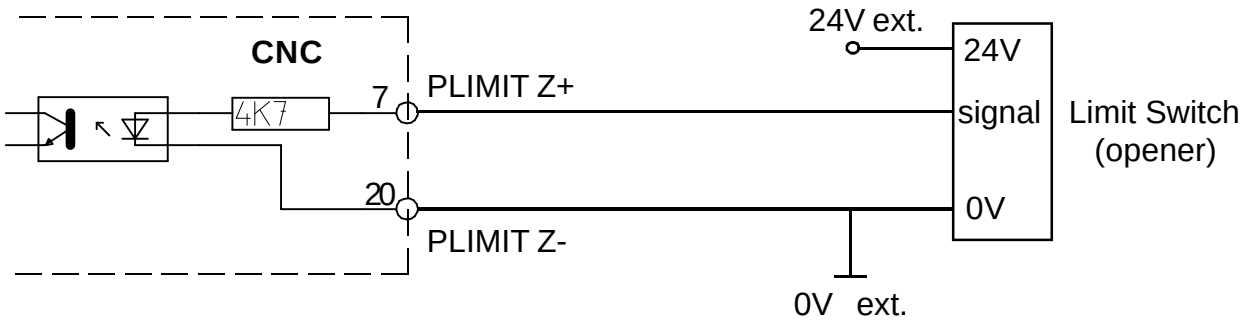
Limit Switch positiv X



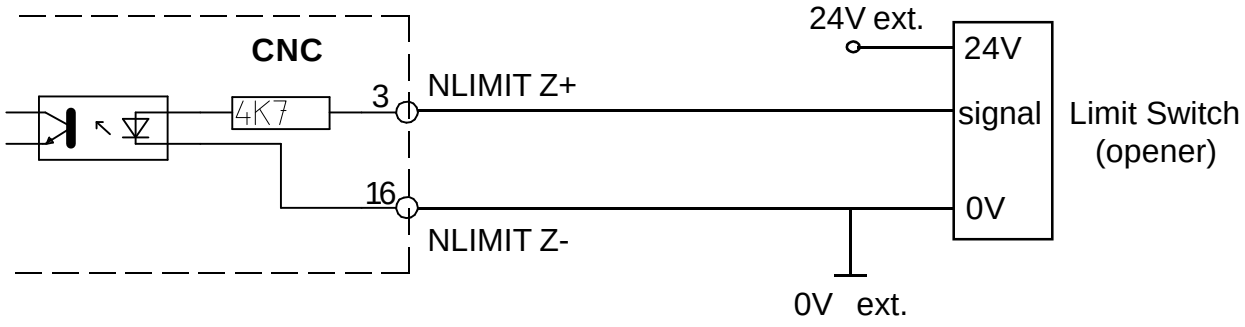
Limit Switch negativ X



Limit Switch positiv Z



Limit Switch negativ Z

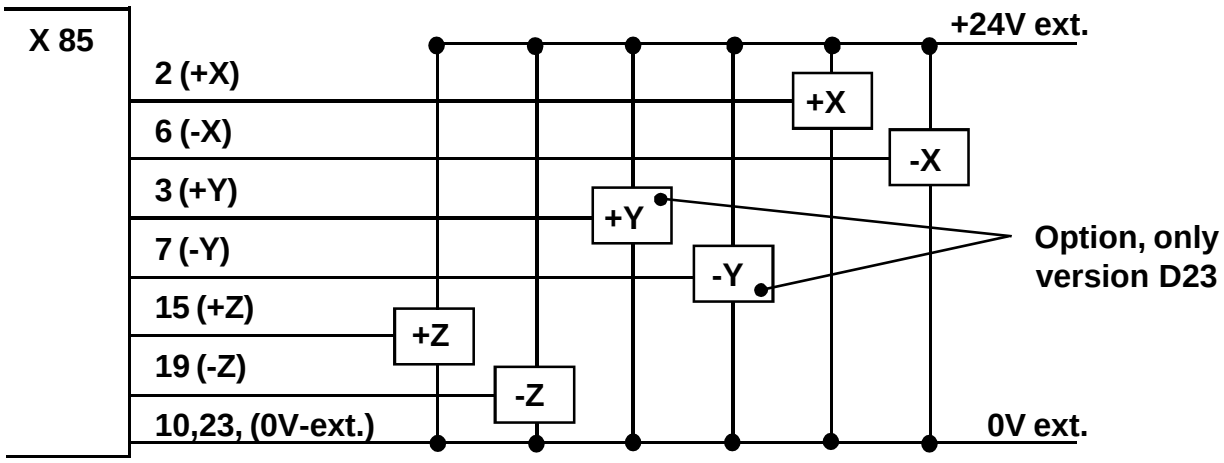


X85 LIMIT SWITCH SUB-D 25pol. female

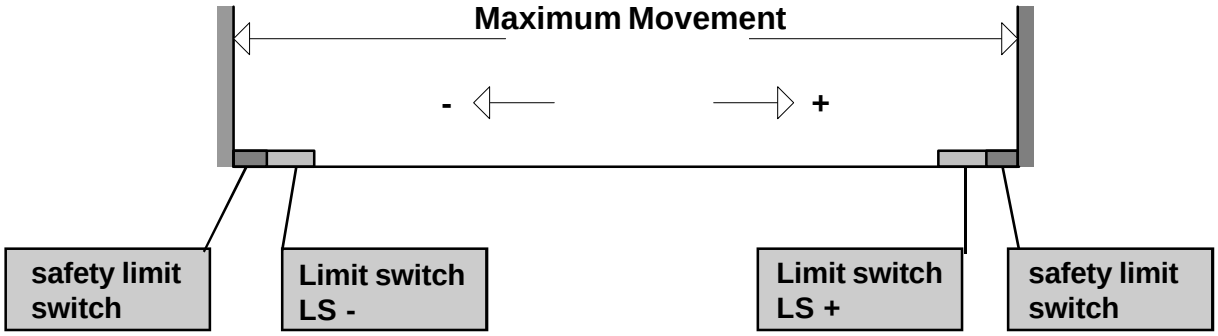
pin	signal	pin	signal
1		14	
2	+X	15	+Z
3	+Y	16	+U
4	+V	17	+A
5	+B	18	+C
6	- X	19	- Z
7	- Y	20	- U
8	- V	21	- A
9	- B	22	- C
10	0 V external	23	0 V external
11	code pin	24	
12		25	
13			

The inputs need 24V, 5mA and are optocoupled. The machinedatum N790 defines for each axis separately if the limitswitches used are opener or closer and if 1 or 2 limit-switches are connected.

Connection for X, Y und Z



If the CNC should fail, no danger causing movement is allowed to result. Therefore, safety limit switches generating an emergency stop with power shut down, should be located after the normal limit switches.



X11-1 / X11-2 SERVO OUTPUT / ENCODER (for servo system)

pin	signal	pin	signal	SUB-D 15 pol. female
1	+5V Internal	9	Motor on +	> Drive Enable
2	0V Internal	10	Motor on -	
3	Ua 1	11	DC +	> Analog Output
4	Ua 2	12	DC -	
5	Ua 1*	13		
6	Ua 2*	14		
7	Ua 0	15	code pin	
8	Ua 0*			

The ouput MOTOR ON is optocoupled and can switch 24V, 20mA.

The servoamplifier must have a **differential input** ± 10V.

The inputs Ua1 - Ua1*, Ua2 - Ua2*, Ua0 - Ua0* are connected to the inputs of an optocoupler. If the encoder is powered externally, it is completely galvanically isolated.
The pins 1 and 2 (+5V and 0V) MUST NOT be connected to an encoder- simulation of a servoamplifier. The encoder simulation must be powered internally from the servoamplifer.
It is recommended not to use the internal 5V power supply at all because of possible electr. noice which could be inducted on the cables.
The amplifier must imediatly be disabled when MOTOT ON is disabled, independently of a feed command on the Analog Output.

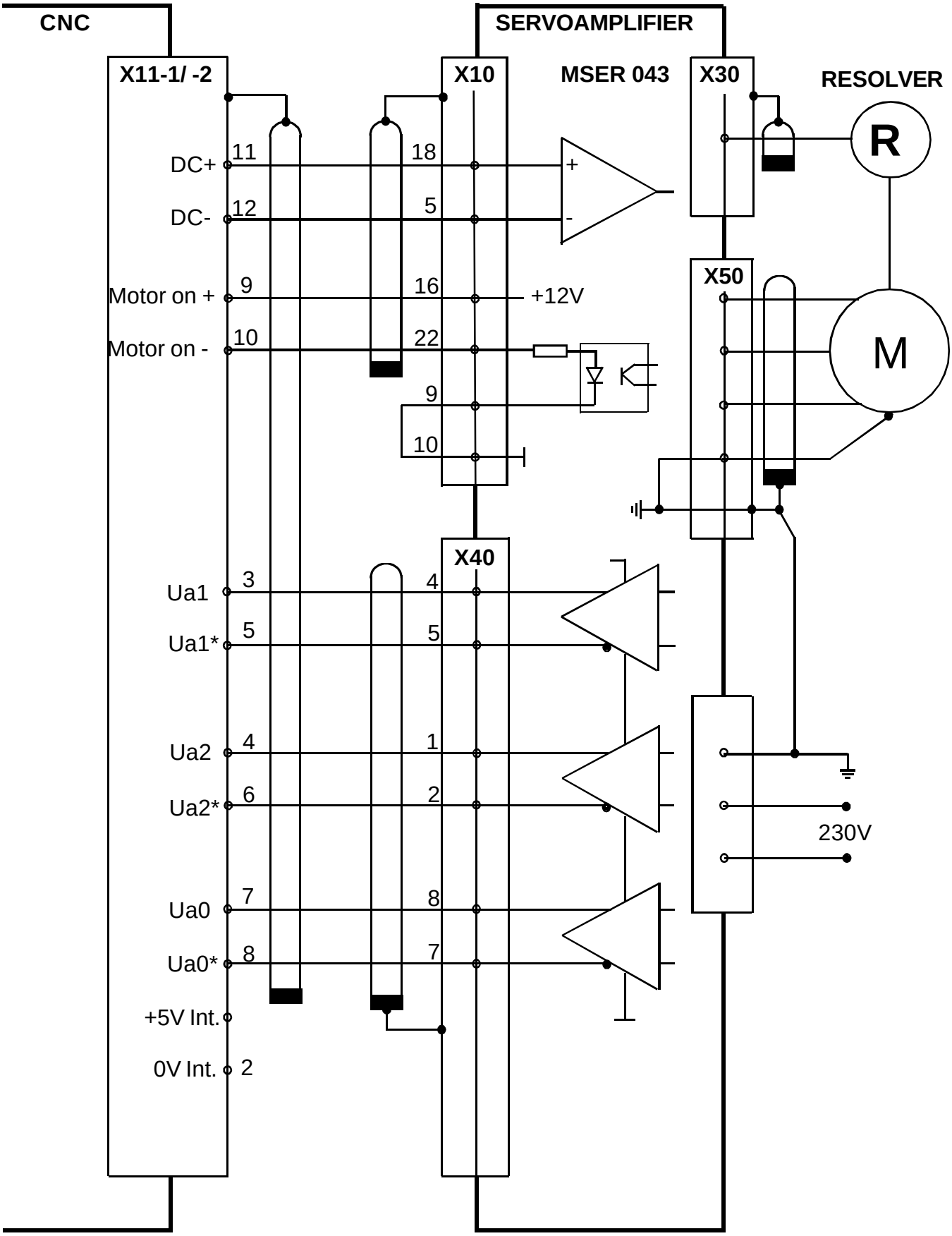
Note: - Use shielded cable. Connect shield to the case of the CNC to the ground connection.
- Use transducer with TTL output!

To invert the counting direction of the measuring system, exchange Ua1 with Ua2 and Ua1* with Ua2*.

Hints for selection of machine data for servo mode:

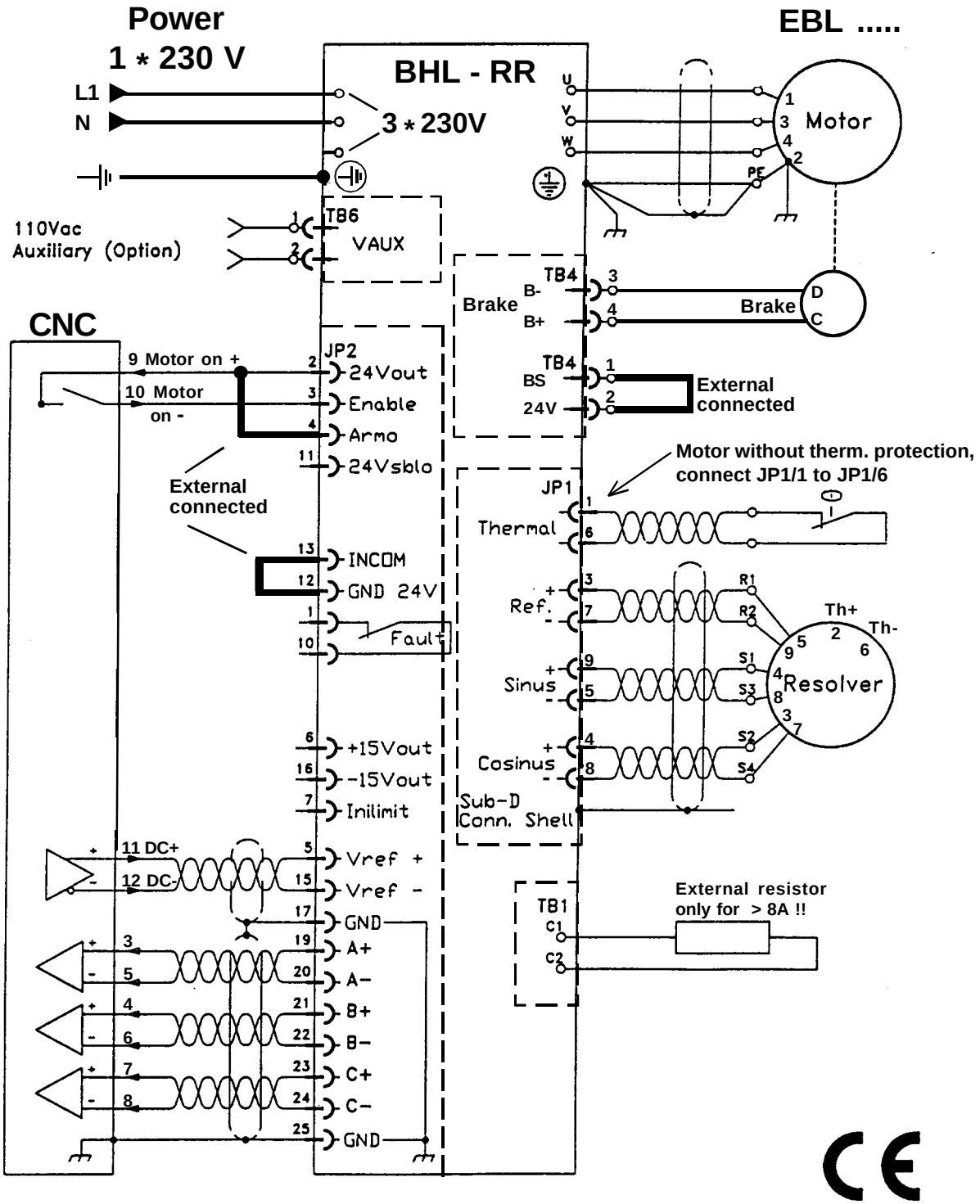
- Switch N790 to servo mode (add the value 16).
  Activate N813X f.e. 3 = 1 + 2 = axes X,Z.
  The axes now can now be moved at low speed, if connector X11.1 (X axis) and X11.2 (Z axis) are correctly cabled.
- Input N700, N706, N707 correctly for each axis.
- By pushing the key „2“ in the MANUAL MODE, the lagerror for each axis is displayed.

CONNECTION OF SERVOAMPLIFIERS TYP MSER 043

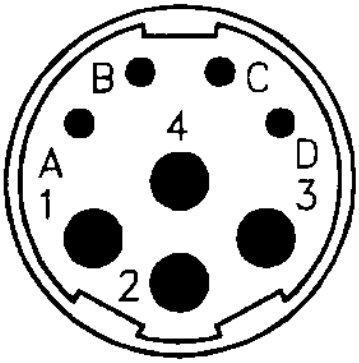


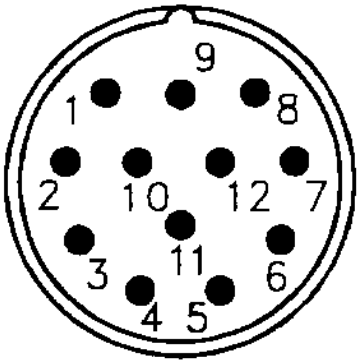
Connection of Servoamplifiers Typ BHL ...

Power supply 1 * 230V or 3 * 230V



Motor - and resolverconnectors : f.e. Engelhardt EBLx - xxx





Motor connector:

1 = phase
4 = phase
3 = phase
2 = Hausing (ground)
A =
B =
C = *(Brake +)
D = *(Brake -)

Resolver connector:

4 = sin +
8 = sin -
3 = cosin +
7 = cosin -
5 = Supply +
9 = Supply -
2 = *(Thermoswitch +)
6 = *(Thermoswitch -)

***(Option)**

When connecting our EBLx - xxx motors at BHL.. amplifiers must be attached the phases as follows:

BHL ...		**EBLx-xxx**
U	---------Phase---------	pin 1
V	---------Phase---------	pin 3
W	---------Phase---------	pin 4

X11-3 Spindlemotor / External sync (G33) (for servo system)

SUB-D 15 pol. female

pin	signal	pin	signal
1	+5V internal	9	Motor on +
2	0V internal	10	Motor on -
3	Ua 1	11	DC +
4	Ua 2	12	DC -
5	Ua 1*	13	
6	Ua 2*	14	
7	Ua 0	15	code pin
8	Ua 0*		

Option :

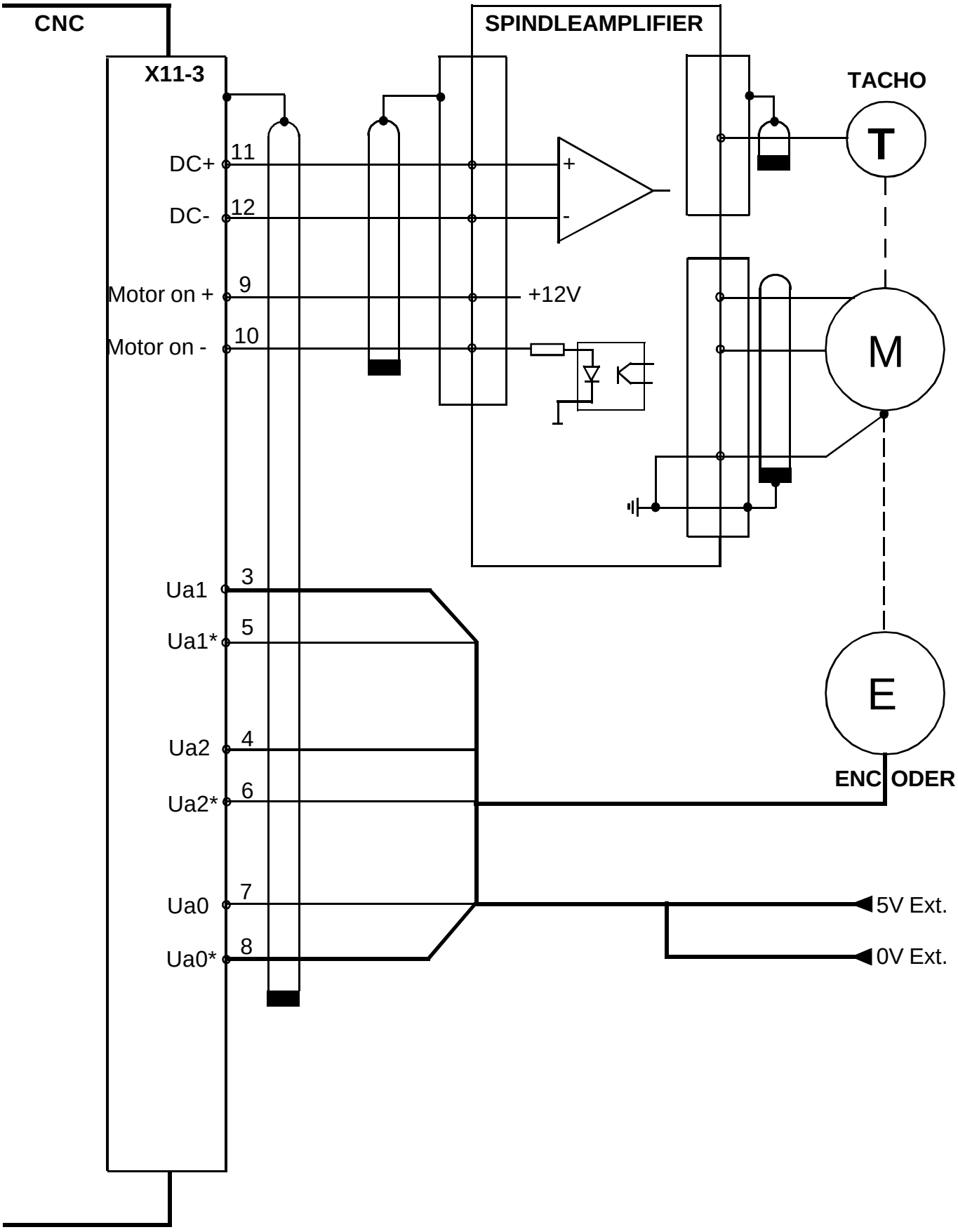
In P0 N900A, this axis must be programmed as spindle axis with the value 3.

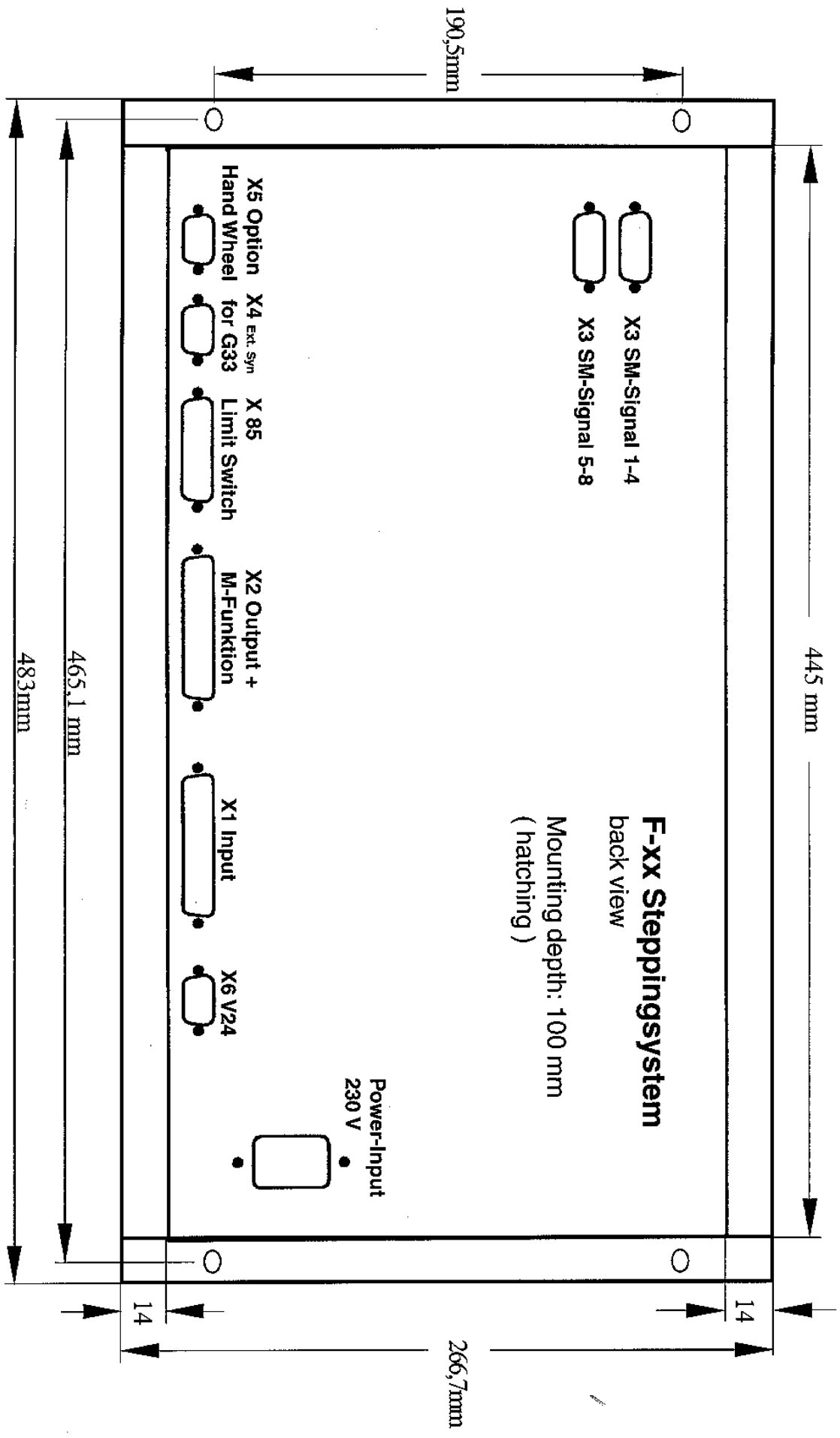
The encodersignals Ua0, Ua1, Ua2, Ua0*, Ua1*, Ua2* are used for synchronizing the axes with the spindle, so that threading (G33) becomes possible.

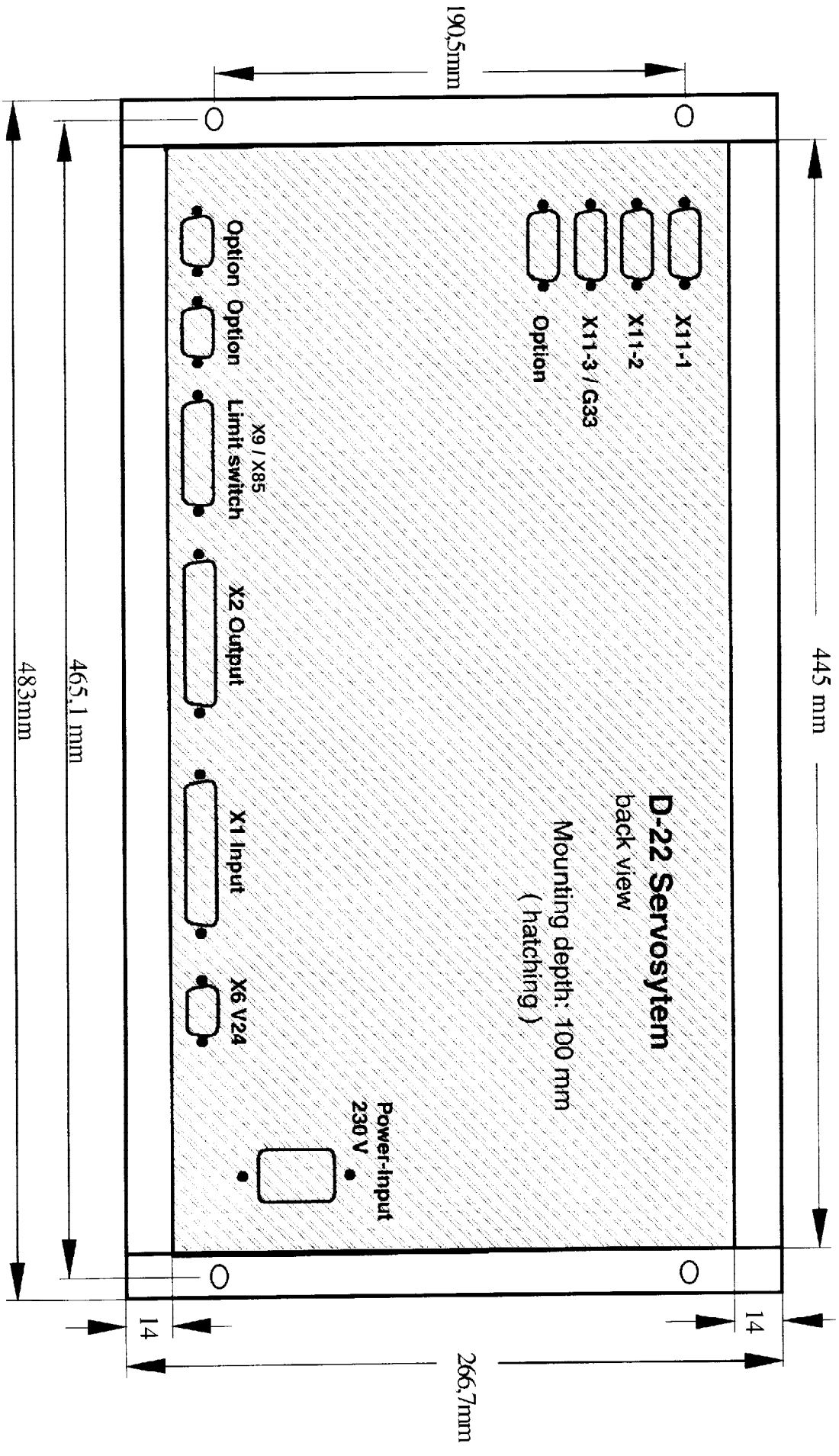
For testing G33, the following program is used:

```
N1  G11     S200     M03          ; spindle on
N2  G91
N3  G33     Z -20    K 1  J 1     ; wait for the reference pulse of the encoder
                                    and then make a thread of 20mm
N4  G00     Z 20                  ; back to the starting point
```

CONNECTION OF SPINDLE MOTOR




190,5mm
445 mm
483mm
465,1 mm
14
14
266,7mm
F-xx Steppingsystem
back view
Mounting depth: 100 mm
(hatching)
X3 SM-Signal 1-4
X3 SM-Signal 5-8
X5 Option Hand Wheel
X4 Ext. Syn for G33
X 85 Limit Switch
X2 Output + M-Funktion
X1 Input
X6 V24
Power-Input 230 V


190,5mm
445 mm
D-22 Servosytem
back view
Mounting depth: 100 mm
(hatching)
X11-1
X11-2
X11-3 / G33
Option
Option
Option
Option
X9 / X85
Limit switch
X2 Output
X1 Input
X6 V24
Power-Input
230 V
483mm
465,1 mm
14
14
266,7mm

SHIELDING PLAN