MANUAL

F - 22 to F - 44

(starting with version 3.05 / 01.07.98)

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. Table of contents

1.1 LIST OF G-FUNCTIONS MILLING

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

G07 CIRCLE WITH ANGLE

G08 ASYNCHRONOUS MOVEMENT

G09 SKIP REST OF TRAVEL

G10 CORNER ROUNDING (OPTION)

G11 ADDITIONAL FUNCTIONS F, S, T

G13 ADDITIONAL M-FUNCTIONS

G17 PLANE XY

G18 PLANE XZ

G19 PLANE YZ

G20 JUMP TO PROGRAM

G22 CALL PROGRAMM

G23 CALL/ JUMP WITH CONDITION

G29 FREE CYCLE

G36 TOOL CHANGE

G40 RADIUS CORRECTION OFF

G41 RADIUS CORRECTION LEFT

G42 RADIUS CORRECTION RIGHT

G53 DISPLACEMENT OFF

G54 DISPLACEMENT ON

G55 DISPLACEMENT

G58 MEMORIZE ZERO POINT

G59 PUT T IN MEMORY

G67 SOFTWARE LIMIT SWITCH -

G68 SOFTWARE LIMIT SWITCH +

G74 REFERENCE POINT

G75 SCALE FACTOR ON

G76 SCALE FACTOR OFF

G78 FREE CYCLE

G79 FREE CYCLE

G80 CYCLE OFF

G81 FREE MODAL CYCLE

G82 DEEP DRILLING (MODAL)

G83 TAP DRILLING

G84 FREE CYCLE

G85 POCKET CYCLE

G86 CIRCLE SEGMENTATION

G87 CIRCLE POCKET

G88 LINEAR SEGMENTATION

G90 ABSOLUTE INPUT

G91 INCREMENTAL INPUT

G92 SET ACTUAL VALUE

1.2 LIST OF M-FUNCTIONS

M00 PROGRAMMED STOP

M02 PROGRAM END

M03 SPINDLE ON CW

M04 SPINDLE ON CCW

M05 SPINDLE STOP

M08 COOLING ON

M09 COOLING OFF

M10 CLAMPING ON

M11 CLAMPING OFF

M15 ACOUSTIC SIGNAL

M16 WAIT FOR “INPUT 1“ ACTIV

M17 WAIT FOR “INPUT 1“ INACTIV

M18 WAIT UNTIL NO MORE KEY IS PRESSED

M19 WAIT UNTIL INTERPOLATING AXES HAVE STOPPED

M2241 (M21) BLOCK UPDATING OFF

M2242 (M22) BLOCK UPDATING OFF IN PROGRAM CALLS

M2243 (M23) FEEDRATE POTENTIO-METER OFF

M2244 (M24) TESTRUN WITHOUT G04 AND M-FUNCTIONS

M2245 (M25) TESTRUN WITH RAPID TRAVERSE

M2246 (M26) KEYBOARD OFF

M2247 (M27) WAIT FOR “IN POSITION“

M2248 (M28) ACTUAL VALUE DISPLAY OFF

M30 PROGRAM END

M33 FEEDRATEPOTENTIOMETER OFF WITH G00

M34 KEY “MENU“ CALLS P9999

M35

M36

M37

M1xx HANDLING OF I / O CARD 1

M2xx HANDLING OF I / O CARD 2

1.3 LIST OF ERROR MESSAGES

Error	Error Code	Hint
INTERPOLATOR	(48)	Interpolatorcard gives no answer,Switch control off and on again.
CHECKSUM	(50)	Checksumerror during serial transmission. Repeat transmission
KEYBOARD	(91)	Keyboard out of order, a key is pushed after switching on, EXT START is active
EPROM	(96)	Error in the eprom.
MEMORIZED PROGRAMS	(101)	Error in one of the memorized programs Clear the complete memory and transmit the programs again via serial interface.
P NOT FOUND	(53)	This program number is not in memory.
N NOT FOUND	(54)	This bloc number is not in memory.
T NOT FOUND	(100)	This T-number is not in P9900.
NO F	(55)	Add a bloc with G11 F.... to your program
CHECK BLOC	(58)	Syntax error while memorizing a bloc
MEMORY FULL	(86)	No more memory free. Transmit some of your programs via serial interface.
TOO MANY SUBPROGRAM CALLS	(73)	Look at the description of G22.
P CANNOT BE EXECUTED	(78)	P0 or P9900 cannot be used for execution.
MOVING DISTACE =0	(79)	A movement of the distanc e 0 cannot be executed.
IN POSITION	(84)	Look at machinedata N803
LAGMAX	(85)	Look at machinedata N804
LAG ERROR	(140)	AXIS cannot follow with the programmed feedrate.
DATAFORMAT	(127)	Dataformat of serial transmission not OK.
V24		Parity or Framing Error durung serial transmission.
LIMIT SWITCH	(126)	Axis touched limit switch while moving.
SOFTW. LIMIT SWITCH	(129)	Softwarelimitswitch violated.
WAY OFF LIMIT SWITCH	(77)	Look at machinedata M712
INTERRUPT	(133)	E1 on IO4 was activated.
B-BUS ERROR	( )	Data transmission between INTERPOLATOR and DILAG failed.
INTERPOLATOR ERROR	( )	Data transmission from CPU to INTER POLATOR failed.

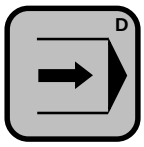
Error messages must be acknowledged with  , ↓ 94 displays a list of error messages.

2. OPERATING MODES

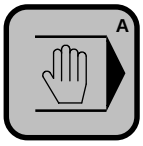
After switching on, the CNC displays the menue of the operating modes. The following 9 modes can be selected:

MANUAL MODE

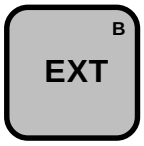
TEACH IN



AUTOMATIC MODE

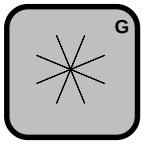


MANUAL INPUT (MDI)

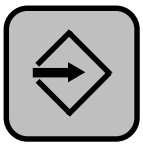


EXTERNAL DATA

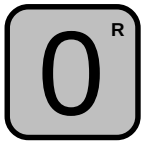
CLEAR MEMORY



CRAPHICS



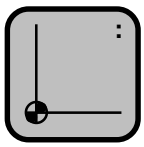
INPUT MODE



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:

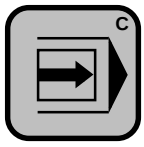


HOME POSITION

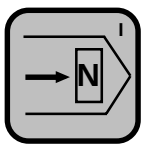
SPINDLE ON CW

SPINDLE ON CCW

RESET

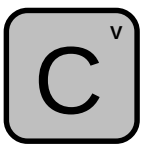


SINGLE STEPP



SEARCH BLOCK

DELATE BLOCK



CLEAR

2.1 MANUAL MODE

After pushing ✋ , 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 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 ✋ or TEACH IN can also be selected.

2.1.3 Hand wheel

The hand wheel can be activated by pressing f.e. +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 and then . The actual position is put into memory. The axes then can be moved to the home position by pushing and then 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 F, S, T, M 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. (Manual input; G91; ; START)
2. Select G00 and input a distance in X, Y 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. (Manual input; G90; ; START)
2. Select G00 and input a position in X, Y or Z.
3. Move to the desired position with START.

2.2.4 Starting a semicircle

1. Switch to incremental mode with G91.
2. Select a feedrate with G11 F..... .
3. Select G02 or G03 and input the endpoint and centerpoint of semicircle.
   G02 X+..50,000 Y...0,000 I+..25,000 J...0,000
4. By pushing " Start " the semcircle will be executed.

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. Wird die angebotene Programmnummer nicht akzeptiert, kann eine andere Nummer über die Tastatur eingegeben werden. TEACH IN in G90 or G91.
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 displayed 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, Y 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#111 Y#112 Z#113

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 .

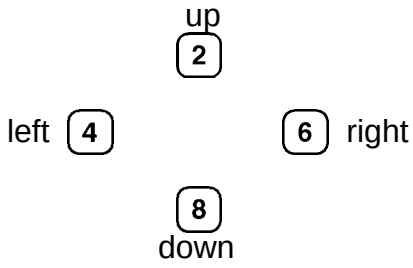
To delete a range of blocks of a program, enter the program number and then .- The CNC requests the starting and the ending block numbers. All blocks in this range will be deleted.

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) .

P9900 Tool table
N001 X ..... X ..... Z ..... R ...10,000

Example: P0001 Test program

N001	G91	Incremental input			
N002	G11	F...200 S.....	T.....1	M	
N003	G01	X+...30,000	Y+ ...20,000	Z	F .....
N004	G01	X	Y	Z-..10,000	F .....
N005	G01	X+...50,000	Y	Z	F .....
N006	G01	X	Y+....30,000	Z	
N007	G03	X-....50,000	Y+.....0,000	Z	
		I-.....25,000	J+.....0,000	Kreis	
N008	G01	X	Y-.....30,000	Z	
N009	G00	X.....	Y	Z+...10,000	
N010	G00	X-....30,000	Y-.....20,000	Z	

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: N0001*G...

G00 RAPID TRAVERSE

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

G01 LINEAR INTERPOLATION

N... G01 X.....,... Y.....,... Z.....,...
Up to 3 axes can be moved together. The feedrate (mm/min.) can be programmed before the block with G11.

G02/G03 CIRCLE INTERPOLATION CLOCKWISE / COUNTERCLOCKWISE

N... G02 X.....,... Y.....,... Z.....,... I.....,... J.....,...
XZ, are the endpoint of the circle segment, IJ 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.

Of course, the programmed endpoint must be a part of the circle.
This is the case, when $(X-I)^2 + (Y-J)^2 = I^2 + J^2 = R^2$.

A complete circle can be programmed as follows:
N... G02 X.....0,000 Y.....0,000 Z.....,... I.....20,000 J....0,000

A helix interpolation will result if in addition to X and Y, also the axis Z is programmed.

G04 DWELL TIME
N... G04 H....,...
Dwell time programming between 0,010 and 9999,990 seconds.

G05/G06 CIRCLE INTERPOLATION WITH INPUT OF RADIUS
N... G05 X.....,... Y.....,... Z.....,... R.....,...
Input the desired endpoint in X and Y, the radius in R. The sign of R determines if a small or large circle segment is generated.
Computer time is necessary for transforming G05 / G06 into G02 / G03, to G05 / G06 must not the used when fast bloc updating is necessary in the automatic mode.

G07 CIRCLE DEFINED BY ANGLE (In design state)
N... G07 X.....,... Y.....,... Z.....,... R.....,... W.....,...
The circle segment is defined by the radius R and the ending angle W. The starting point of the circle is tangent to the preceding bloc. X, Y have no meaning, the other axes can interpolate with X,Y. Only in G91 and in G40 !!

G08 ASYNCHRONOUS MOVEMENT
N... G08 X.....,... Y.....,... Z.....,... F .....,... W.....,...
Beside an interpolation in progress, the axes here defined begin to move the programmed distance. F defines the speed, W the number of repetitions of the movement. Function G13 M91 interrupts G08.
The programmed values are always in G91.

Exampels:

- G08	X100	F100	W99	:	100 Pendelbewegungen mit F100
- G08	X100	F200	W0	:	Asynchrone Bewegung um 100 mm
- G08	X0	F300	W0	:	X is running continously with F300

G09 SKIP REST OF TRAVEL
N... G09 X.....,... Y.....,... 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.
In addition the NZ Flag, which can the interrogated with "↓ " 54, is set.

Possible uses:
- recognization of tool fracture
- digitization of workpieces

G10 CORNER ROUNDING
N... G10 X....,... Y....,... X....,... Y....,... R....,...
In the incremental mode G 91, the 1. XY input field is to be programmed with the first line, in the 2. XY the second line and in R the radius of the corner between these lines.

N... G10 X.100,000 Y...0,000 X...0,000 Y.100,000 R...5,000

If the 2. XY input field is not programmed, R is interpreted as rotating angle for the 1. XY distance.
N... G10 X.100,000 Y...0,000 X....,... Y....,... R..45,000

With G10, the resulting CNC blocs are calculated and memorized already in the input mode. Tool correction G41/G42 is possible.

G11 ADDITIONAL FUNCTIONS F,S,T,M
N... 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
N... G13 M.... M.... M.... M....
G13 allows to program several M-functions in one block.

G17 PLANE XY, (Initial state)
G18 PLANE XZ,
G19 PLANE YZ
N... G17 XY plane
This modal function will switch to the XY plane. After reset, G17 will be effective and will be overwritten by G18 or G19.
During path compensation (G40 - G46), the plane must not be changed.

During G41 / G42, the plane must not be changed. The center of the circle is always programmed in I, J, also in the plane XZ and YZ. The end point of the circle is entered in XZ during G18 and in YZ during G19.

G20 JUMP TO PROGRAM
N... 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

N... 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

N... 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.

G36 TOOL CHANGE

N... 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)

N... 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:

- G41, G42 works in the XY plane, the tool length compensation works on the Z axis.
- Before using a path compensation, an appropriate tool (P9900) 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 XY 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.
- In the compensation mode it is not allowed to move the Z axis together with X or Y at the same time. So, the following program is not allowed.

Example for path compensation:

P9900 Tool table
N001 X....0,000 Y....0,000 Z....0,000 R....10,000

Exampel: P0001 Test program

N001	G91	Incremental input			
N002	G11	F...200 S.....	T.....1	M	
N003	G42	Radius correction right			
N004	G01	X+...30,000	Y+ ...20,000	Z	F .....
N005	G01	X	Y	Z-..10,000	F .....
N006	G01	X+...50,000	Y	Z	F .....
N007	G01	X	Y+....30,000	Z	
N008	G03	X-....50,000	Y+.....0,000	Z	
		I-.....25,000	J+.....0,000	Kreis	
N009	G01	X	Y-.....30,000	Z	
N010	G00	X.....	Y	Z+...10,000	
N011	G40	Correction off			
N012	G00	X-....30,000	Y-.....20,000	Z	

P0002 **Circle with radius correction**
N001 G11 F...200 S..... T.....1
N002 G42 Radius correction right
N003 G01 X-..10,000
N006 G02 X-....0,000 Y+.....0,000 I+..10,000 J+...0,000
N008 G40 Correction off
N009 G01 X+.10,000

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 ON
N... G54 X.....,... Y.....,... 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
N001 G90 Absolute input
N002 G00 X.....0,000 Y.....0,000
N003 G01 X...20,000 Y.....0,000
N004 G01 X...20,000 Y...20,000
N005 G01 X.....0,000 Y.....0,000

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

.
.
N010 G90 Absolute input
N011 G54 X..100,000 Y...50,000
N012 G22 P0010

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

G55 DISPLACEMENT
As G54, however must be switched off with G55 X0, Y0, Z0. This function must not be used together with G86 in the TURNING MODE.

G58 MEMORIZE ZERO POINT

N... G58 X.....,... Y.....,... Z.....,...

With G58, the zero point of the workpiece can be memorized, to after switching on the CNC, it can be moved at automatically.

The following program must be in memory:

```
P0074
N001 G11      T 0              ;T0 must be selected with G11 or G36
N010 G74      Z 0              ;      The positiv limit-switches
N020 G74      X 0              ;      must be moved to!!!
N030 G74      Y 0              ;
N040 G92      X0   Y0   Z 0    ;This bloc number must be N40 !!!!
```

After switching on, P0074 must be invoke. After that, a tool must be selected with G36. After that, any axis, f.e. Z, is moved to the working piece until contact.
Then, in MANUAL INPUT MODE; G58 Z0 is input and executed with START.
Then, the same is done with the other axes and the workpiece - zero point is memorized. The determined values are deposit in bloc N40. With the next call of P74, these values are finally put into the actual value display.

G59 PUT T IN MEMORY

N... G59 X.....,... Y.....,... Z.....,...

G59 calls P9959 in the MANUAL INPUT MODE and transmit the actual cursor position #79.

G67/68 SOFTWAR LIMIT SWITCH - / +

N... G67 X.....,... Y.....,... Z.....,...

If these limits are passed, the CNC will stop and display the error message “SOFT-WARE LIMIT SWITCH“.
If the destination point of the programmed bloc falls behind the software limit switches, also an error message is displayed.
The software limit switch is disabled by programming G67, X0 f.e.
During an active G67/G68, G92 must not be programmed!

G74 REFERENCE POINT
N... G74 X.....,... Y.....,... 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:
N... G74 X....0,000
N... G74 Y.....0,000
N... G74 Z-...1,000

X will be moved to the positive, Z 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, Y and Z to their reference point and to put the actual counter to the correct value for X, Y and Z. See G58.

G75 SCALE FACTOR ON (Initial state)
G76 SCALE FACTOR OFF
N... G75 X.....,... Y.....,... Z.....,... W.....,...
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, Y and Z. A negative value will switch the polarity of the movement.
Under W, a rotation angle can be programmed. A contur in X, Y will be rotated with this angle.
In the actual value display, small rounding errors can appear at the end of the contour.

G78 FREE CYCLE
N... G78 X.....,... Y.....,... Z.....,... U.....,... V.....,... A.....,...
Will put the values X, Y, Z, U, V, A in the registers #80 - #85.

G79 FREE CYCLE
N... G79 X.....,... Y.....,... Z.....,... U.....,... V.....,... A.....,...
Will put the values X, Y, Z, U, V, A in the registers #80 - #85.

G80 CYCLE OFF (Initial state)
Switches off G81 - G82 .

G81 FREE MODAL CYCLE
N... G81 X.....,... Y.....,... Z.....,... U.....,... V.....,... A.....,...
Will put the values X, Y, Z, U, V, A in the registers #70 - #75.
After each movement then the program P9981 will be invoked.

G82 DEEP DRILLING
N... G82 Z.....,... Q.....,... V.....,... H.....,... F.....,...

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

Cycle:
- with rapid traverse to the work piece surface - 0,5 mm
- with feedrate to the depth Q and dwell time H
- with rapid traverse back to security distance V
- with rapid traverse to the depth Q - 0,5 mm
- with feedrate to 2Q
.
.
- the remaining distance is executed
- dwell time H
- rapid traverse back to security distance V

Exampel:
```
N010  G82 Z-...10,000 Q-..4,000 V....1,000 H....1,000 F.....,...
N011  G00 X10
N012  G00 X20
N013  G00 X30
N014  G80
```
G82 is a modal cycle. A bloc with this function does not execute a deep drilling cycle! However after each following bloc a deepdrilling cycle is until the cycle is switched off with G80.

G83 TAP DRILLING WITH Ausgleichsfutter
N... G83 Z.....,... K.....,... Tapdrilling

Eingabe:
Z: depth
K: pitch

The feedrate is input with G11 F....., then G83 calculates the corresponding spindle speed. If the resulting spindle speed is lower than 60 rot/min, a error message is displayed.

G84 FREE CYCLE
N... G84 X.....,... W.....,... Z.....,... Q.....,... V.....,...

G85 POCKET CYCLE
N... G85 X.....,... Y.....,... Z.....,... Q.....,... V.....,...

The length and width of the pocket must be entered in X and Y. The milling cutter must be positioned at security distance above the center of the pocket.
X and Y must exceed 4*R. R must exceed 0,5 mm. Z and Q must be negative.
R = Radius of cutter.

Cycle:
- move to starting point of the pocket
- rapid traverse to surface -0,5 mm
- with feedrate to Q
- mill outside frame of pocket
- move back to starting point +0,5 mm in X and Y
- mill pocket content in meander form
- move back to security distance
- repeat until desired depth is reached
- move to security distance

Example:
N005 G11 F1000 T0001
N010 G85 X...40,000 Y...20,000 Z-..10,000 Q-...4,000 V....1,000

The pocket can be milled in the opposite direction by first programming G75 X-...1,000.

G86 CIRCLE SEGMENTATION
N... G86 X.....,... Y.....,... Z.....,... D.....,... O..... P.....

The starting and ending angle of the circle segment must be programmed with X and Y. D is used to program the diameter of the circle, O to program the number of segments. After each segment,the program P will be called. An additional movement in Z, which will also be divided in O segments, can be programmed. Input Q as negative number, if program P should also be called at the first programmed angle.

Cycle:
- move to first segment
- call program P
- move to next segment
.
.
- call program P a last time

Note:
- The actual angle is stored in parameter #46. The program P can use it for further calculation.
- G86 can be used with an active scaling factor G75 (f.e. X....2,000). The circle is extended to an ellipse.

Exampel:
P0001
N010 G86 X....0,000 Y...90,000 Z....0,000 D...50,000 V...4,000 P 2

P0002
N001 G00 X....5,000
N002 G00 X-...5,000

G87 CIRCLE POCKET
N... G87 D.....,... Z.....,... Q.....,... V.....,... A.....,...
The diameter of the circle pocket must be programmed in D. The milling cutter must be positioned at the center of the pocket at security distance V.
A is the starting diameter, used when f.e. an existing hole must be enlarged. Before a tool with a radius > 0 must be activated.

Cycle:
- with rapid traverse to surface -0,5 mm
- with feedrate to Q
- mill circle pocket beginning at the center
- move to starting point
- with feedrate to Q
- etc.

Example:
G92 X....0,000 Y....0,000 Z....1,000
G11 F1000 T1
G87 D...50,000 Z-..10,000 Q-..10,000 V....1,000 A....0,000

If V is programmed with a negative value, only.

G88 LINEAR SEGMENTATION
N... G88 X.....,... Y.....,... Z.....,... O..... P.....
A linear 3D interpolation programmed in X, Y, Z is divided into O segments. After each segment, the program P is called. In this program, the user can define the functions to be executed after each segment.

Exampel:
P0001
N010 G88 X...50,000 Y...30,000 Z....0,000 O.....7 P.....2

P0002
N001 G00 X....5,000
N002 G00 X-...5,000

With positioning controls program P cannot be input.

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.

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.

G92 ZERO OFFSET

N... G92 X.....,... Y.....,... Z.....,...

The programmed values are taken into the actual value counter. If G54 or a tool is active, these values are calculated to the actual counter. So, the value programmed in G92 will not necessarely appear on the screen.

3.2 THE M-FUNCTIONS

Die M-Funktionen werden im Grafikmodus nicht ausgeführt.
The M-functions have the following meaning:

M00	programmed stop
M01	programmed stop with acoustic signal
M02, M30	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
M19	Wait until interpolating axes have stoped
M41/51	Movement without acceleration or deceleration. Gives continous movement also with non-tangent blocs
M90	Pendeln (G08) at the end of a hub stop
M91	asynchronous axis (G08) stop
M97	Wait until all axes are in position

Hint: G13 M19 will stop the bloc look ahead function.

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“ can 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.
The CNC calculates internally with integer values, the number X + 1,000 is internally 1000, the number F100 ist internally 100. If the number of digits after the decimal point is 2, then X+1,00 internally 100!

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 Y.....,... 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.:

N002 ↓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.

N003 ↓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.

N100XR Ballscrew errorcompensation in X
N200YR Ballscrew errorcompensation in Y
N300ZR Ballscrew errorcompensation in Z

More information on page 7/4 .

N698XYZ Reserved for N790, value 64.

N699XYZ CORRECTION REFERENCE POSITION (0)
This value will be moved after the referencepulse with the feedrate programmed in N902A. R = 0.

N700XYZ 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:

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

N701XYZ F START (100) **N702XZ F STOP** (100)
Start and Stop frequency in mm/min of an interpolation. The smallest programmable value is 1.

N703XYZ B START (500) **N704XZ B STOP** (500)
Acceleration and deceleration in multiples of 10 mm/sec².

Hint:
To small values (f.e. <10) in N 701 to N 704 together with a small value (f.e. 50) in N 706 (steps/mm) can result in a problem that the axes won´t start.

N705XYZ F 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:

N706 X...1000 N707 X.....5

If the number of steps/mm smaller than 100, it is reco.....to select cm as base unit. So you input steps/cm and F in cm/min.

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

With a rotary table (N790 X2) the number of steps per rotation of the table must be input.

Example: A stepping motor with 800 steps drives a rotary table with a gear of 18:1.
The resulting values are: 800 steps x 18 = 14400 steps for 360 degrees.
N706 X..14400
N707 X....360

For servomotors the resolution of the measuring system multiplied by 4 must be input.

Example: A transducer on the X axis gives 250 pulses per (1) mm.
N706 X...1000
N707 X.....1

By this kind of input also values like 200 steps per 3 millimeters can be processed.
When using EXT SYNC (X...), N706C will define the steps/rotation of the external transducer.

N708XYZ MODULO (0)
Modulo function for the position display. Linear axes: 0, rotary axes: 360000.

N709XYZ LIMIT SWITCH DEBOUNCE TIME F (10)
During this time in ms at least the limit switch signal must be stable.
Max. value 255.

N710XYZ F REFERENCE (500)
Feedrate in mm/min for G74.

N711XYZ 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.

N712XYZ 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.

N713XYZ MAXIMAL DISTANCE FOR STOP (0)
If this value is programmed, a movement is stoped within this maximal distance, when a limit switch is reached.

N714XYZ BACKLASH in µm (0)
This value in µm will be added to the following movement at each change in the moving direction.

N716 XYZ SOFTWARE LIMIT SWITCH - (0)

N717 XYZ SOFTWARE LIMIT SWITCH + (0)

N722XYZ F FOR BACKLASH COMPENSATION (0)
With a value of 0, the feedrate of the compensation is the value from N701.

N790XYZ 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	Funktion
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.
04	A rotary axis will move off the limit switch in negative direction.
08	Axis with spline interpolation. (Option)
16	Axis driven by a servomotor and not a stepping motor.
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 REFPULS (N902A)** until the reference pulse of the transducer is detected. Here the internal counters of the CNC 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,Y,Z = 771 (1+2+256+512),

Servoaxis: SUM 787 = (1+2+16+256+512)

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

N800XYZ 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.

N803XYZ 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. This value must be reached within 2 seconds, also an error message will be displayed.

N804XYZ LAGMAX F (1600)
If the lagerror becomes graeter than this value, the CNC stops and displays an error message.

N812XYZ 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	**Y**	
4	**Z**	**f.e. (Axis X + Y + Z = 7)**

N813Z FATAL LAGERROR (32000)
If the lagerror of one axis exceeds this value, the servoamplifiers are disabled. This value should always be greater (min. 30%) than N804XYZ. This value is common to X, Y, Z. The high input is 32000!

N900A SPINDLEAXIS (0)

0: None of the axes X, Y, Z is a spindleaxis.

1-3: Axis X.. or Y.. or Z.. is a spindleaxis.
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.
This axis must be defined in P0 N790 as servoaxis and must be activatied in P0 N813.

0: no spindle axis, 1-3: axis 1-3 is spindle axis

M03 or M04 activates the spindleaxis, it can be programmed with G11 S.... .
This axis must be defined in P0 N790 as servoaxis and must be activatied in P0 N813.
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.

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.

N900 Y
User code to programmed P000 (Machinedata)

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 I (0)

The date N902X is composed from the following numbers:

01: Ignore error message after RESET.
02: Disabel keybord control via V24.
64: XON-XOFF protocoll in EXTERNAL DATA for serial data input. Only with baudrate=9600 and function 2 in EXTERNAL DATA.
Erroneous blocs will not cause an error message and will be memorized.

N902Y RESERVED CONTROLCODES I 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.
1024: Initial state G90

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.

N902 A F REFPULS (20)

Feedrate in mm/min. for search of referencepulse on transducer, if N790 is programmed with 32.

N903XY I / O - INITIALVALUES (0)

These values will be put to the ouputs of the I/O cards 1-2, when switching on or when changing to MENU.

Exampel: A Value of 3 on X set the Outputs 1 and 2 on I / O 1

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 ext.
2048: Anzeige des programmierten S-Wertes kommt aus # 105.

N905X MISCELLANEOUS CONTROLCODES

1: If a bloc in the Automatic Mode is stoped, the spindle and the cooling will be switched off

32: In the normal mode, only one keystroke is necessary for C and ↓ .

512: External interrupt will be generated by activating input 8 on X1 (signal IN 8) and P 9998 will be invoked. If the input is activated, the spindle and the axes are stoped. Then P9998, it in memory, is executed.

N905A Waiting time for IN POSITION M27 (2000)

N905C External Stop

In N905C, an input can be selected as function " EXTERNAL STOP " (page 9/2)

I/O bloc 0 has the value 1, I/O bloc 1 has the value 2.

Calculation of input value: bloc number 1 oder $2 * 256 + 2^{A}$

A = 0 for IN1
1 for IN2
2 for IN3

Exampel: I/O bloc 0 IN5 is defined

$(1 * 256) + 2^{4} = 272$

I/O bloc 1 IN12 is defined

$(2 * 256) + 2^{3} = 512 + 8 = 520$

N906A Lubrication pulse (0)

Output 8 (OUT 8) delivers a lubrication pulse of 1 sec. length, the intervall between each pulse is input in N 906A in minutes.

N923U Teiler für externes Handrad

Mit 4112 wird der Teiler auf 4 eingestellt

N921XYZU SCREEN ADAPTION

5" LCD Display	**9" LCD Display**	
X 320	X 640	pixel in X
Y 240	Y 480	pixel in Y
Z 114	Z 192	mm in X
U 85	U 142	mm in Y

N925X SPINDLE PULSES per rotation of the spindle (0)

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.

Using the I/O cards

The CNC is to move to position X100, Y50. There a cylinder with a magnet valve (24V) should be activated. The cylinder will move down where it activates limit switch nr 1. Then the cylinder should move upwards until limit switch nr 2 is activated.

Connections:
Connect limit switch nr 1 to I1 of the I/O card 1.
Connect limit switch nr 2 to I2 of the I/O card 1.
Connect the magnet valve to O1 of the I/O card 1.

Program:

```
N001 G90                ; Absolute input
N002 G00 X100 Y50
N003 G13 M0141          ; Magnet valve on
N004 G13 M0161          ; Wait for limit switch nr 1 to go active
N005 G13 M0151          ; Magnet valve off
N006 G13 M0162          ; Wait for limit switch nr 2 to go active
N007 G00 X200 Y100
```

The blocks N003 to N006 can be concatenated as follows:
N003 G13 M0141 M0161 M0151 M0162

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 (see page 7/1).

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 Y .... The pin connections can be found on page 9/10 + 9/12. We deliver DC- and AC-Servo motors.
You select the appropreate connector.
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 feedrate 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 6/2). 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.

7.5 LIMIT SWITCHES

3

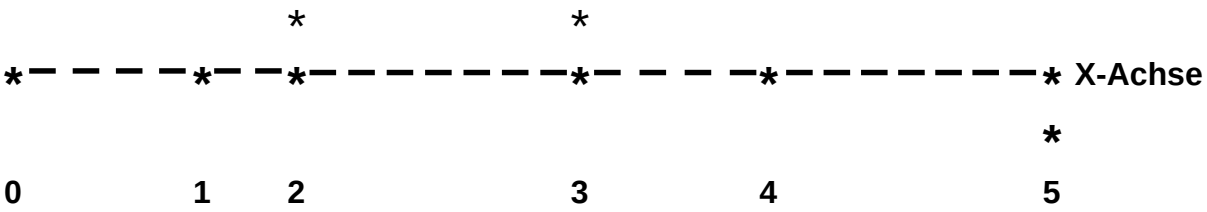
The limit switches will be connected to X9 f.e. (for servo system) page 9/7, 9/8, 9/9). As always, first only the X-axis will be wired. An external 24V- supply will be used for powering the limit switches. The limit switch which will be activated when the axis is moving in the postive direction, will 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 X9/21with 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 6/4.

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

7.6 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. Stützpunkt,	Fehler	+3mm
N102	X.180,000	R.182,000	;2. Stützpunkt,	Fehler	+2mm
N103	X.370,000	R.372,000	;3. Stützpunkt,	Fehler	+2mm
N104	X.460,000	R.459,000	;4. Stützpunkt,	Fehler	-1mm
N105	X.570,000	R.568,000	;5. Stützpunkt,	Fehler	-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.

Explanation of machine data P0000

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

N813  X..7   Y..... Z..... U..... V..... A.....   ;Servoachsen X, Y, Z sind aktiviert

N902    X..... Y0000512 Z0000001 U.....  V.....  A.....
                        |                |
                        |                └── Textdisplay in english
                        |
                        └── Tool is behind spindle axis

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 for referencing all the axes.

The following program must be in memory:

```
P0074
N001 G11     T 0              ;T0 must be selected with G11 or G36
N010 G74     Z 0              ;     The positiv limit-switches
N020 G74     X 0              ;     must be moved to!!!
N030 G74     Y 0              ;
N040 G92     X0   Y0   Z 0    ;This bloc number must be N40 !!!!
```

this program is used by G58. (see page 3/7)

The determined values are deposit in block N40. With the next call of P74, these values are finally put into the actual value display.

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, Y and Z.
X, Y, Z of the remaining tools should contain the lenght difference to T1.

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

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 ↓51   (JUMP POS TO) N0190
N153 ↓96    #004   #085   #...   #...   #...   #...   #...
N154 ↓01    #081 = #086 + @+...0,001
N155 ↓02    #080 = #081 - @+...0,005
N156 ↓52   (JUMP NEG TO) N0159
N157 ↓00    #081 =       @+...0,001
N159 ↓53   (JUMP    TO) N0100
N190 G11 F..... S..#081 T....   M....
N199 ↓53   (JUMP    TO) N0030

N200 ↓02    #081 = #082 - @+...0,001
N201 ↓54   (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 ↓53   (JUMP    TO) N0480

N220 ↓02    #081 = #082 - @+...0,002
N221 ↓54   (JUMP NZ  TO) N0240
;change to tool #3
N222 G23 P....   N0490   W....   M0162
N223 G13 M0141 M0162 M0151 M.... M....
N224 ↓53   (JUMP    TO) N0480
N240 ↓02    #081 = #082 - @+...0,003
N241 ↓54   (JUMP NZ  TO) N0260
;change to tool #3
N242 G23 P....   N0490   W....   M0163
N243 G13 M0141 M0163 M0151 M.... M....
N244 ↓53   (JUMP    TO) N0480

N260 ↓02    #081 = #082 - @+...0,004
N261 ↓54   (JUMP NZ  TO) N0280
;change to tool #4
N262 G23 P....   N0490   W....   M0164
N263 G13 M0141 M0164 M0151 M.... M
N264 ↓53   (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 #103 in the MANUAL MODE IS PUSHED. P9974 is programme SO THAT A „HOME“ POSITION“ can be memorzed 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 execute when the CNC is switched on.

```
;N002 #102, #103 and #104 contain the actual value of X, Y, and Z
before the CNC was switched off.
;programming of X-axis.
N004 G92 X.....#102 Y.....#103 Z.....#104 SET ACTUAL VALUE
P9936 N900 looks what tool is active.
N005 G20 P9936   N0900    JUMP PROGRAM
```

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:

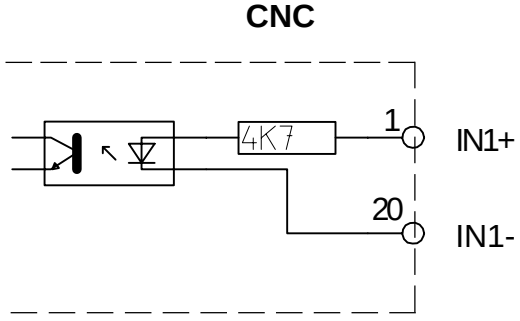
- All connections to the CNC must be shielded, the shield must be firmly connected to the metallic case of the connector.

X1 Input

SUB-D 37 pol. female — see machine data N905X

pin	value	signal	pin	signal	M-function	
1	0	IN1+	20	IN1-	M161	(TW01)
2	1	IN2+	21	IN2-	M162	(TW02)
3	2	IN3+	22	IN3-	M163	(TW03)
4	3	IN4+	23	IN4-	M164	(TW04)
5	4	IN5+	24	IN5-	M165	(TW05)
6	5	IN6+	25	IN6-	M166	(TW06)
7	6	IN7+	26	IN7-	M167	(TFIN)
8	7	IN8+	27	IN8-	M168	Ext.interrupt(N905)
9	0	IN9+	28	IN9-	M261	(S10A)
10	1	IN10+	29	IN10-	M262	(S20A)
11	2	IN11+	30	IN11-	M263	(S30A)
12	3	IN12+	31	IN12-	M264	(S40A)
13	4	IN13+	32	IN13-	M265	(MFIN)
14	5	IN14+	33	IN14-	M266	
15	6	IN15+	34	IN15-	M267	
16	7	IN16+	35	IN16-	M268	
17	---		36			
18	---		37			
19	---------					

Internal diagram of inputs **IN1bis IN16.**



Input voltage 20 - 30V

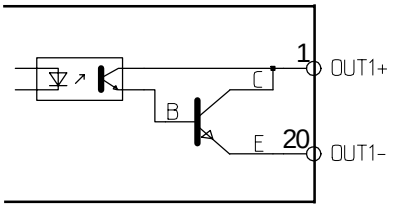
X2 Output

SUB-D 37pol male

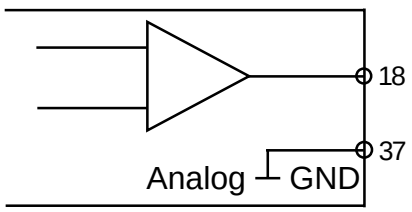
pin	signal	pin	signal	M-function	
1	OUT1+	20	OUT1-	M141	(TDZ Toolchanger forward)*
2	OUT2+	21	OUT2-	M142	(TDZ Toolchanger backward)*
3	OUT3+	22	OUT3-	M143	
4	OUT4+	23	OUT4-	M144	
5	OUT5+	24	OUT5-	M145	
6	OUT6+	25	OUT6-	M146	
7	OUT7+	26	OUT7-	M147	
8	OUT8+	27	OUT8-	M148	(Lubrication pulse N906A)
9	OUT9+	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 program-	
19	-------			med spindle speed **S** in SM version)	

*** Exampel for a toolchanger**

Internal diagram of outputs :



Outputs: 30V / 0,5A max.



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)**

SUB-D 9pol. female

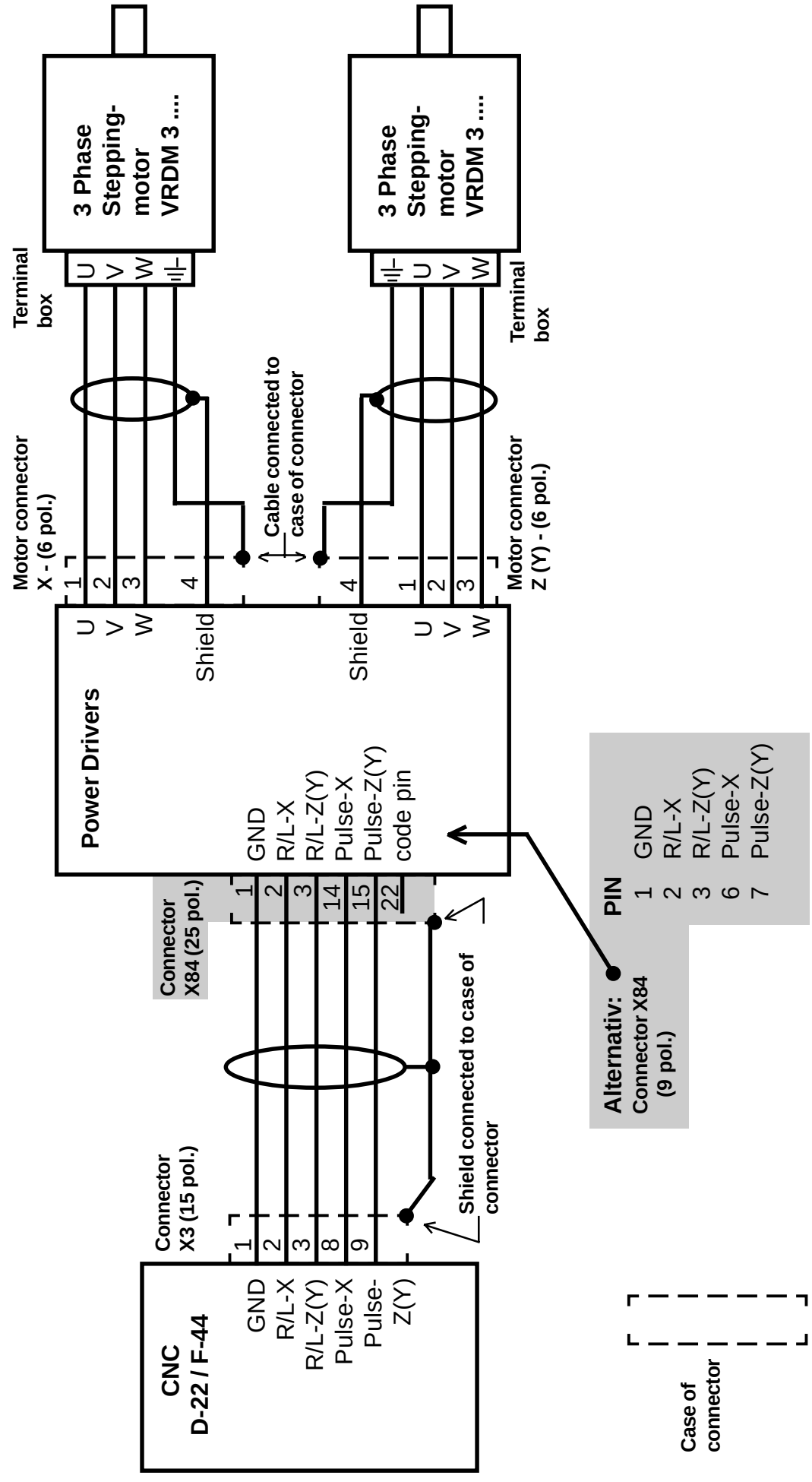
Pin	Signal	Pin	Signal
1	+ Vcc	6	B*
2	GND	7	nc
3	A	8	nc
4	B	9	code pin
5	A*		

X6 V24 SUB-D 9pol. male

CNC		PC (9 pol.)			PC (25pol.)
pin	**signal**	**pin**		**signal**	**pin**
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.

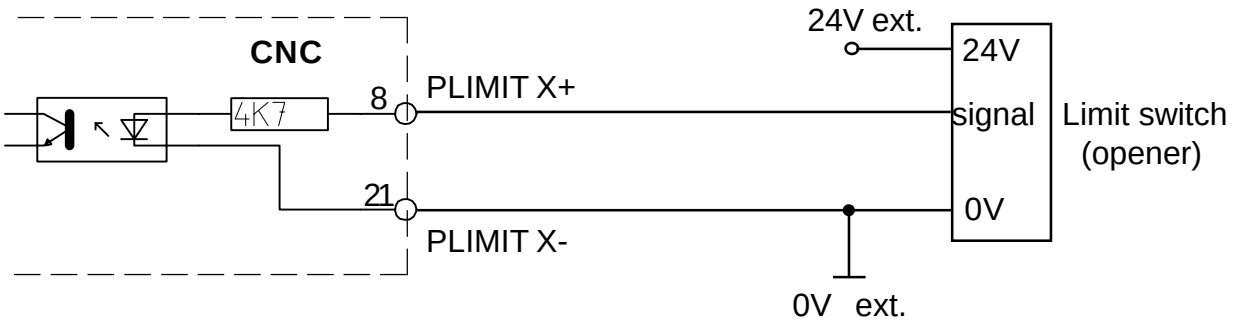

3 Phase Stepping-motor VRDM 3 ....
Terminal box
U V W
Motor connector X - (6 pol.)
1 2 3 4
Cable connected to case of connector
Shield
Motor connector Z (Y) - (6 pol.)
Power Drivers
GND
R/L-X
R/L-Z(Y)
Pulse-X
Pulse-Z(Y)
code pin
Connector X84 (25 pol.)
1 2 3 14 15 22
Alternativ:
Connector X84 (9 pol.)
PIN
1 GND
2 R/L-X
3 R/L-Z(Y)
6 Pulse-X
7 Pulse-Z(Y)
Shield connected to case of connector
Connector X3 (15 pol.)
1 2 3 8 9
CNC D-22 / F-44
GND
R/L-X
R/L-Z(Y)
Pulse-X
Pulse-Z(Y)
Case of connector

X9 LIMIT SWITCH

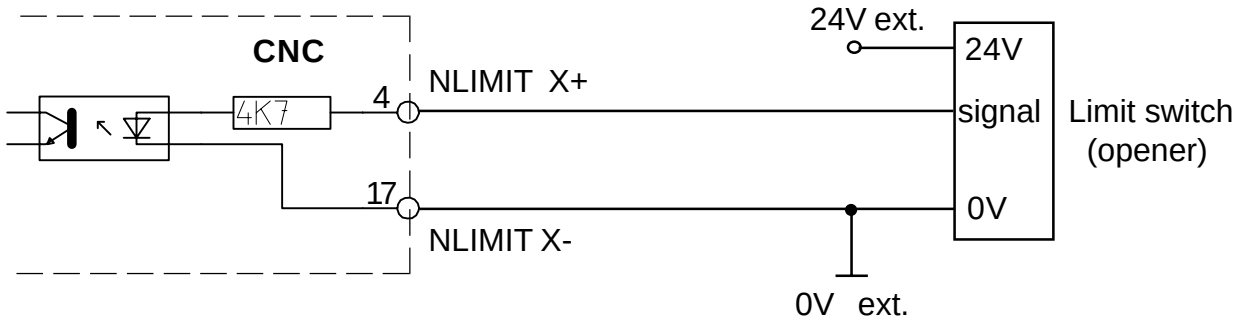
SUB-D 25pol. female

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

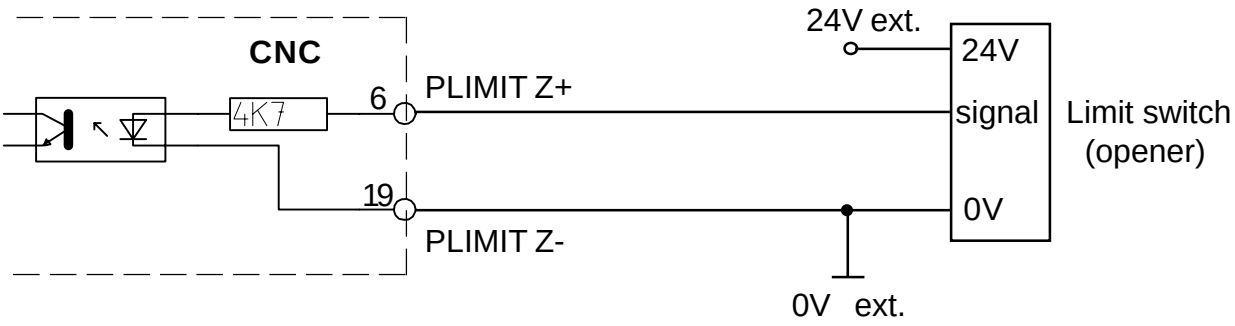
Limit switch positive X



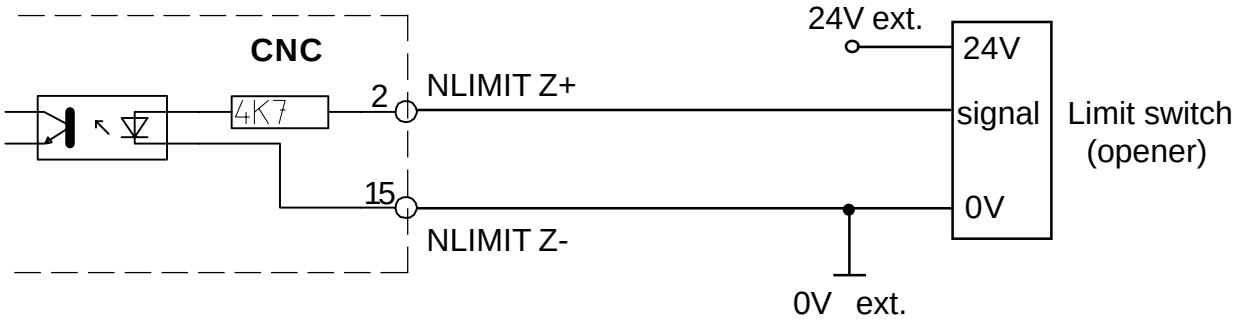
Limit switch negative X



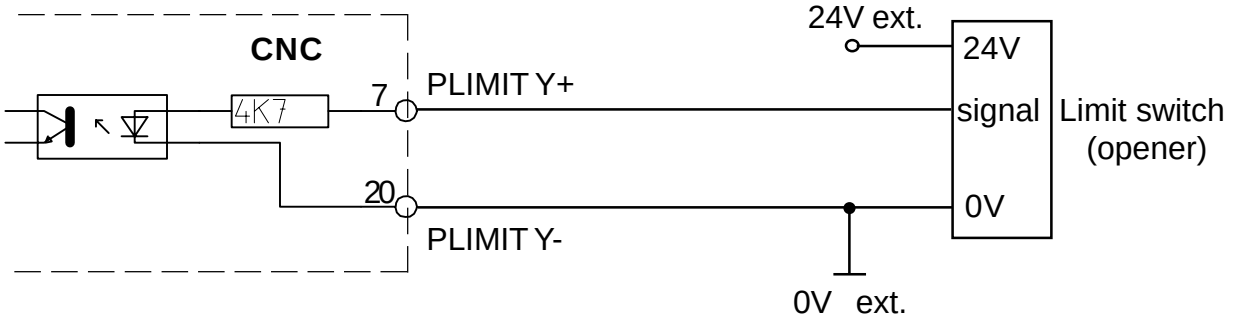
Limit switch positive Z



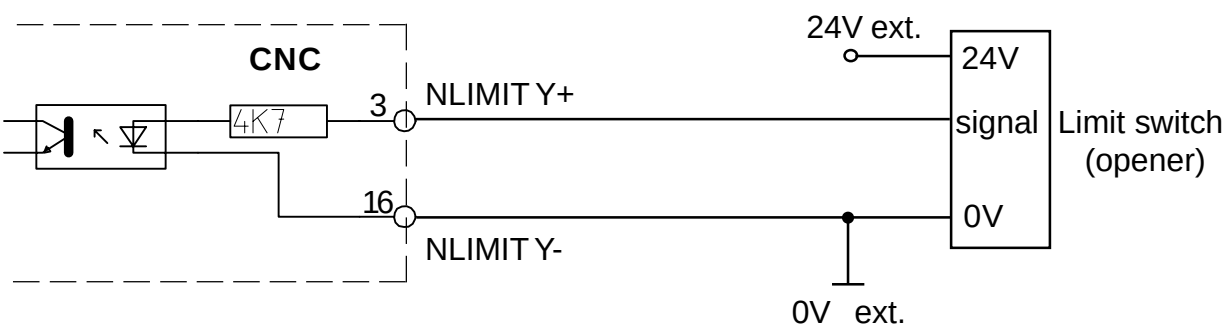
Limit switch negative Z



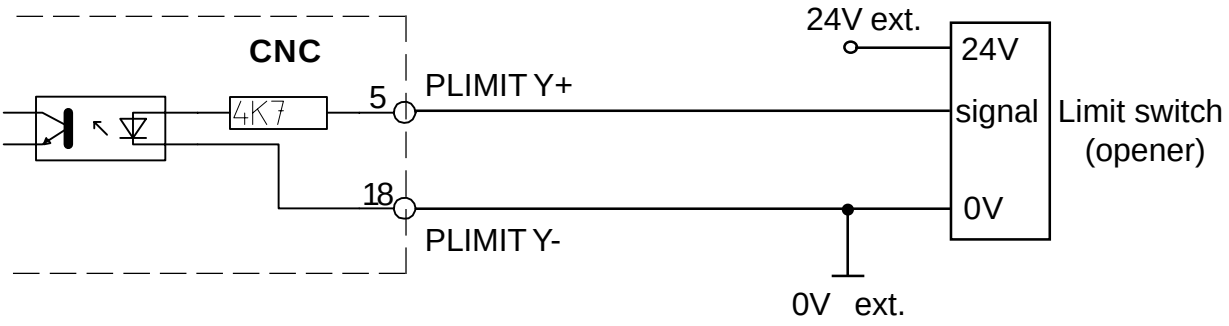
Limit switch positive Y



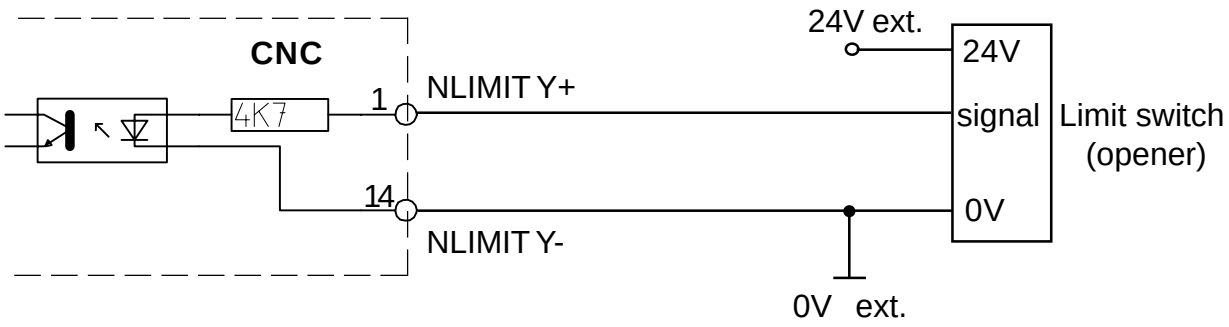
Limit switch negative Y



Limit switch positive U



Limit switch negative U

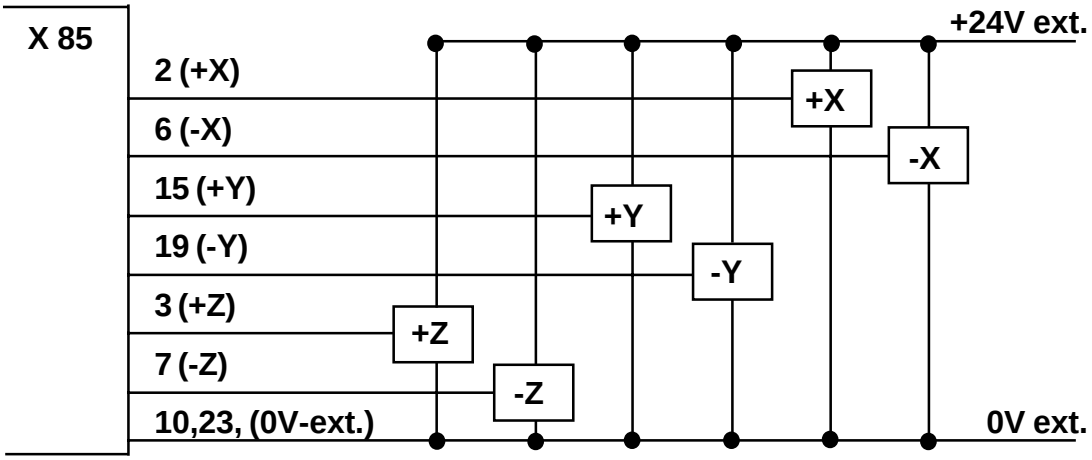


X85 LIMIT SWITCH **SUB-D 25pol. female**

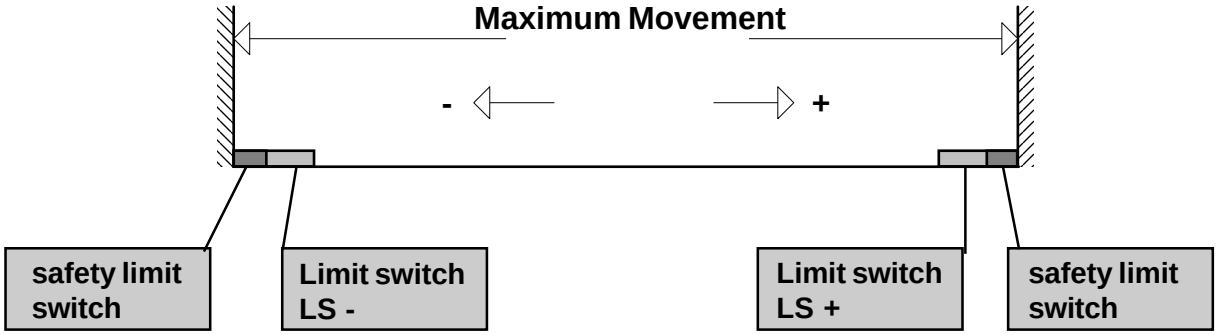
pin	signal	pin	signal
1		14	
2	+X	15	+Y
3	+Z	16	+U
4	+V	17	+A
5	+B	18	+C
6	- X	19	- Y
7	- Z	20	- U
8	- V	21	- A
9	- B	22	- C
10	0 V external	23	0 V external
11	code pin	24	
12		25	
13			

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 / X11-3 / X11-4 SERVO OUTPUT / ENCODER

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*			

X11-1 = X - axes, X11-2 = Y - axes, X11-3 = Z - axes, X11-2 = U - axes.

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 of the CNC at all because of possible electr. noice which could be inducted on the cables.
The amplifier must immediatly be disabled when MOTOR ON is disabled, independently of a feed command on the Analog Output.

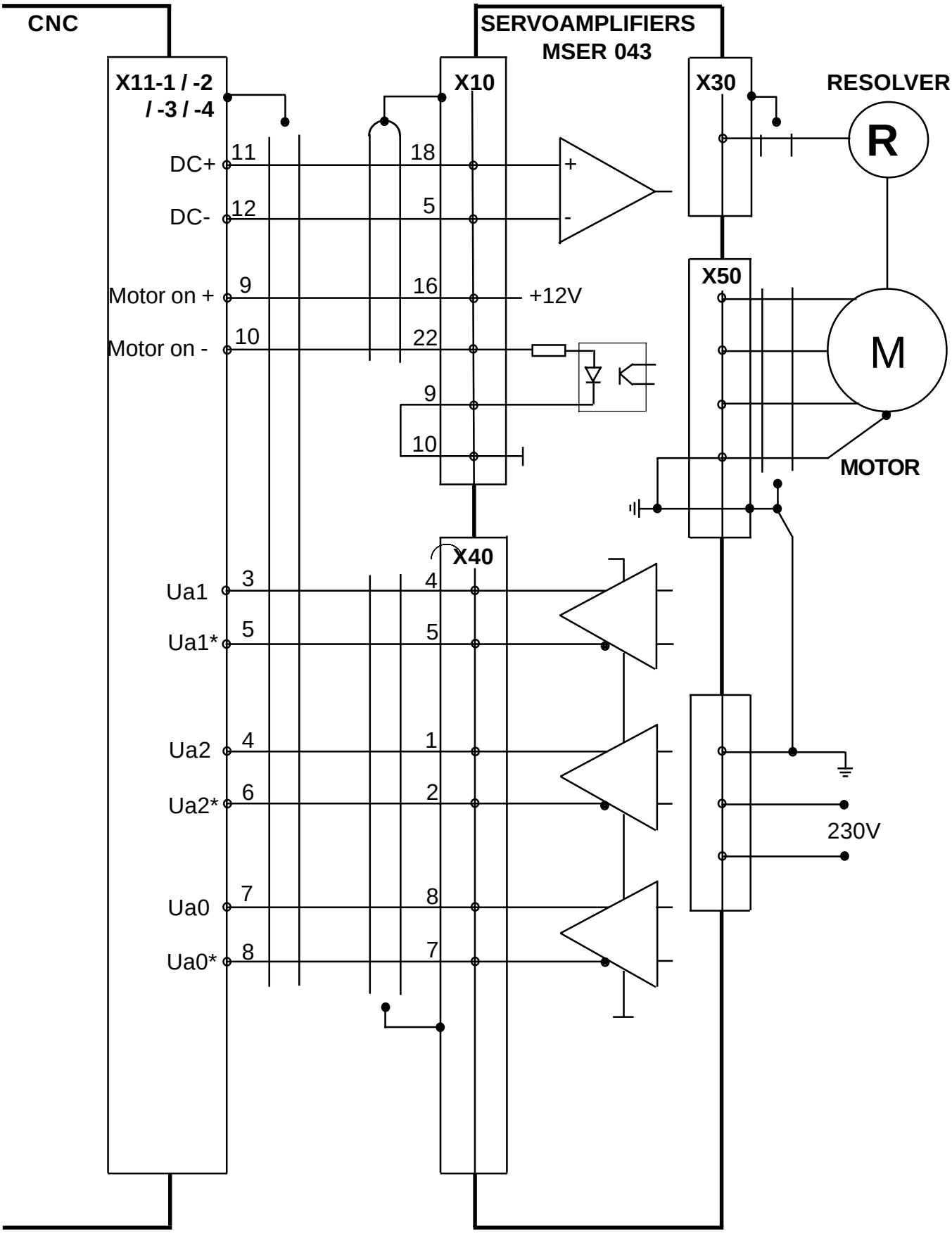
Note:
- Use shielded cable. Connect shield to the case of the CNC to the ground connection.
- Use transducer with TTL output!

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:

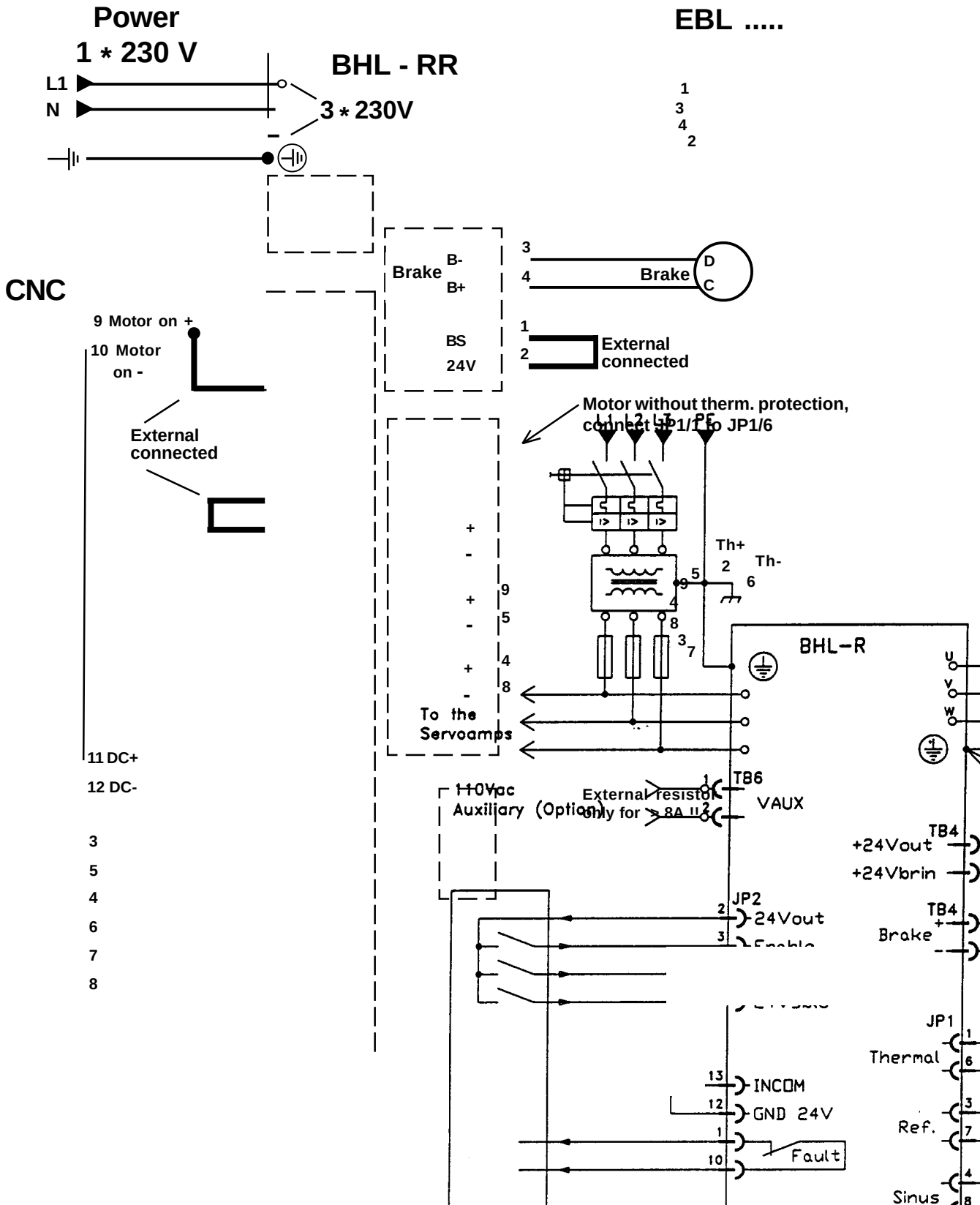
- Switch N790 to servo mode (add the value 16).
  Activate N813X f.e. 3 = 1 + 2 = axes X,Y.
  The axes now can now be moved at low speed, if connector X11.1 (X axis) and X11.2 (Y axis) are correctly cabled.
- Input N700, N706, N707 correctly for each axis.
- By pushing the key „2“ in the MANUAL MODE,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

CONNECTION OF SERVOAMPLIFIERS TYP MSER 043

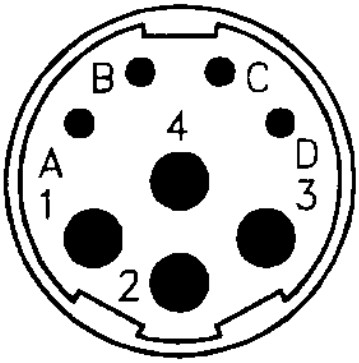


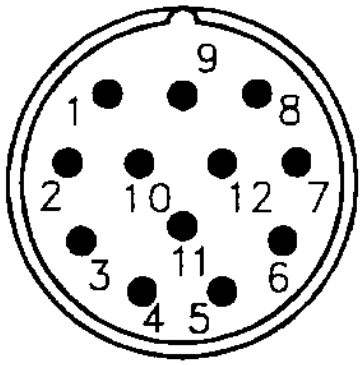
Connection of Servoamplifiers Typ BHL ...

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

X11-4 Spindlemotor / External sync (G33) (for servo system)

pin	signal	pin	signal	SUB-D 15 pol. female
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*			

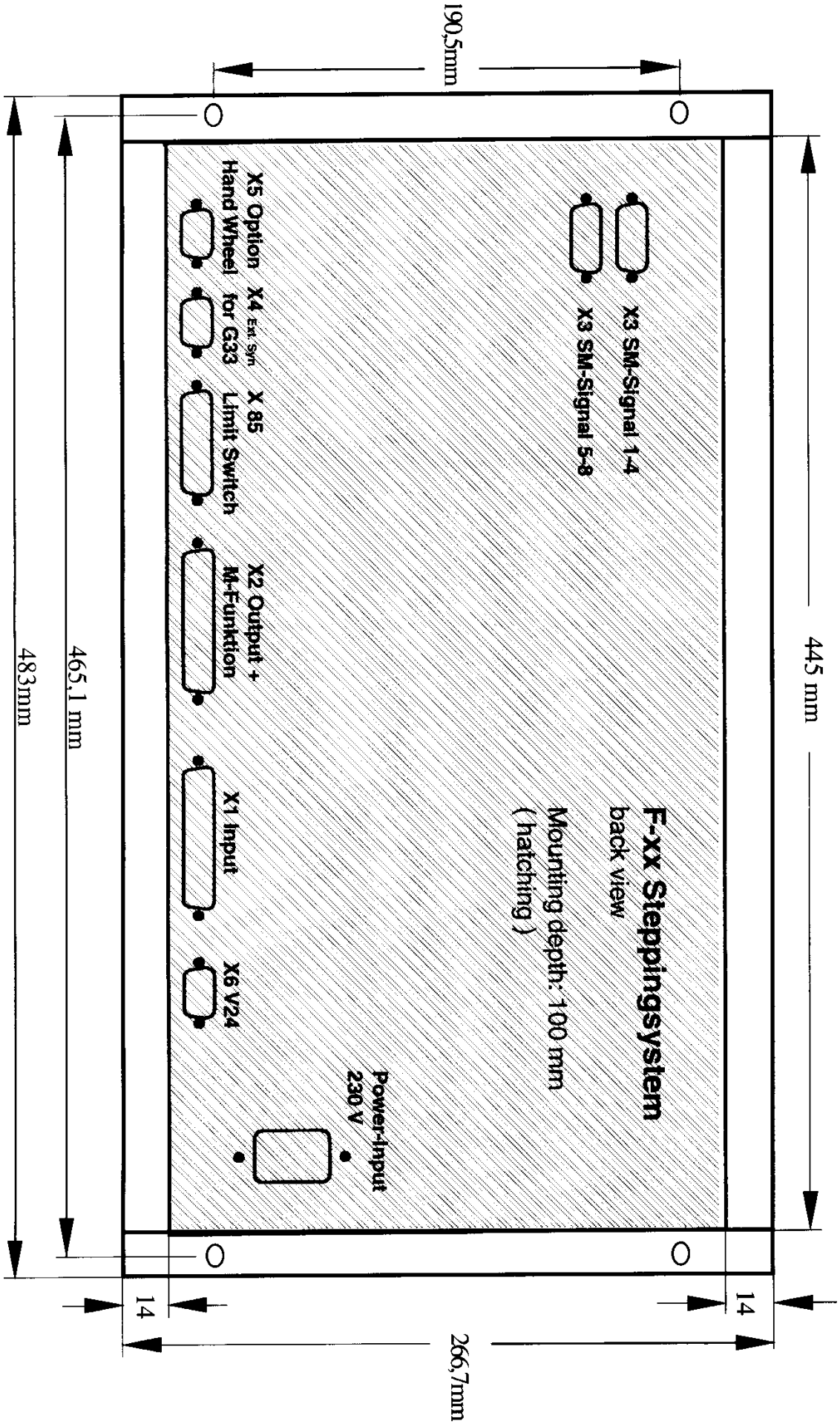
Option :

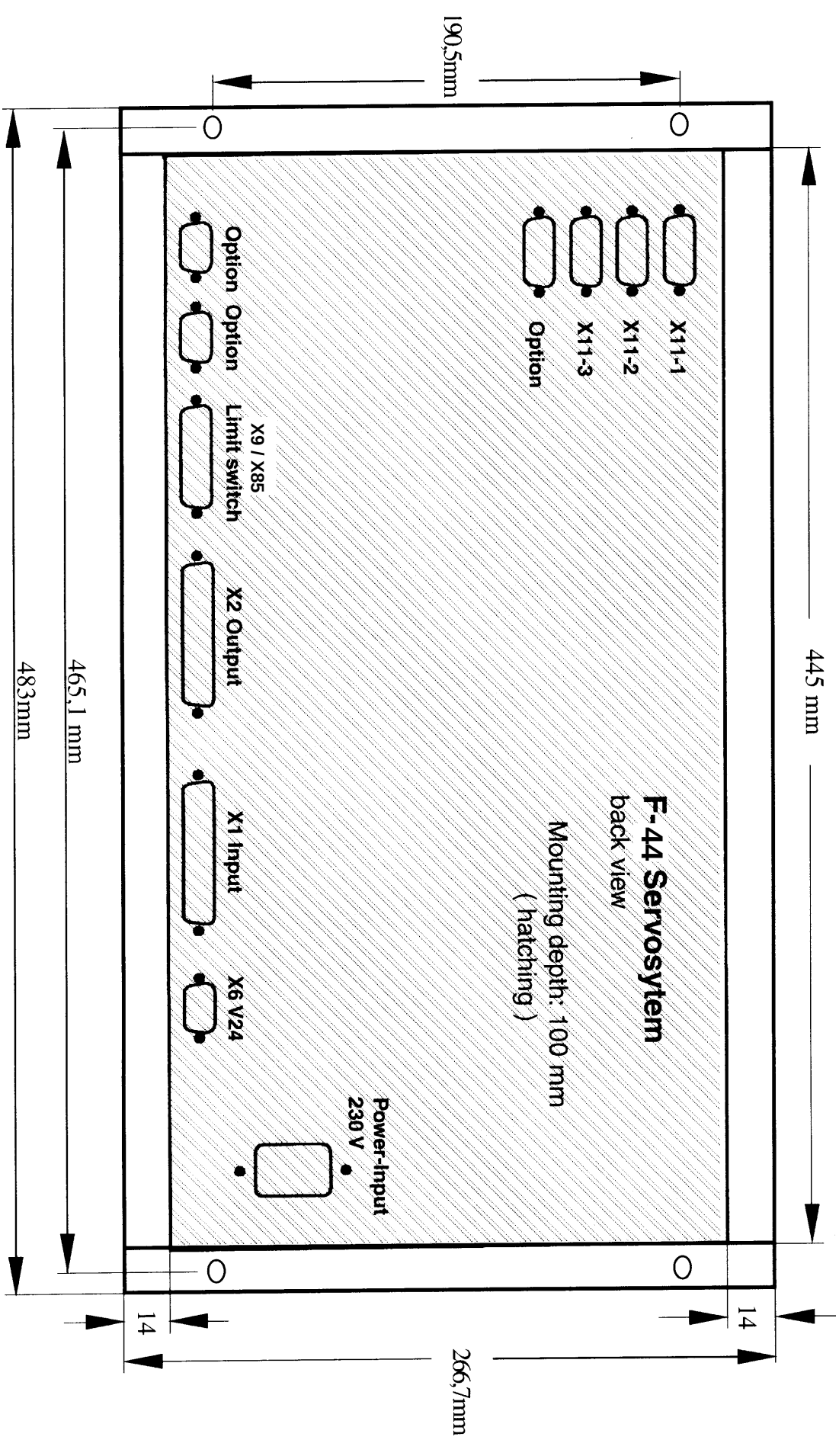
In P0 N900A, this axis must be programmed as spindle axis with the value 3.

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
```


190,5mm
445 mm
F-xx Steppingsystem
back view
X3 SM-Signal 1-4
X3 SM-Signal 5-8
Mounting depth: 100 mm
(hatching)
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
14
14
266,7mm
465,1 mm
483mm


190,5mm
445 mm
483mm
465,1 mm
266,7mm
14
14
F-44 Servosytem
back view
Mounting depth: 100 mm
(hatching)
X11-1
X11-2
X11-3
Option
Option
Option
Option
X9 / X85
Limit switch
X2 Output
X1 Input
X6 V24
Power-Input
230 V

SHIELDING PLAN