CNC3000 Series

CNC3460/3560 M700

Installation manual

4822 873 20418

900330

Numerical control

PHILIPS

CONTENTS

1. SYSTEM CONFIGURATION

1.1. GENERAL SYSTEM CONFIGURATION
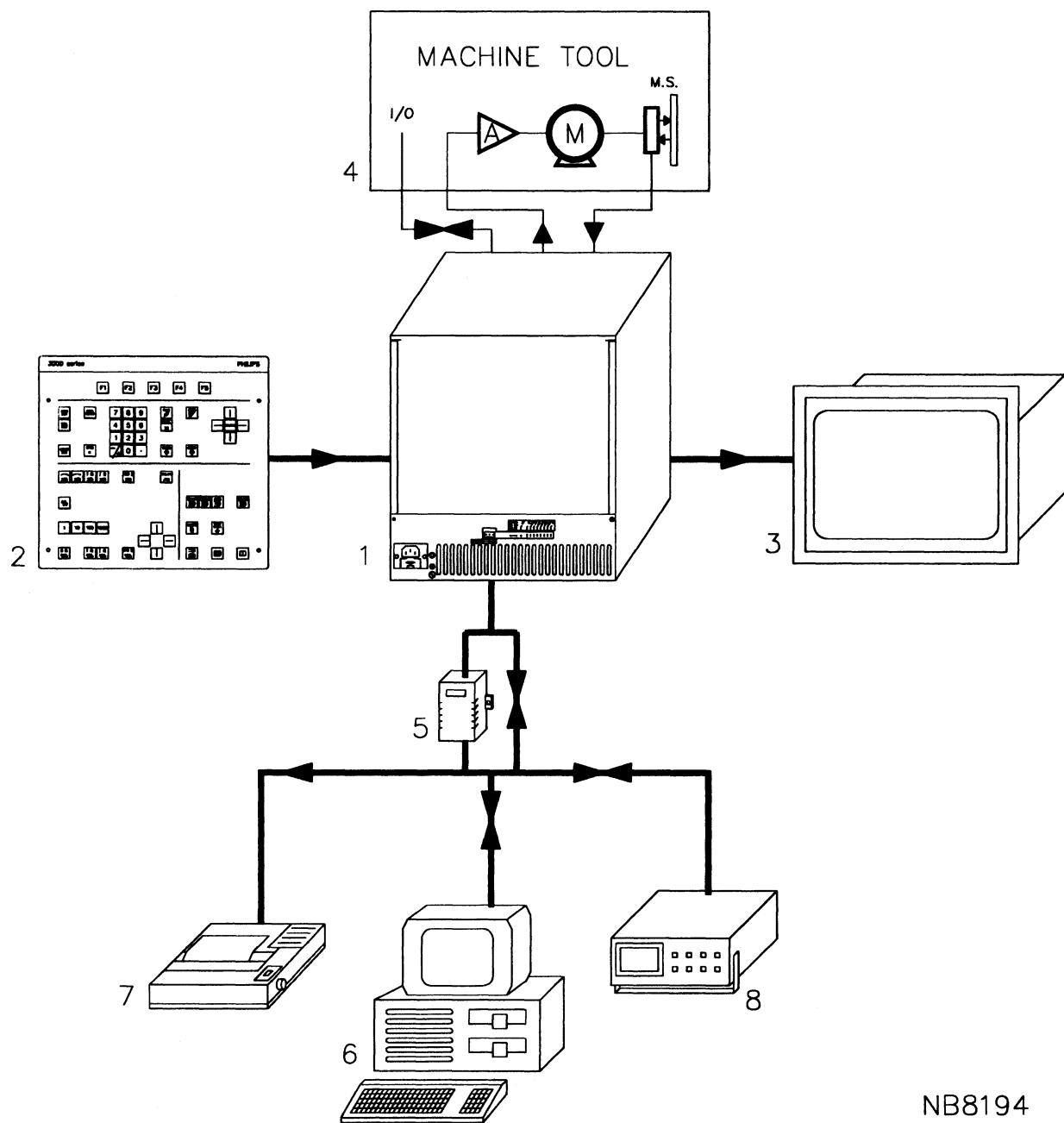
1.-	Control Unit	5.-	V24-to-Current Loop Convertor
2.-	Flat Panel	6.-	Computer System
3.-	Monitor	7.-	Printer
4.-	Machine tool	8.-	Cassette Recorder

Fig. 1.1.-1   **Overview of equipment that may be used in conjunction with the CNC 3460/3560.**
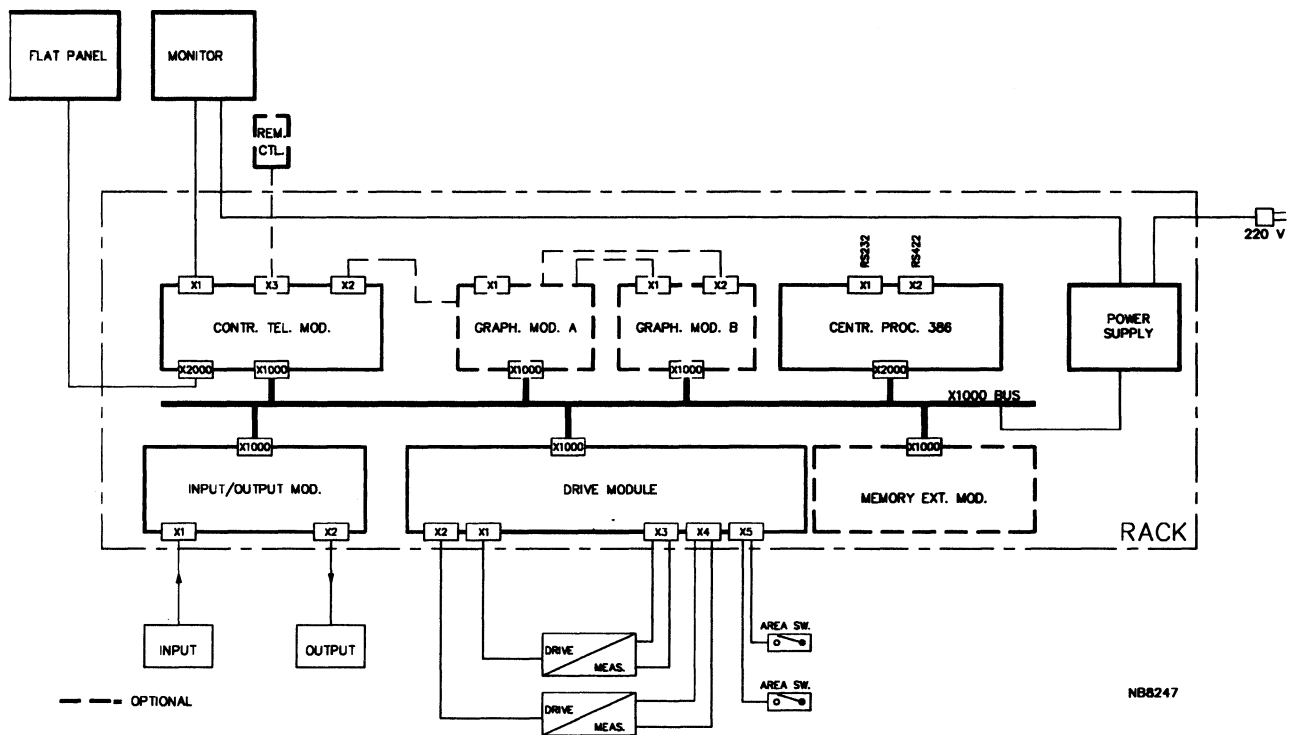
Fig. 1.1.-2   System configuration for the CNC 3460/3560 M700.

1.2. ARRANGEMENT OF THE MODULES IN THE RACK

The number of modules, and therefore the arrangement of the modules, depends on the configuration of the control unit.
The version 700 will be delivered in three different racks; one with 7 slots (positions for modules), one with 15 slots and one with 20 slots.
Make sure that the module screws are securely tightened after replacing a module!

1.2.1. Arrangement of the modules in the 7 slots rack

The configuration with a maximum of modules is given in figure 1.2.1-1.
When the CNC has less features and options a smaller amount of modules is necessary. If for later extensions in the options extra boards are to be added, they can be placed with the help of figure 1.2.1.-1.
Settings on the modules are discussed in the appropriate sections of the installation manual.
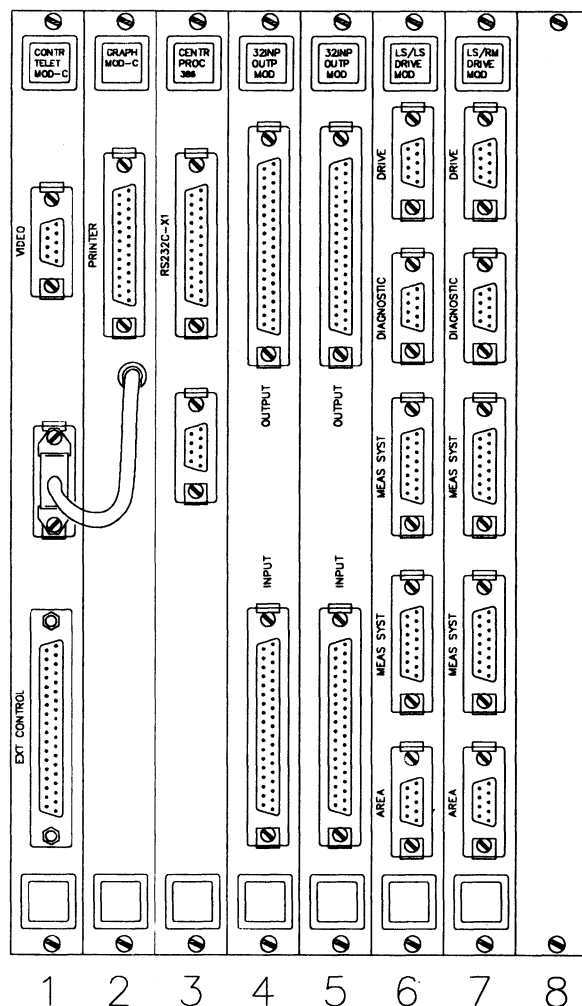
NB8243

Fig. 1.2.1.-1   Maximum configuration of a 7 slots rack.

Pos.	Designation	Abbreviation
1	CONTROL TELETEXT MODULE	CONTR. TELET. MOD-C.
2	GRAPHICS MODULE 2 PLANES	GRAPH. 2P. MOD.
3	CENTRAL PROCESSOR MODULE	CENTR. PROC. 386.
4	INPUT/OUTPUT MODULE 1	32 INP. OUTP. MOD.
5	INPUT/OUTPUT MODULE 2	32 INP. OUTP. MOD. or
5	DRIVE MODULE 1	LS/LS (RM/RM) (LS/RM)
6	DRIVE MODULE 2 (or 1)	LS/LS (RM/RM) (LS/RM)
7	DRIVE MODULE 3 (or 2)	LS/LS (RM/RM) (LS/RM)
8	NOT USED	blind panel.

1.2.2. Arrangement of the modules in the 15 slots rack

The configuration with a maximum of modules is given in figure 1.2.2-1.
When the CNC has less features and options a smaller amount of modules is necessary. If for later extensions in the options extra boards are to be added, they can be placed with the help of figure 1.2.2.-1.
Settings on the modules are discussed in the appropriate sections of the installation manual.
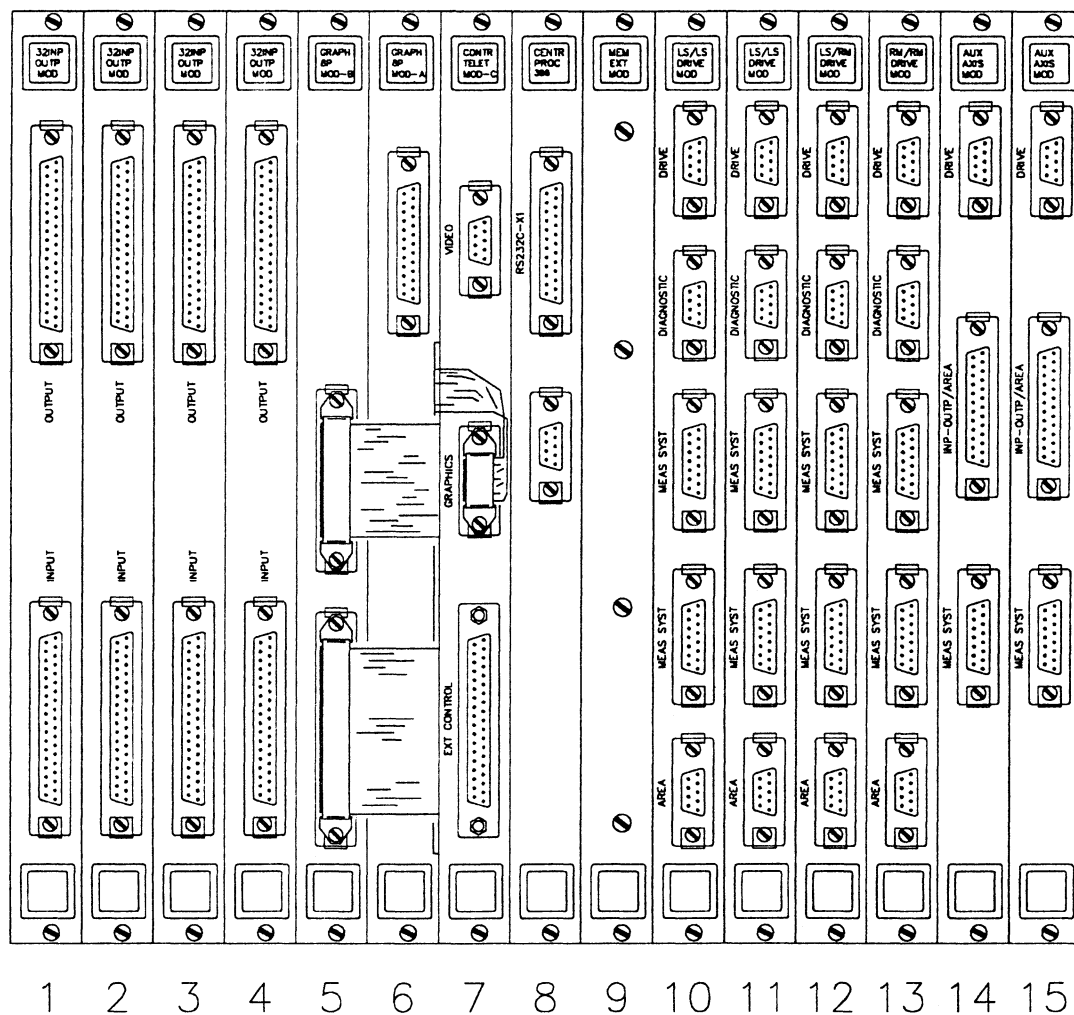
NB8244

Fig. 1.2.2.-1   Maximum configuration of a 15 slots rack.

Pos.	Designation	Abbreviation
1	INPUT/OUTPUT MODULE 1	32 INP. OUTP. MOD.
2	INPUT/OUTPUT MODULE 2	32 INP. OUTP. MOD.
3	INPUT/OUTPUT MODULE 3	32 INP. OUTP. MOD.
4	INPUT/OUTPUT MODULE 4	32 INP. OUTP. MOD.
5	GRAPHICS MODULE B 8 PLANES	GRAPH. 8P. MODB.
6	GRAPHICS MODULE A 8 PLANES	GRAPH. 8P. MODA. or
6	GRAPHICS MODULE 2 PLANES	GRAPH. 2P. MOD.
7	CONTROL TELETEXT MODULE	CONTR. TELET. MOD-C.
8	CENTRAL PROCESSOR MODULE	CENTR. PROC. 386.
9	MEMORY EXTENSION MODULE	MEM. EXT. MOD.
10	DRIVE MODULE 1	LS/LS (LS/RM) (RM/RM)
11	DRIVE MODULE 2	LS/LS (LS/RM) (RM/RM)
12	DRIVE MODULE 3	LS/LS (LS/RM) (RM/RM)
13	DRIVE MODULE 4	LS/LS (LS/RM) (RM/RM)
14	AUXILIARY AXIS MODULE 1	AUX. AXIS. MOD.
15	AUXILIARY AXIS MODULE 2	AUX. AXIS. MOD.

1.2.3. Arrangement of the modules in the 20 slots rack

The configuration with a maximum of modules is given in figure 1.2.3.-1. When the CNC has less features and options a smaller amount of modules is necessary. If for later extensions in the options extra boards are to be added, they can be placed with the help of figure 1.2.3.-1. Settings on the modules are discussed in the appropriate sections of the installation manual.
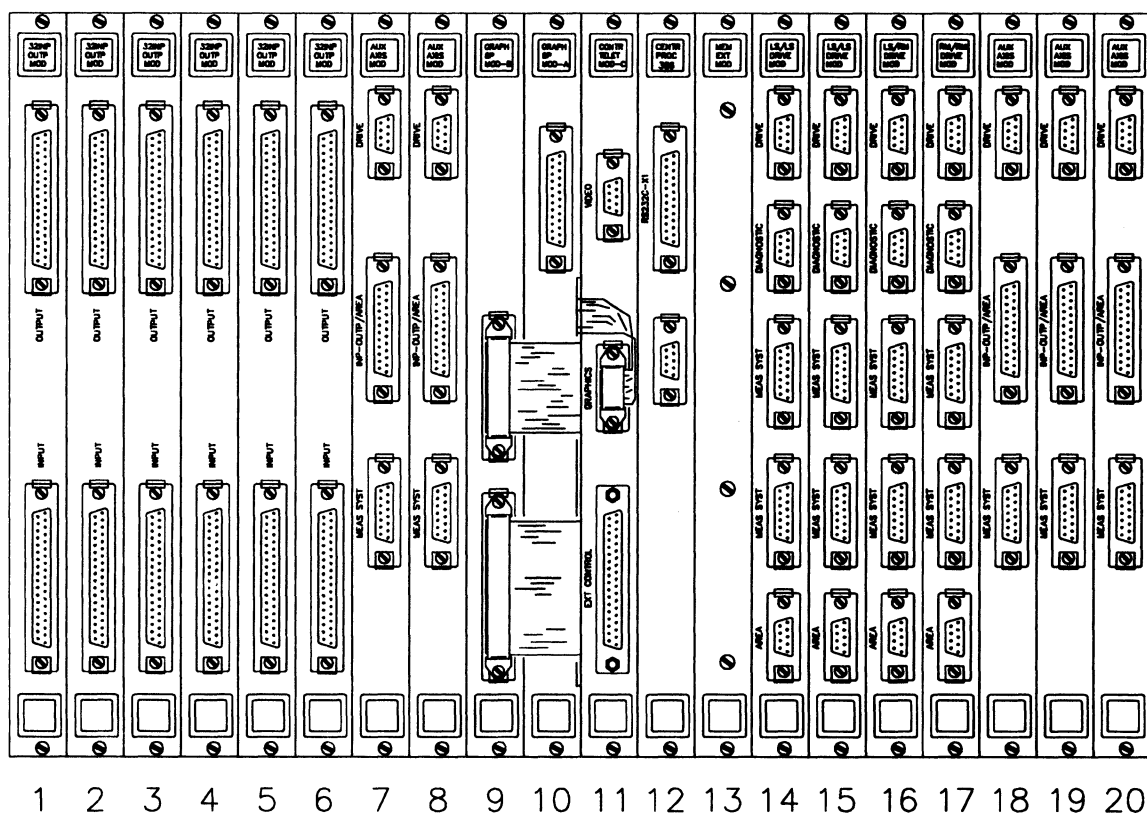
NB8236

Fig. 1.2.3.-1 Maximum configuration for the 20 slots rack

Pos.	Designation	Abbreviation
1	INPUT/OUTPUT MODULE	32 INP. OUTP. MOD.
2	INPUT/OUTPUT MODULE	32 INP. OUTP. MOD.
3	INPUT/OUTPUT MODULE	32 INP. OUTP. MOD.
4	INPUT/OUTPUT MODULE	32 INP. OUTP. MOD
5	INPUT/OUTPUT MODULE	32 INP. OUTP. MOD.
6	INPUT/OUTPUT MODULE	32 INP. OUTP. MOD. or
6	AUXILIARY AXIS MODULE	AUX. AX. MOD.
7	INPUT/OUTPUT MODULE	32 INP. OUTP. MOD. or
7	AUXILIARY AXIS MODULE	AUX. AX. MOD.
8	INPUT/OUTPUT MODULE	32 INP. OUTP. MOD. or
8	AUXILIARY AXIS MODULE	AUX. AX. MOD.
9	GRAPHICS MODULE B 8 PLANES	GRAPH. 8P. MOD B.
10	GRAPHICS MODULE A 8 PLANES	GRAPH. 8P. MOD B. or
10	GRAPHICS MODULE 2 PLANES	GRAPH. 2P MOD.
11	CONTROL TELETEXT MODULE	CONTR. TELET. MOD. C
12	CENTRAL PROCESSOR MODULE	CENTR. PROC. 386
13	MEMORY EXTENSION MODULE	MEM. EXT. MOD.
14	DRIVE MODULE 1	LS/LS (LS/RM) (RM/RM)
15	DRIVE MODULE 2	LS/LS (LS/RM) (RM/RM)
16	DRIVE MODULE 3	LS/LS (LS/RM) (RM/RM)
17	DRIVE MODULE 4	LS/LS (LS/RM) (RM/RM)
18	AUXILIARY AXIS MODULE	AUX. AX. MOD.
19	AUXILIARY AXIS MODULE	AUX. AX. MOD.
20	AUXILIARY AXIS MODULE	AUX. AX. MOD.

2.   MOUNTING THE CNC3460/3560 M700

All the measures given in this section are in millimeters.

Leave above the rack a free space of 100 mm to ensure sufficient cooling.

The plugs with the cables need 150mm free space at the front of the rack.

Do not cover the air inlet openings at the bottom front!

2.1.   MOUNTING THE 7 SLOTS RACK

NB7897   NB7896

The weight of the 7 slots rack including the modules is appr. 8 kg.

Fig. 2.1.-1.   Dimensions of the 7 slots rack.

2.2. MOUNTING THE 15 SLOTS RACK
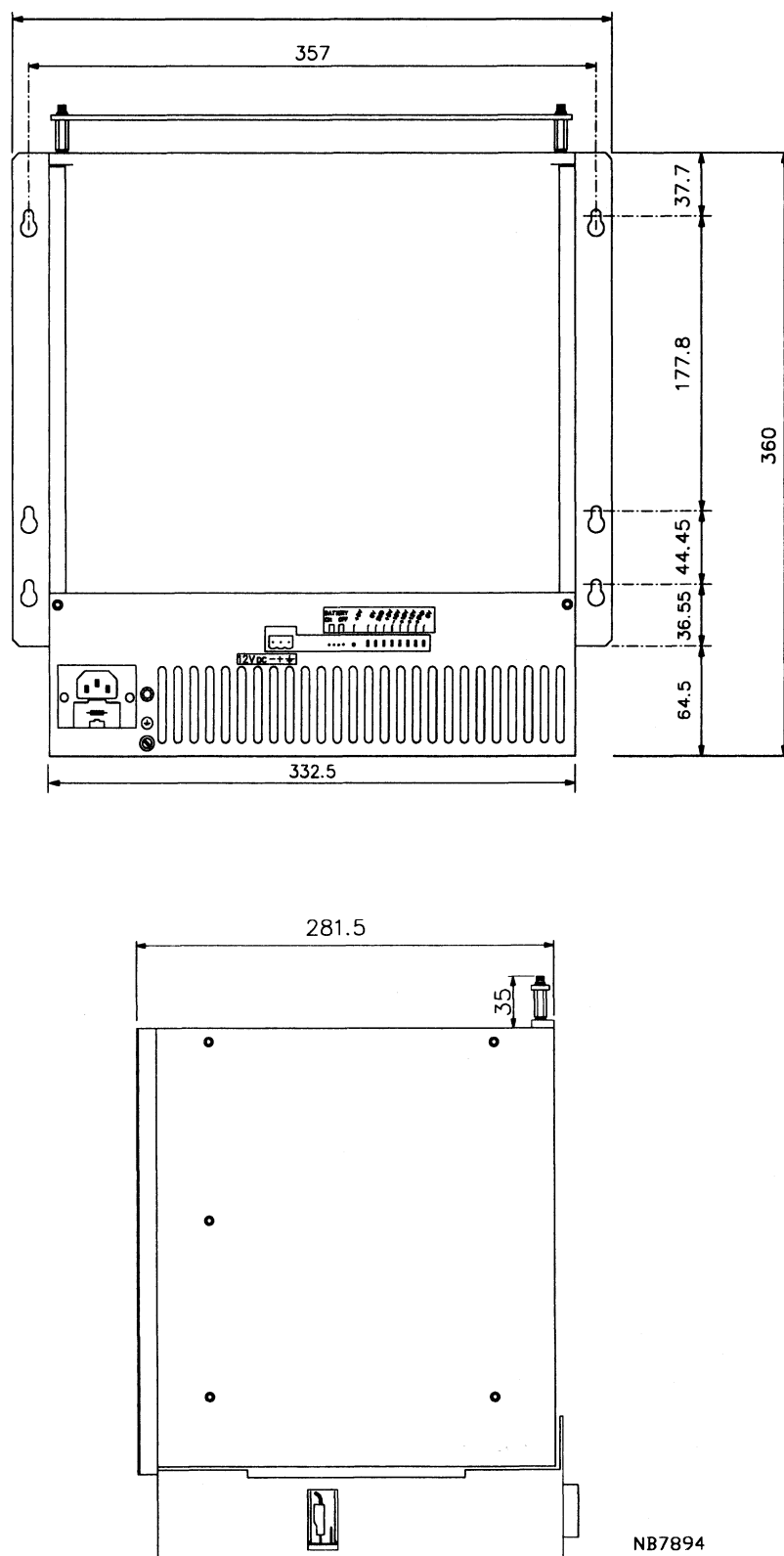
The weight of the 15 slots rack including the modules is appr. 15 kg.

Fig. 2.2.-2. **Dimensions of the 15 slots rack.**

2.3. MOUNTING THE 20 SLOTS RACK
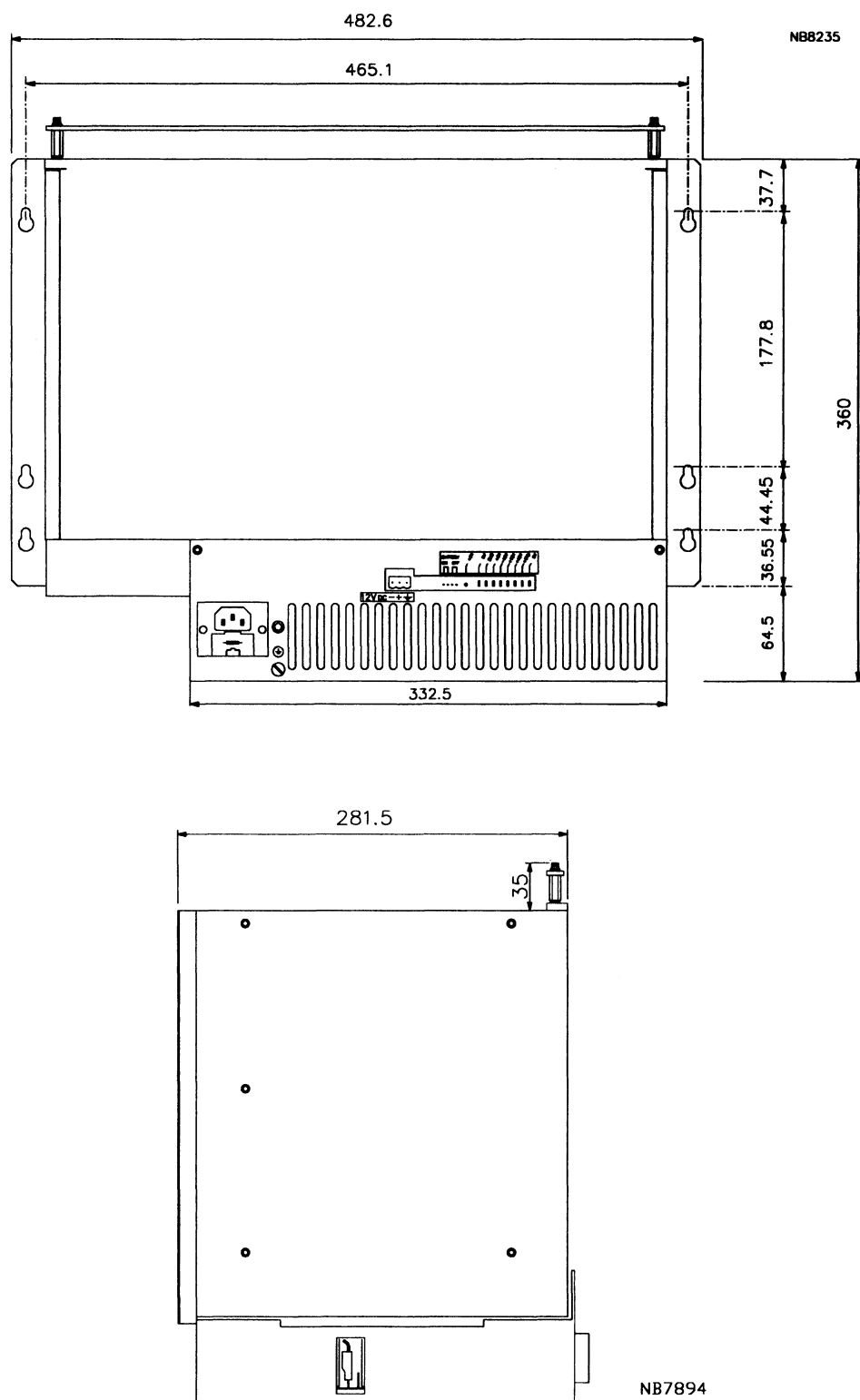
The weight of the 20 slots rack including the modules is appr. 20 kg.

Fig. 2.3.-1. **Dimensions of the 20 slots rack.**

2.4. MOUNTING THE COLOUR MONITOR
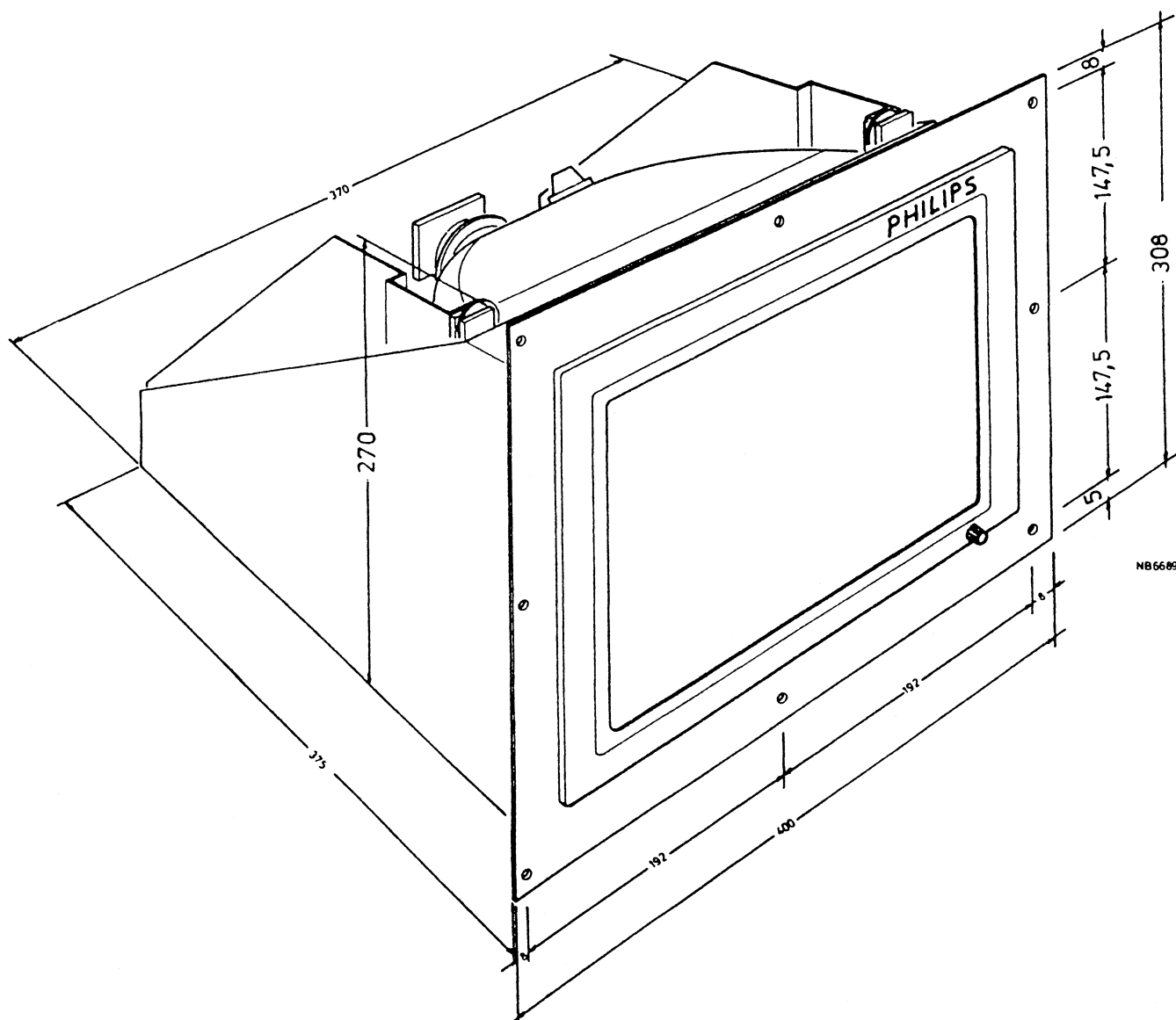
The weight of the colour monitor is appr. 7 kg.

Fig. 2.4.-1. **Dimensions of 14" colour monitor**

8 x M5

295

283

370

384
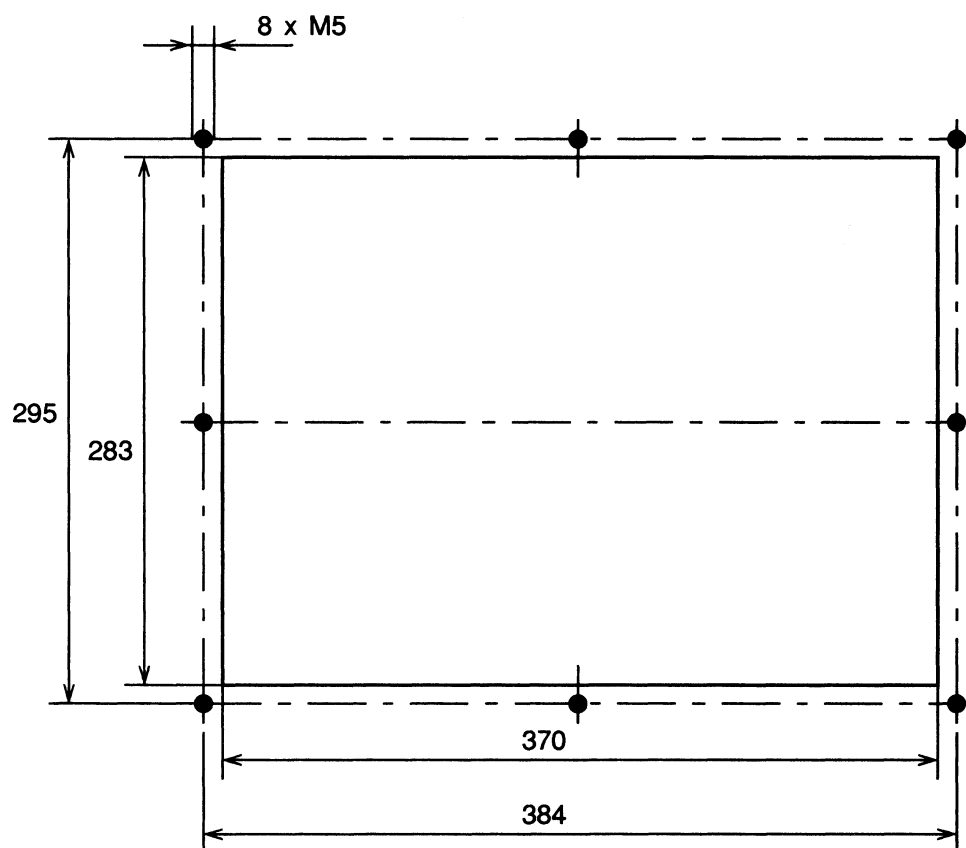
NB7622

The mounting depth of the 14" colour monitor is 380 mm.

Fig 2.4.-2.   **Mounting dimensions of 14" colour monitor.**

2.5. MOUNTING THE MONOCHROME MONITOR
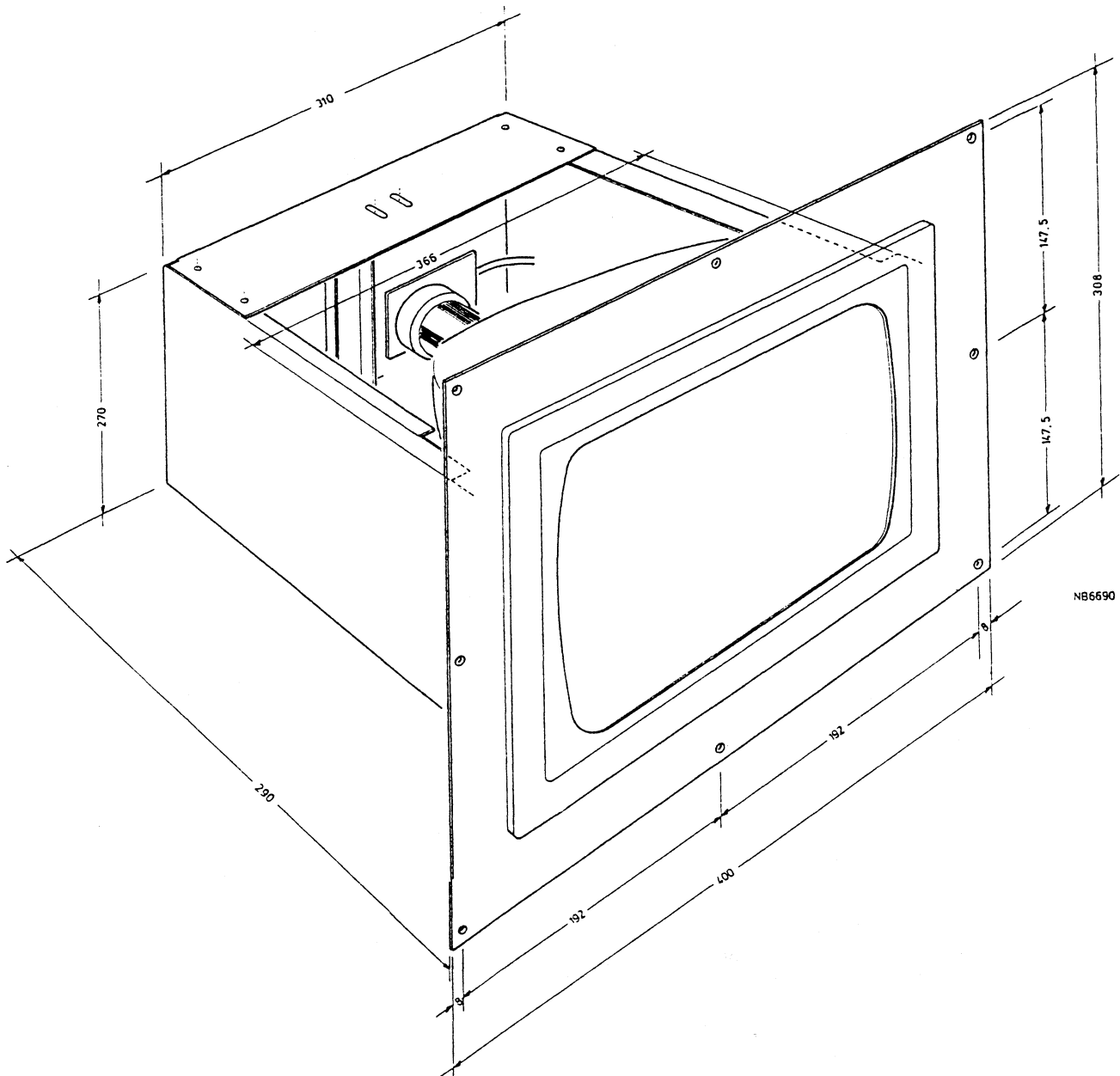
The weight of the monochrome monitor is appr. 7 kg.

Fig. 2.5.-1. **Dimensions of the 12" monochrome monitor.**

8 x M5

295

283

370

384
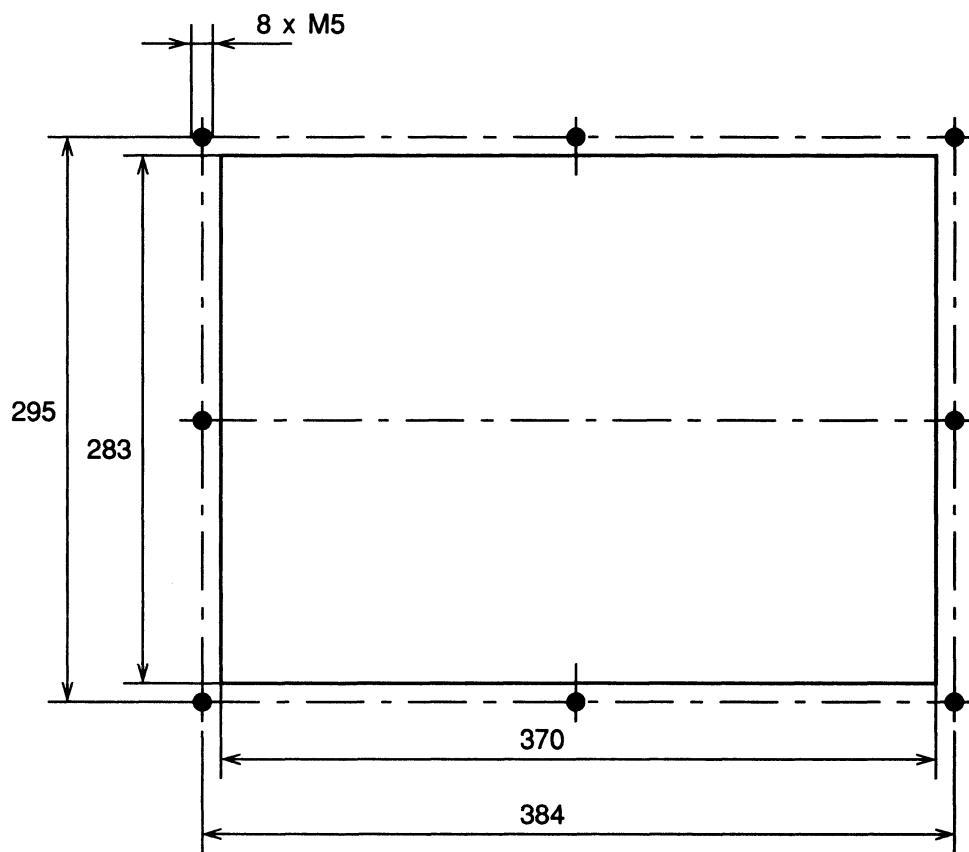
NB7622

The mounting depth of the 12" monochrome monitor is 380 mm.

Fig. 2.5.-2. **Mounting dimensions of 12" monochrome monitor.**

2.6. MOUNTING THE OPERATING PANEL

The CNC3460 operating panel can be mounted with or without a mounting frame.

Fig 2.6.-1.   **CNC3460 operating panel dimensions (without frame)**
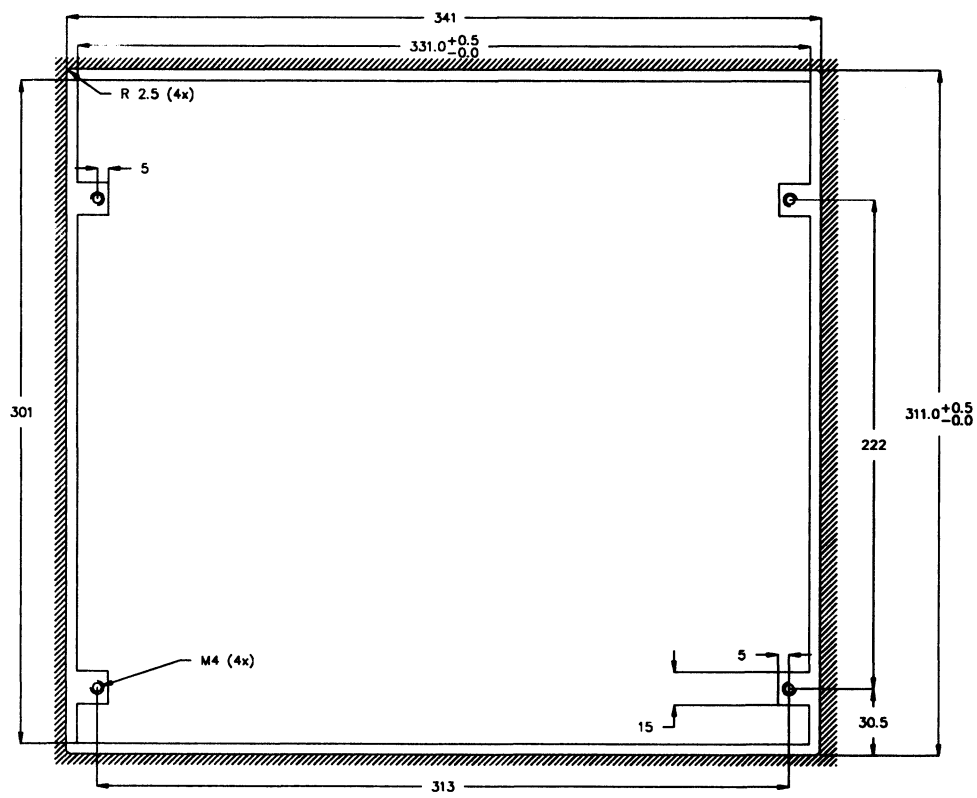
MOUNTING DIMENSIONS FLAT PANEL (WITHOUT WINDOW)

Mounting depth of connector is 50 mm

Fig 2.6.-2. **Mounting dimensions of operating panel (without frame)**

MOUNTING DIMENSIONS WINDOW FOR FLAT PANEL

NB8081
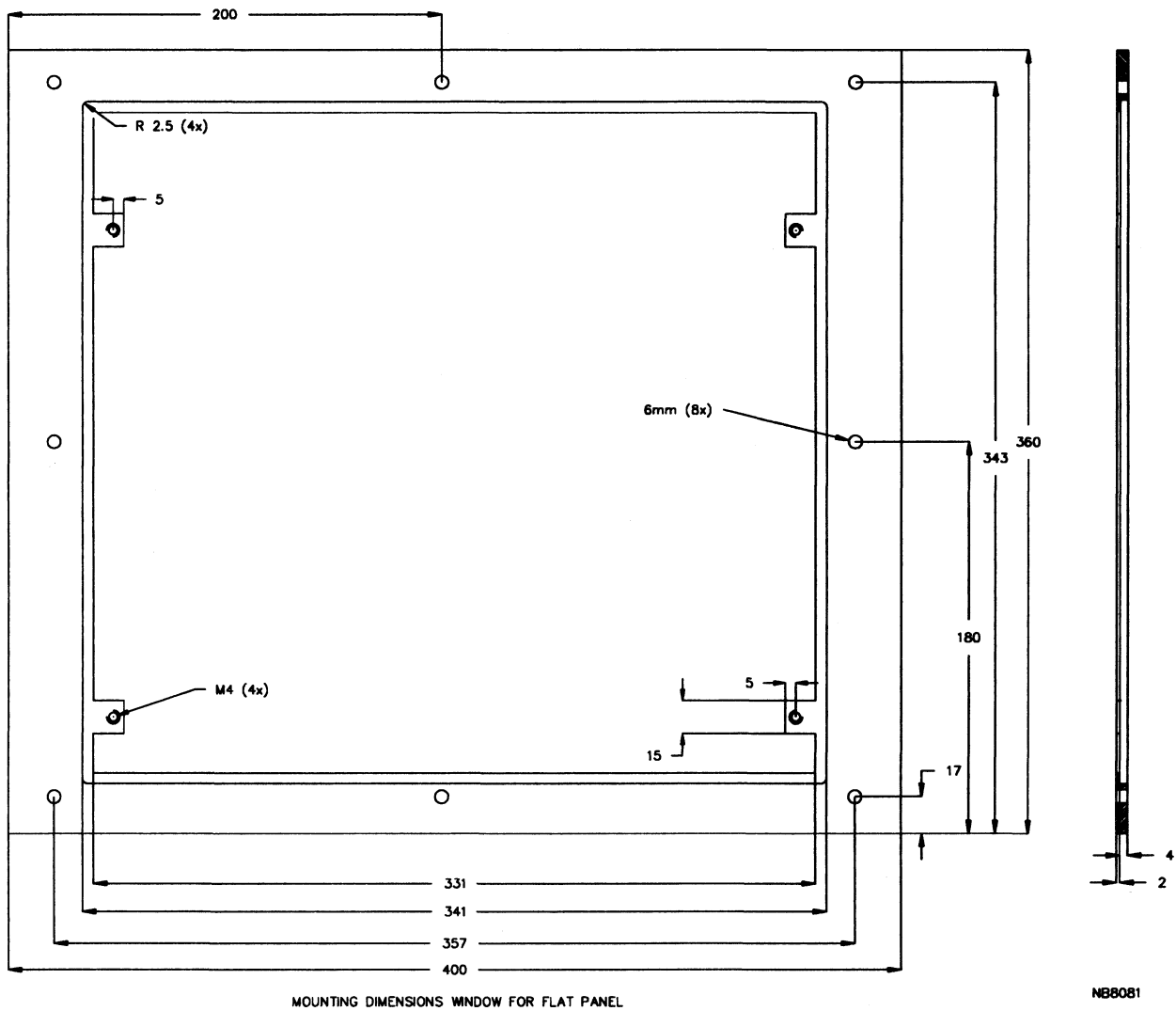
Fig 2.6.-3.   **Mounting dimensions of the mounting frame**

2.6.1   INSTALLING THE FLAT PANEL

When the complete CNC system is installed the flat panel must be initialized when the system is switched on the first time. The following procedure must be followed:

-    While keeping softkey F1 depressed switch on the CNC

Now there are two possibilities:

1)   The text "PRESS COOLANT TO CONTINUE" appears.
       Depress the coolant button and the CNC continues the power up check.

2)   The text "PRESS COOLANT ELSE OPER" appears.
       If the operating panel contains a coolant button press this button, only when one is sure that this button does not exist press the oper button.
       The control continues the power up check.

Do not depress any other button during switch on. This will lead to a "FAILED" announcement during the power up check and so new installation of the operating panel is performed.

2.7.   COOLING

The CNC3460 7 slots cabinet is cooled by one fan, the 15 slots cabinet with two fans, and the 20 slots rack is cooled by three fans. In all cases they are located in the power supply unit at the bottom of the cabinet.
The openings at the front bottom provide the air inlet, dust filters are not provided. In principle the interface cabinet in which the rack is mounted must have sufficient dust filtering. If this is not the case, dust filters can be mounted on the rack, but make sure that the coolant air flow is not obstructed.
Leave a free space of 150 mm in front of the air inlet openings.
The air outlet is done via the openings at the top of unit.
Leave a free space of 100 mm to allow sufficient air flow.

The ambient temperature in the interface cabinet in which the CNC unit is build must not exceed 55 degrees Celsius.

3. EARTHING

3.1. GENERAL INFORMATION ABOUT EARTHING

The numerical control and the peripheral equipment connected to it, such as measuring system, servo drives, digital I/Os, video monitor and data peripherals can be represented in a block diagram.
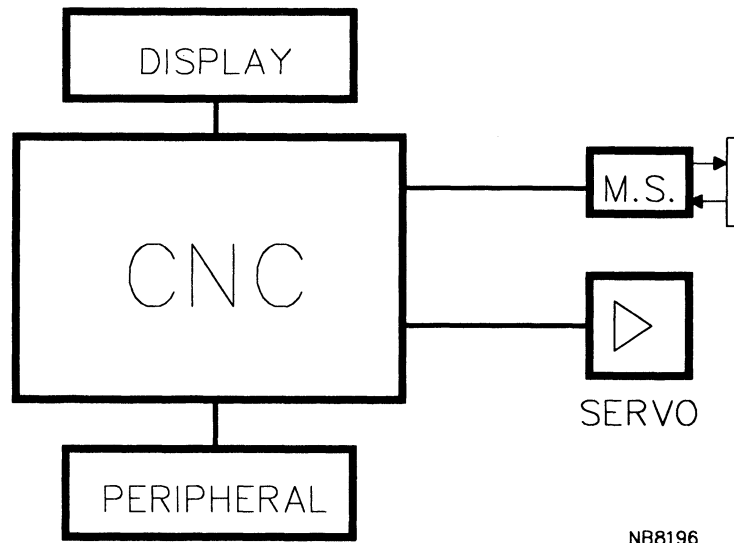
NB8196

Fig. 3.1.-1.   Block diagram of CNC with peripheral equipment

Signal transfer between control and peripheral occurs via one or more signal cables. A connection like this, presented as a block diagram, is illustrated in figure 3.1.-2. The signal generator is shown on the left hand side, the signal receiver on the right hand side.
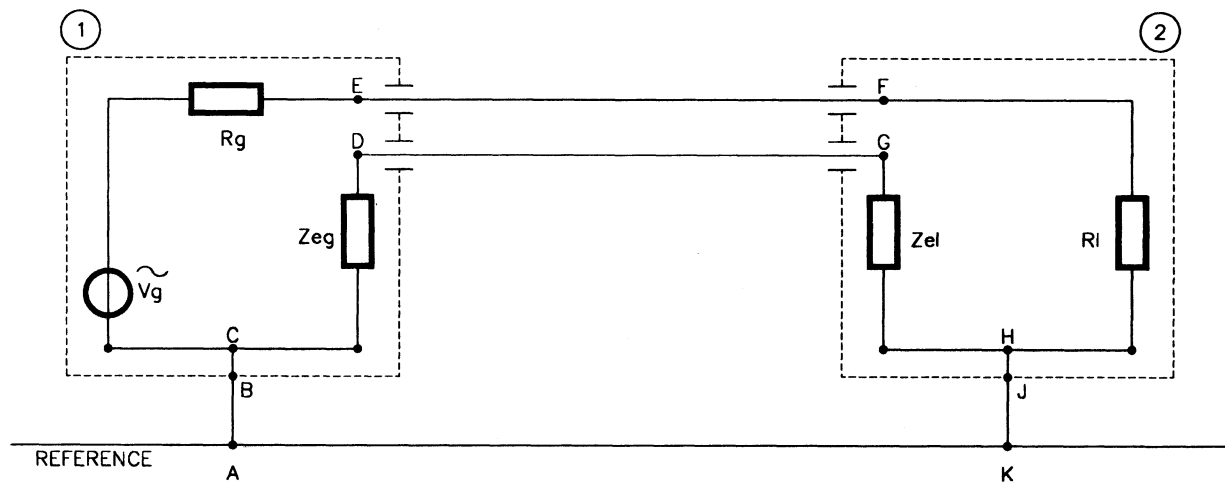
NB7817

Fig. 3.1.-2.   Block diagram of a connection

The signal generator consists of a signal source with open voltage V_g, internal resistance R_g and ground impedance Z_{eg} between zero of the voltage source and the return lead of the cable.

The resistor consists of a load resistance R_L and an impedance Z_{EL} between zero of the load resistance and the return lead of the cable. An interference field is produced by switching relays, thyristors, motors, engaging brakes, switching valves etc.
The circuitry illustrated in figure 3.1.-2 is "seen" by the interference field as two loops, into which it will induce an interference voltage.

As a result, an interference current will flow through these loops. These current loops are visualized in figure 3.1.-3 and 3.1.-4.
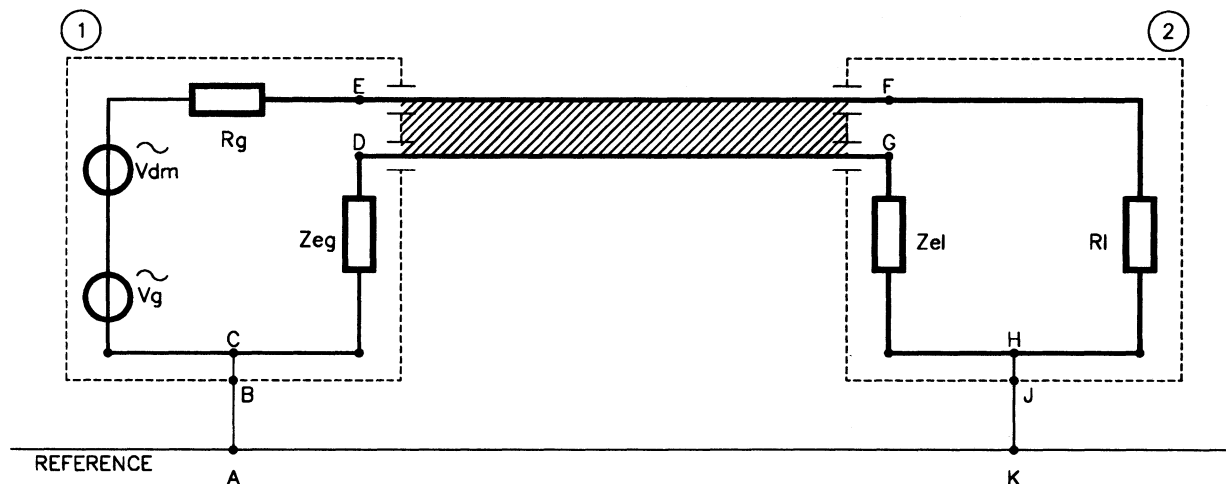
NB7820

Fig. 3.1.-3.   Interference current in the signal circuit
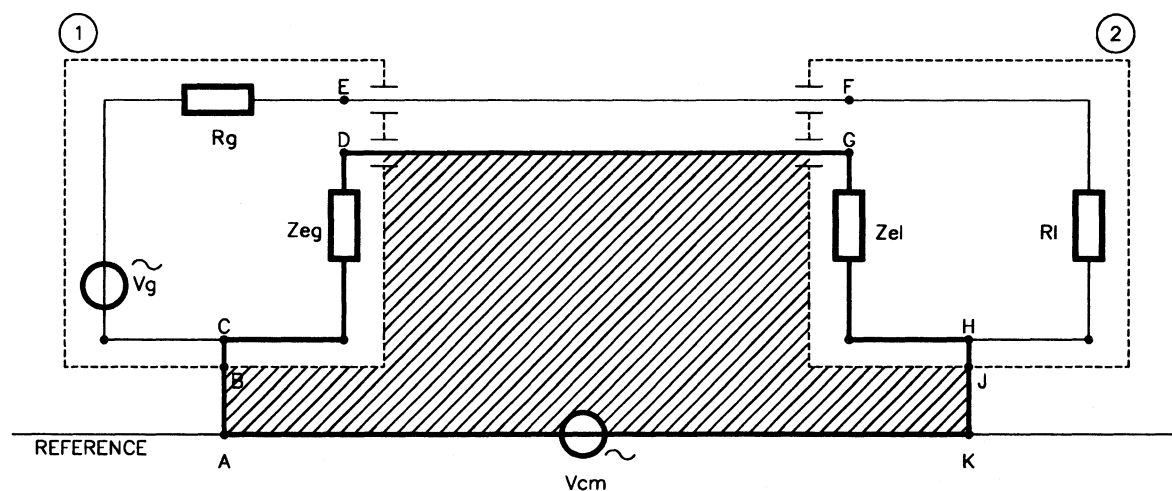
NB7824

Fig. 3.1.-4.   Interference current in the reference circuit

The interference current in the signal circuit (loop CEFHGD) is called "differential mode" interference, the one in the reference circuit (loop ABCDGHJK) "common mode" interference. They are represented by signal sources V_{DM} and V_{CM} respectively.

Certain rules are to be observed in order to limit these interference currents and enable a proper signal transfer.
First the possibilities on how to limit these interference influences will be outlined, after which chapter 3.1.2. provides some general directives. Chapter 3.1.3. describes some specific practical matters.

3.1.1.   Limiting interference influence

"Differential mode" interference is caused by radiation of the interference source into the signal circuit CEFHGD.
This interference source may be a cable adjacent to the signal circuit or electromagnetic fields.
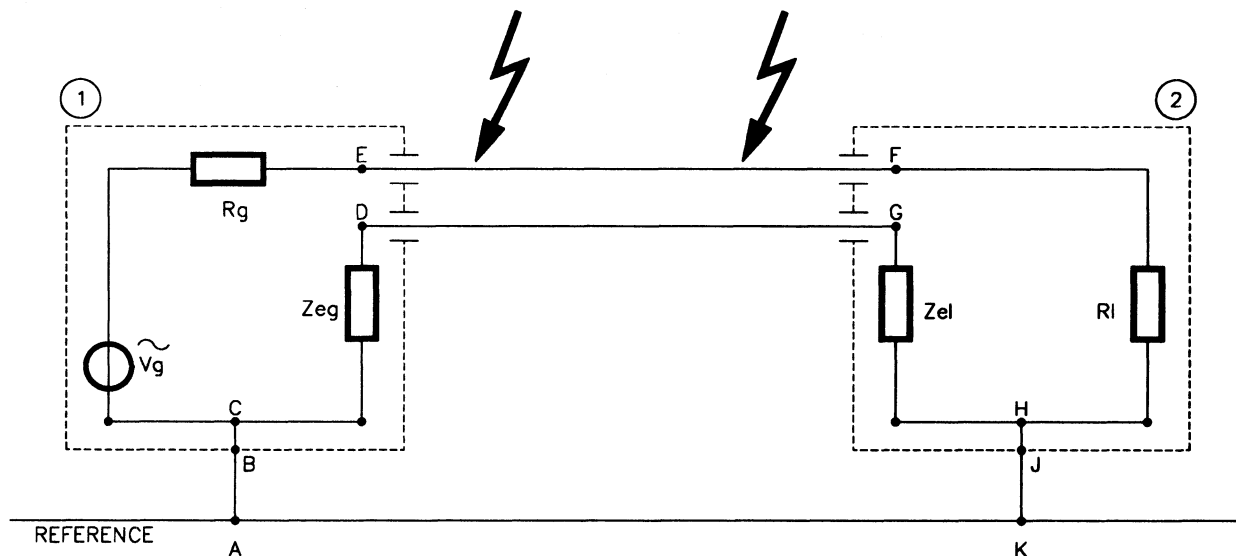
NB7818

Fig. 3.1.1.-1.   Generation of "differential mode" interference

The influence of the current source can be suppressed by providing the connection cables with a screen, which is "optically closed" to the interference field and connected to the reference on one side.
"Optically closed" means that the current source does not see the signal lead, only the screen.
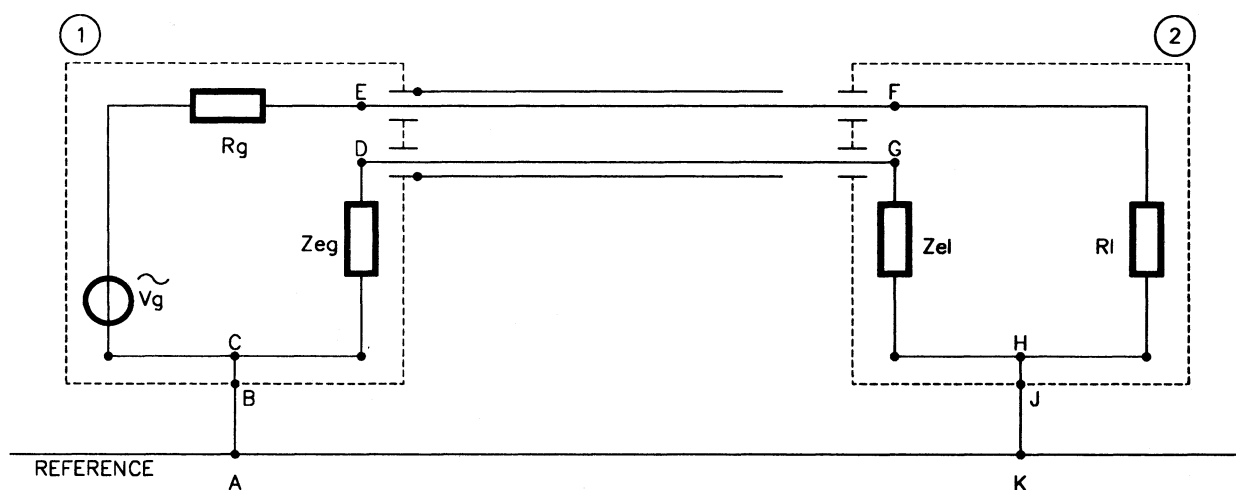
NB7814

Fig. 3.1.1.-2.   Screen connected to the reference on one side

This does however not present an adequate solution to the interference current in the reference circuit (ABCDGHJK). The "common mode" current (V_{CM}) still flows through the signal return lead, and introduces undesired interference in the signal circuit. Consequently, the screen is required to be always connected to the reference **on both sides**.
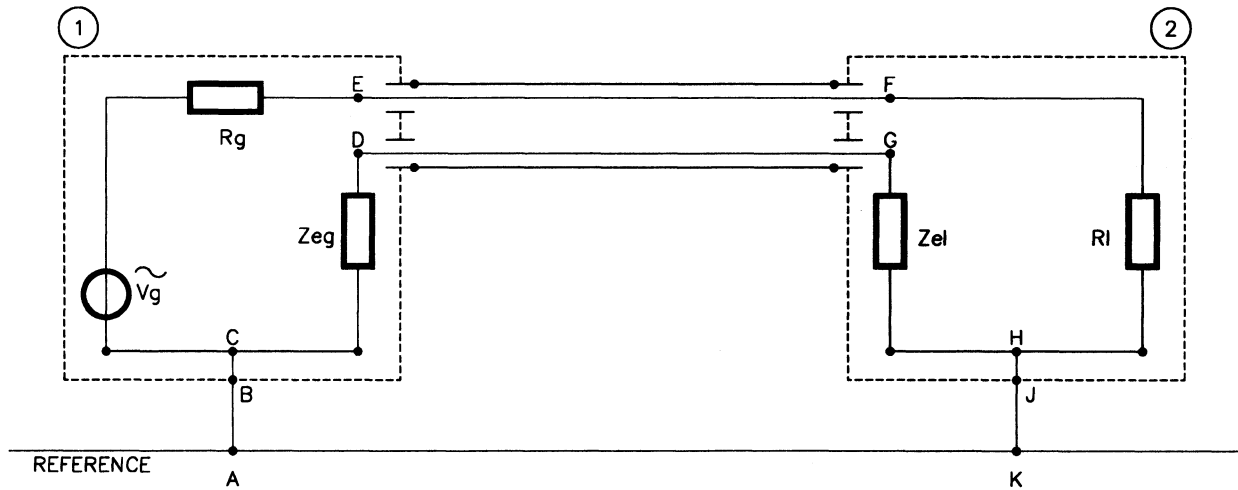
NB7815

Fig. 3.1.1.-3. **Screen connected to the reference on both sides**

In practice this proves to be an adequate shield for the signal circuit against capacitive interference sources. Magnetic interference however will generally not be stopped by screening, so that a signal connection in the vicinity of an interference source may still suffer from interference. Since switching relays, motors, engaging brakes and switching valves represent strong magnetic interference sources, they may still affect the signal circuit.

The influence of magnetic interference sources can be substantially suppressed by twisting the signal and signal-return leads.
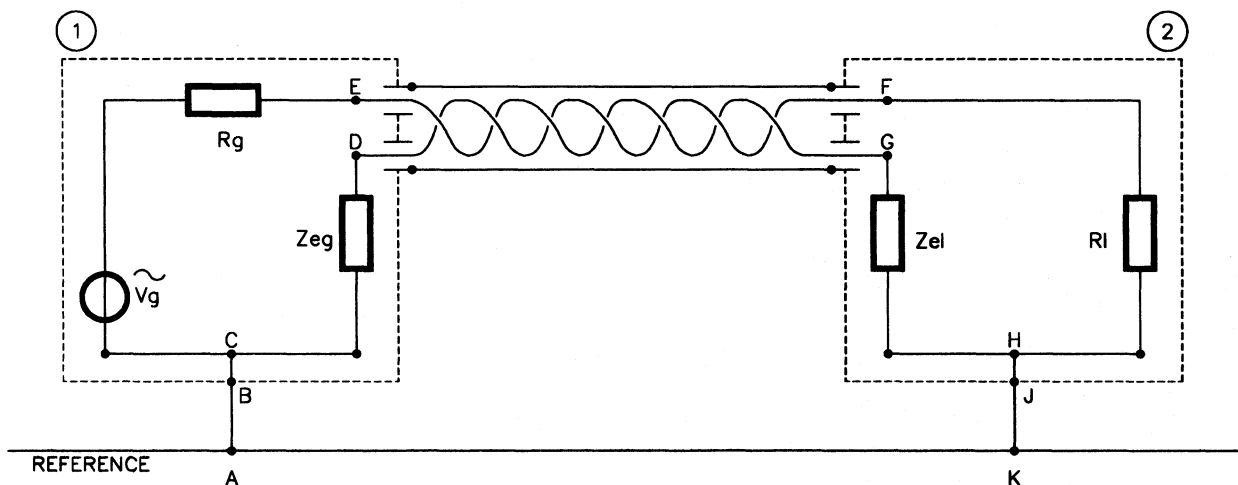
NB7816

Fig. 3.1.1.-4. **Signal and signal-return leads twisted**

If one side of the screen is not properly connected, this spot will present a transition impedance (Z_R), as indicated in figure 3.1.1.- 5. This impedance Z_R will also manifest itself for high-frequency interference signals in the event of too poor a screening being connected to the reference circuit via a relatively long lead. The "common mode" interference current will take the line of least resistance and flow through the signal-return lead (signal path ABCDGHJK). Too poor a connection of the screen or too long a lead from the screen to the reference should therefore be avoided.
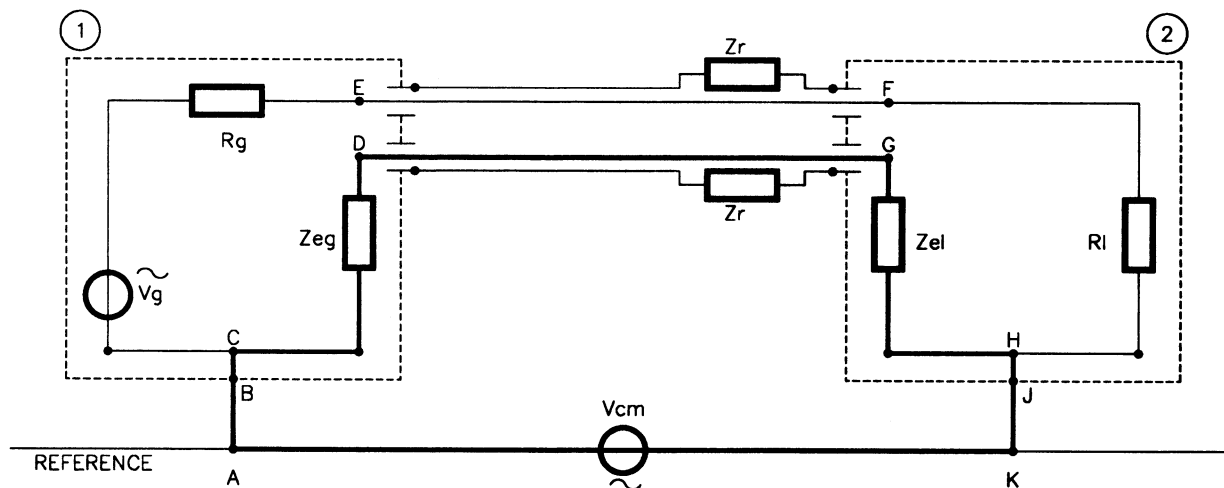
NB7819

Fig. 3.1.1.-5. **Path of the interference current when one side of the screen is poorly connected to the reference or via too long a lead**

3.1.2. General instructions to prevent interference

In order to avoid problems which may arise from the interference susceptibility of the overall system, the following points should be observed:

1. Always connect both sides of the cable screen to the case of the equipment. Also see to it that the screen be connected in accordance with the instructions given in the installation manual.

2. Always use the type of cable specified for a given connection, for this cable will have the properties required for a proper connection, for example an "optically closed" screening or a good coupling between the conductor pairs. Therefore always connect the conductor pairs as specified.

3. Ground the entire system, i.e. each metal case must have at least one connection to the ground system. Always arrange the ground connections to be shaped as a star, that is each point to be grounded must have its own connection to the central ground point.

4. See to it that the front sides of the pc boards are secured to the rack by screws, for in this way the signal return lead is connected to the rack via a filter and hence to system ground. If the front sides are not secured by screws, the filter action will be suppressed.

3.1.3. Exceptions CNC3000

All peripherals for the CNC3000 series must be connected to the rules given in the previous paragraph, with the exception of the LS- transducer and the serial interfaces RS232 and RS422.

Grounding the LS-transducer

The block diagram for the linear transducers, when taking into account the rules 1, 2 and 3 from the previous paragraph, looks as follows:
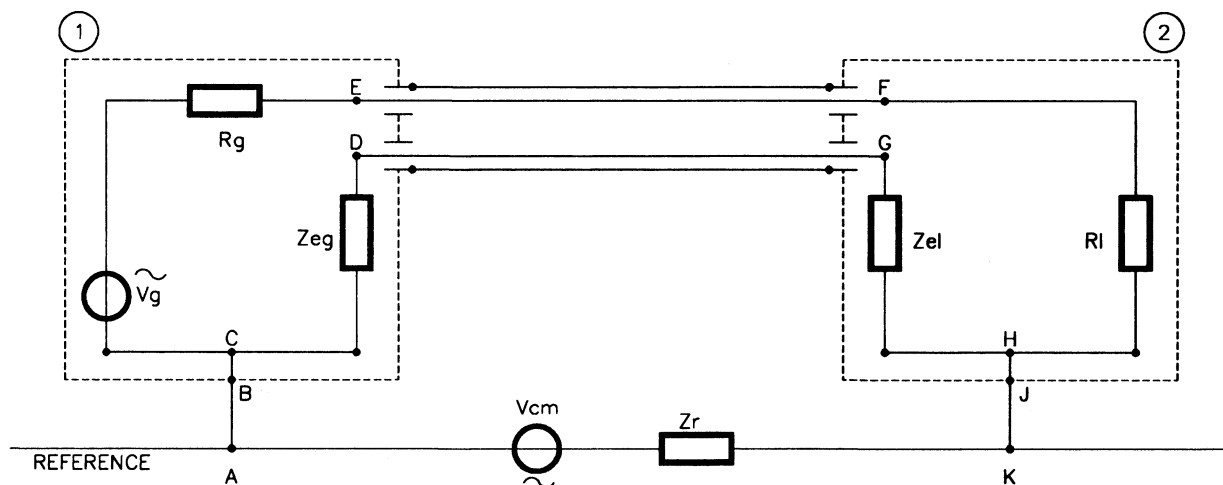
NB7821

Fig. 3.1.3.-1. LS-transducer connected to the rules

The transducer is generally mounted to a part of the machine-tool that lies outside the interface cabinet and is moving. This will give a (poor and) long connection to the reference that may pick up a lot of interference (impedance Z_r in figure 3.1.3.-1).

Current source V_{cm} will produce a high voltage between the points A and K. As a result, the screen current will be large and may cause cross talk between the screen and the two conductors. Consequently, interference may occur in the signal circuit.

The effect on high frequencies is much less, since the impedance Z_r to high frequencies is much lower and hence the screen current. In order to keep low-frequency interference out of the signal circuit, the transducer must be connected as shown in figure 3.1.3.-2.
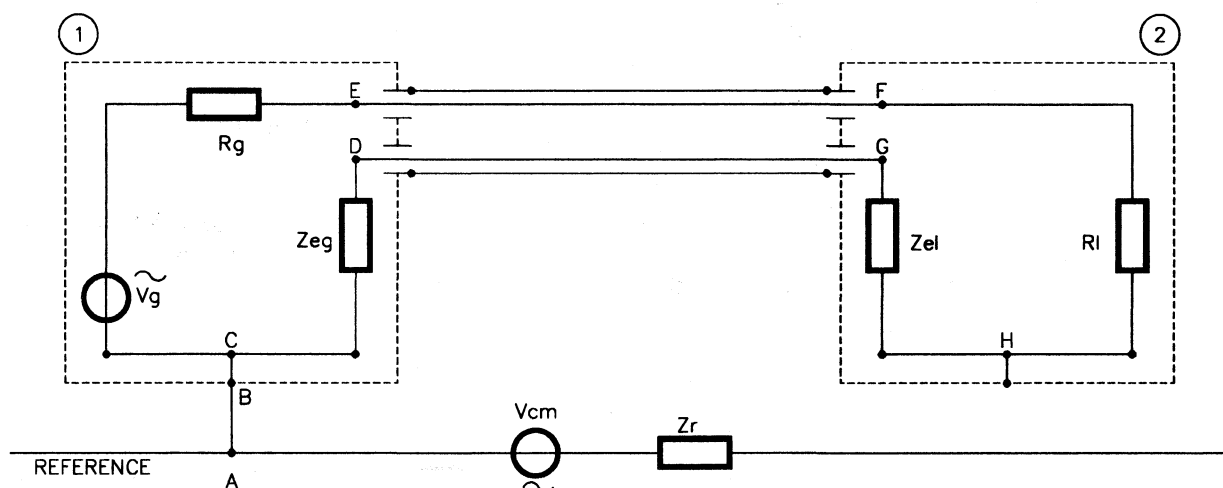
NB7822

Fig. 3.1.3.-2 Block diagram of the LS-transducer connection

Low-frequency interference currents cannot flow through the screen, due to the galvanic separation between the points J and K. For higher frequencies, the current sees a capacitance between the points J and K, which is short-circuited via the impedance of the screen.

RS232/RS422 connection

The problems of the serial interface connection lie in the fact that it is unknown what is happening on the peripheral side. The peripheral may have a power supply which causes interference. The mains supply of the peripheral may be strongly interfered. The grounding of the peripheral may be poor. The cable screen on the peripheral side may not be grounded.

As the signal return lead carries large currents, the voltage drop across Z_{eg} will become too large very soon.

In order to reduce Z_{eg} and therefore be less dependant on the peripheral on the other side of the cable, the signal return lead (the neutral conductor) in the connector on the control side is connected to the screen. The block diagram will then be as illustrated below.

NB7823

CNC3000   PERIPHERAL
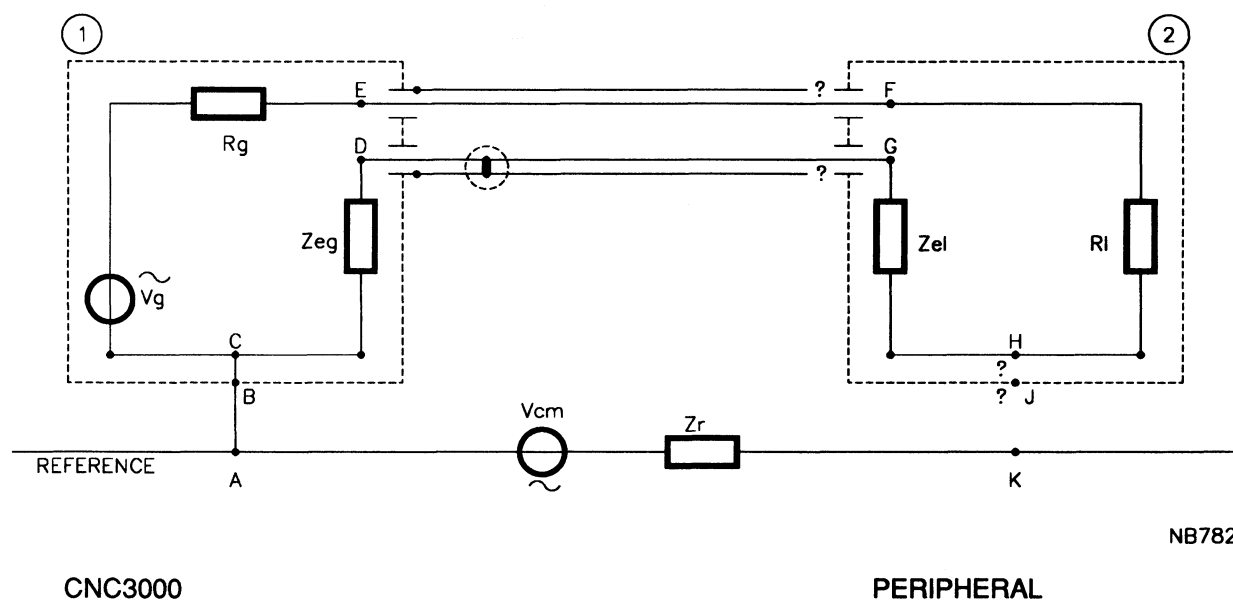
Fig. 3.1.3.-3.   Block diagram for the RS232/RS422 connection

Figure 3.1.3.-4 illustrates how the connection between screen and return lead in the CNC-side connector is made.

In the RS422-connector pin 5 is connected to the screen, in the RS232-connector pin 7.
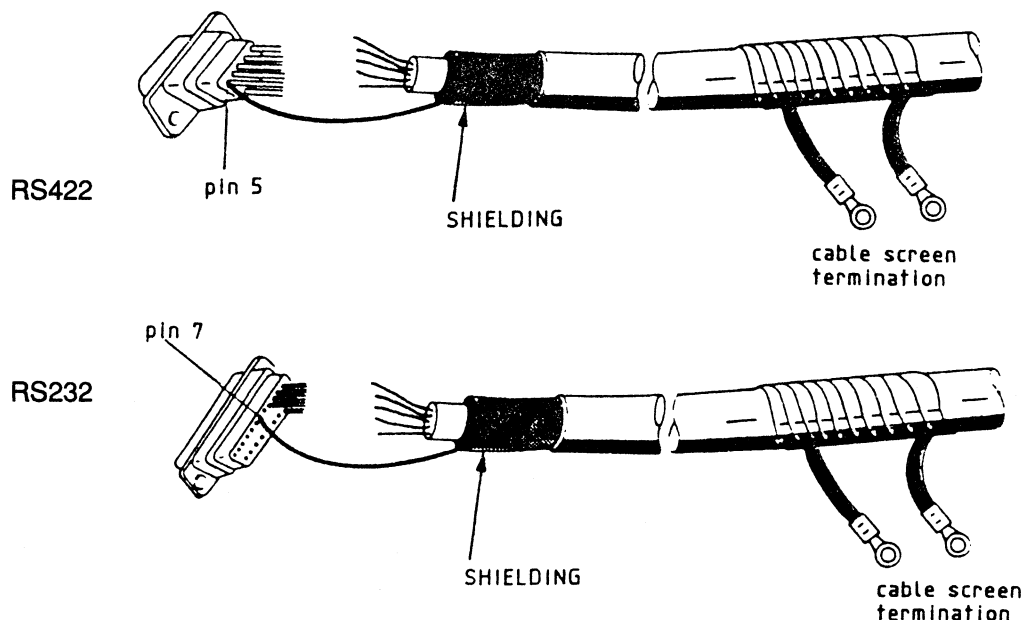
Fig. 3.1.3.-4.   Connection between screen and return lead

3.2.   EARTHING THE DIFFERENT RACKS

The rack must be connected to a reliable earth system via a 6mm^2 earth lead (minimum value) from the central earth terminal at the front of the rack.
Earth loops which can be formed by the screens of the connection cables between the rack and the machine tool should be avoided.

Central earth terminal
for 6 mm^2 earth lead
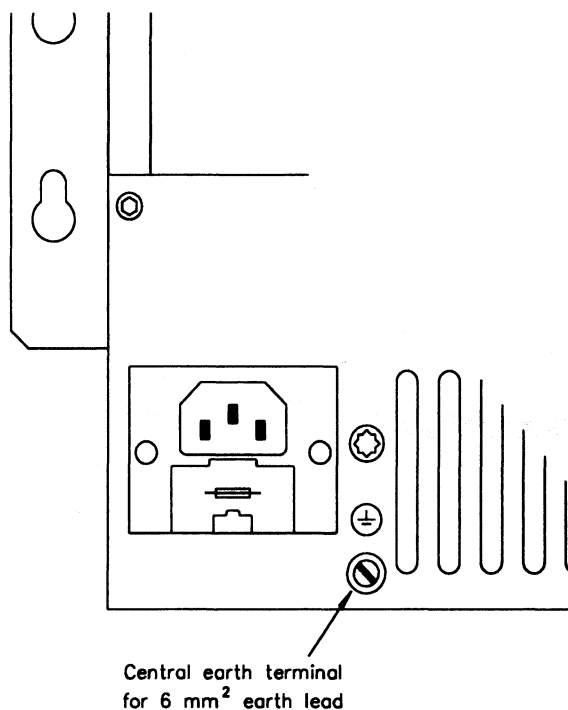
Fig. 3.2.-1.   Rack showing earth terminal.

WARNING

Before connecting the equipment to the mains of the building installation, the proper functioning of the protective earth lead of the building installation needs to be verified.

3.3. THE CONNECTION CABLES

3.3.1. Mounting the earth strip

For the 7 and 15 slots racks an extra earth strip can be mounted on the top side of the rack
For the 20 slots racks the earth strip can be mounted in three different ways:

- On the top side of the rack
- On the left side of the rack
- On the right side of the rack

So the 20 slots racks are always provided with two extra strips; a long strip for mounting on the top side, and a short strip for mounting on the left or right side (these strips are both standardly mounted on the top of the rack).
The mounting screws, distance pieces, earth rings and spring washers will be delivered in a separate plastic bag.

Mounting the strip on the top of the rack

First remove the upper strip from the lower strip and remount (if necessary) it to ensure sufficient grounding assembly with the distance pieces as shown in figure 3.3.1-1.

SPRING WASHER
EARTH RING
UPPER EARTH STRIP
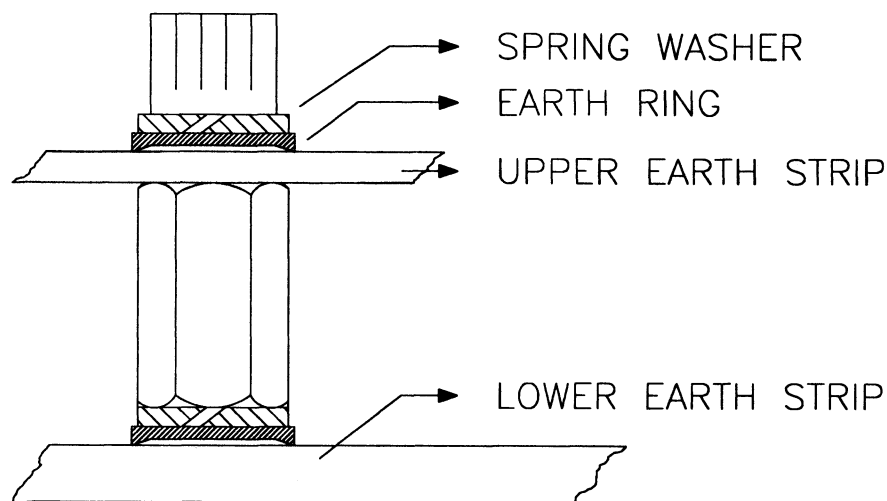
LOWER EARTH STRIP

NB7891

Fig. 3.3.1.-1   Mounting the extra earth strip

Mounting the strip on the left or right side of the rack

First remove both the earth strips from the lower strip on the top side of the rack.
The shortest strip must be mounted with the help of the two already existing screws and the two distance pieces on the right or the left side of the rack.
An example is given in figure 3.3.1.-2 for mounting the strip on the right side of the rack
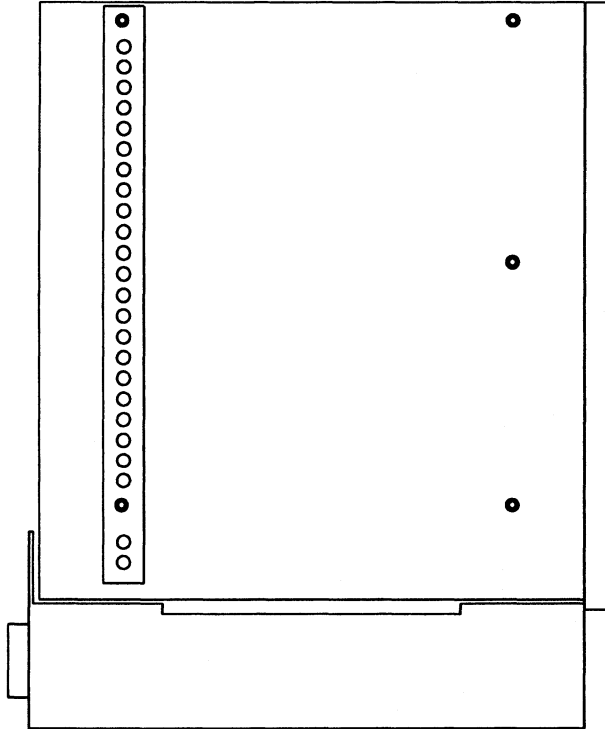
Fig. 3.3.1.-2 Mounting the extra earth strip

3.3.2.   Final assembly of the cables to the rack.

The cable clamps and the cable tags must **NEVER** be placed together under the same screw, otherwise sufficient grounding cannot be guaranteed.
Clamps of two different cables may join one screw.
Always use spring washers!

The overall length of 450 mm includes the plug, irrespective of its size.
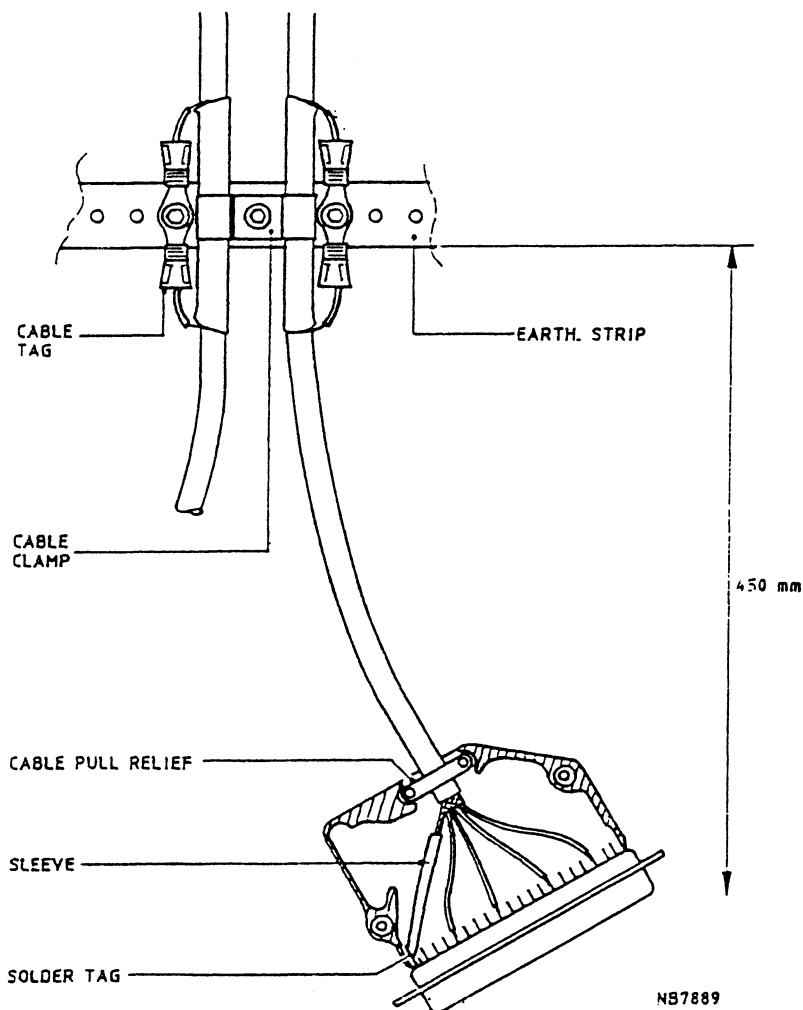
Fig. 3.3.2.-2   Connection cables mounted to the rack.

3.3.3. Cable screen termination
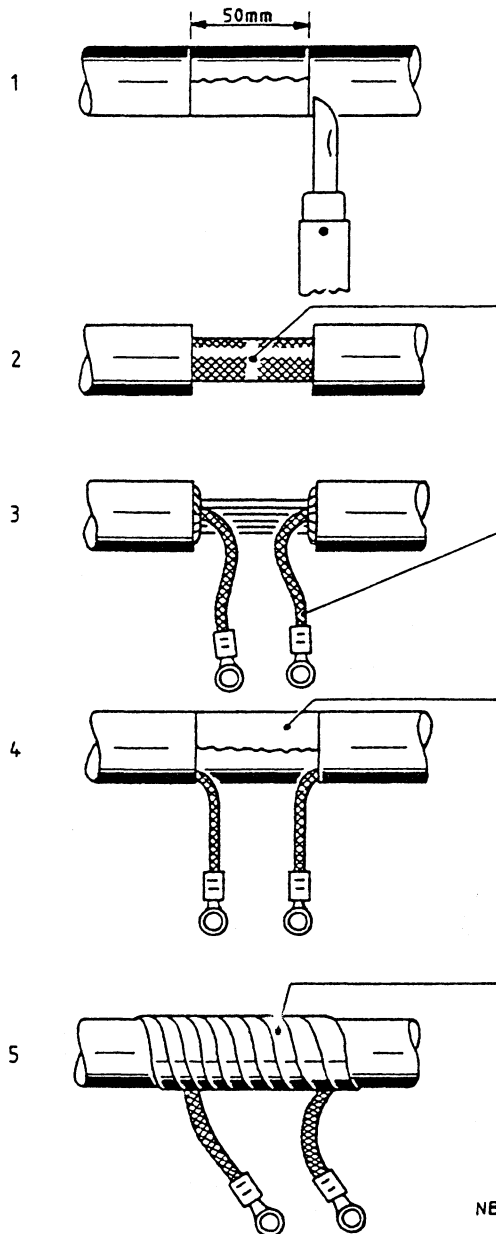
Carefully incise the outer sheath 450 mm away from the cable end and remove it over approximately 50mm.
Do not discard; it used again in phase 4.

Cut the screen lengthwise and across in the centre.
Be careful not to damage the wires; use a pair of scissors.

Unravel and twist the screen; solder a cable tag.

Replace the outer sheath, removed in phase 1.

Tape the cable.

NB7890

Fig. 3.3.3.-1   Cable screen termination.

3.3.4. Wiring the plug
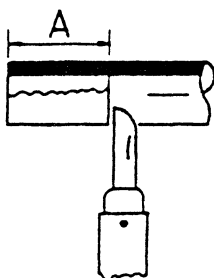
Carefully incise the cable sheath as indicated and remove it.

Cut the screen lengthwise and twist it.
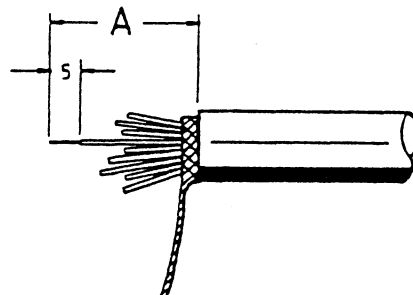
Strip and tin the wire ends as indicated.

Fig. 3.3.4.-1   Cable termination.

NB6711
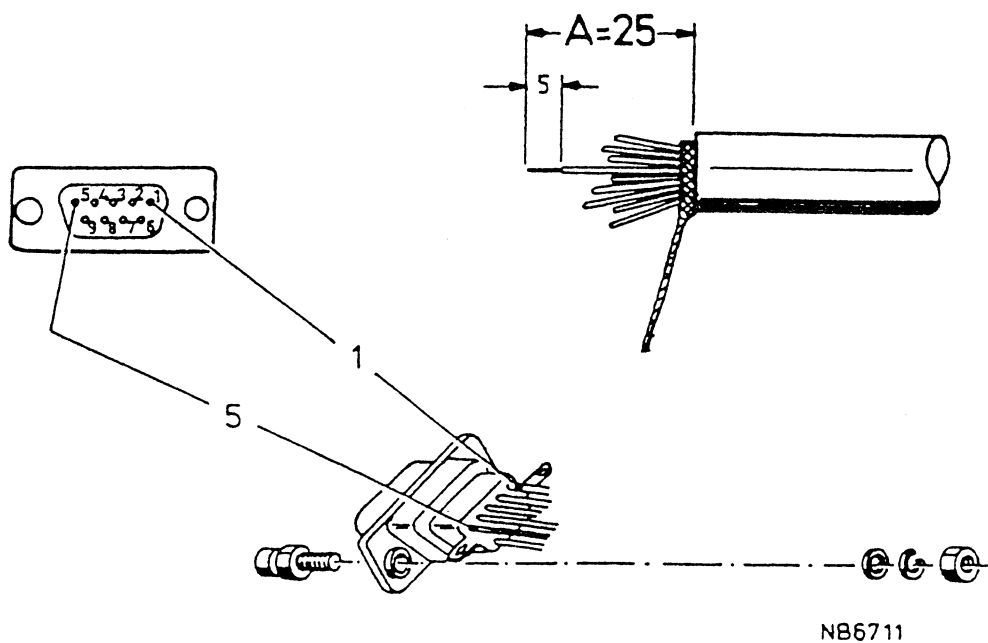
Fig. 3.3.4.-2   Wiring the 9-way plug

NB6712
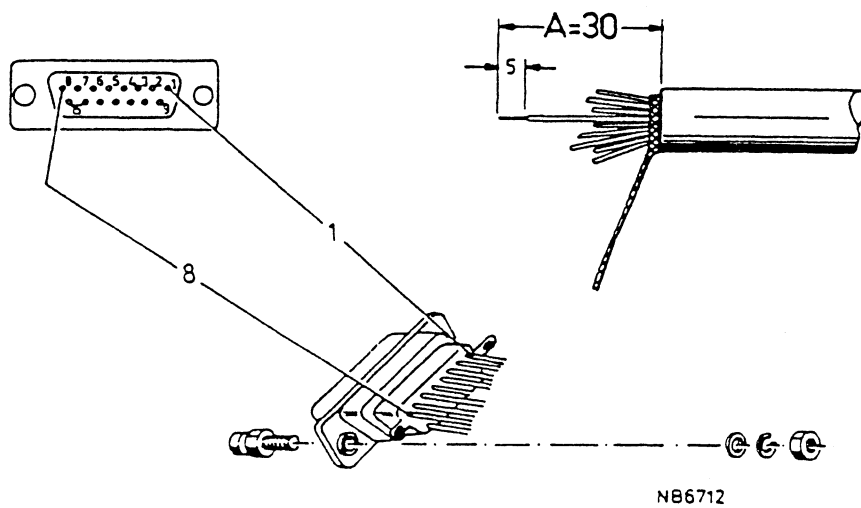
Fig.3.3.4.-3   **Wiring the 15-way plug**

NB6713
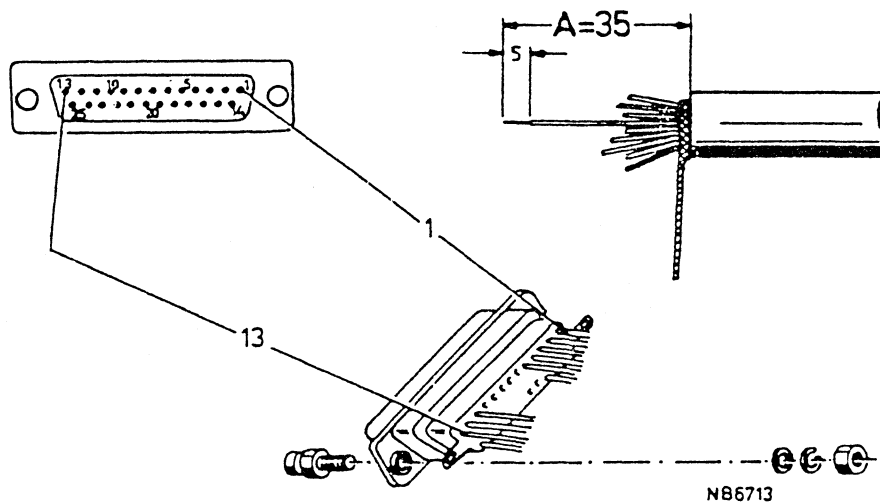
Fig. 3.3.4.-4   **Wiring the 25-way plug**
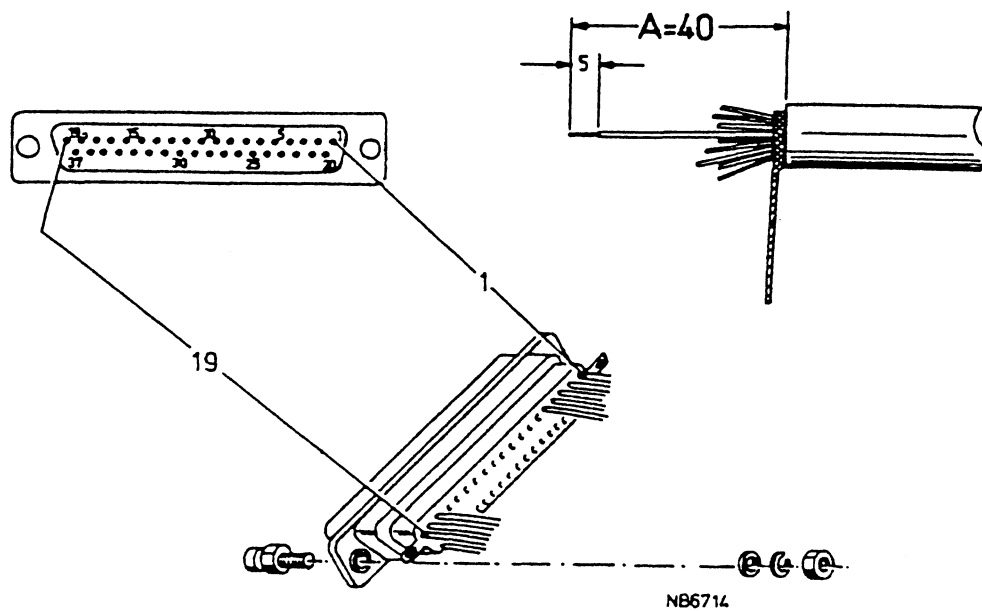
Fig. 3.3.4.-5   Wiring the 37-way plug

3.3.5. Locking and unlocking the connectors
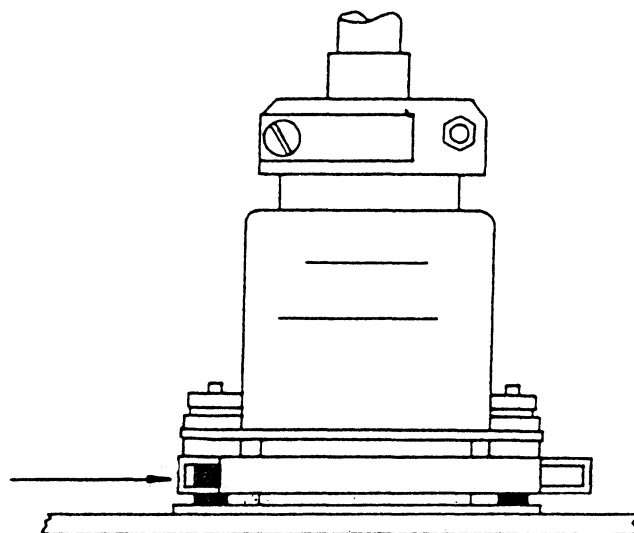
To remove the plug from the module socket,
first unlock by pressing in arrow direction.

Fig. 3.3.5.-1   Plug unlocked.

Insert plug into module socket and lock by
pressing in arrow direction.
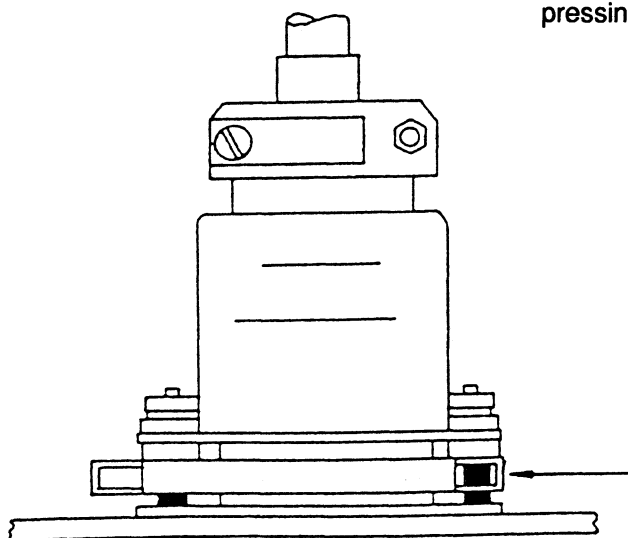
NOTE: To avoid interference during operation, make sure that all plugs on the modules are locked.

Fig. 3.3.5.-2   Plug locked.

Unlike other sockets, the control teletext socket X2 has been provided with a locking mechanism using two clamps and screws.
This is done to improve earth contact with the rack.
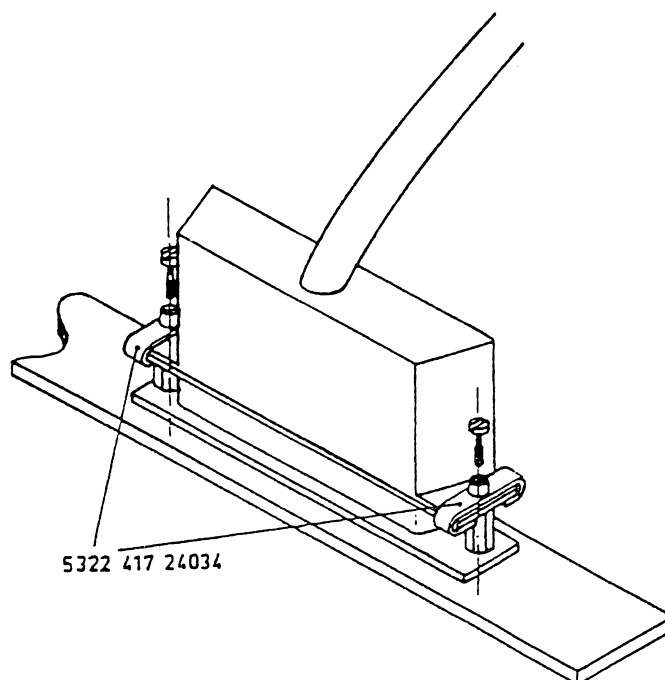
Fig. 3.3.5.-3   Locking mechanism for contr. telet. module socket X2.

4.   CONNECTING THE MAINS SUPPLIES

The units are set to 220V when leaving the factory.

4.1.   MAINS SUPPLY 220V.

4.1.1.   The rack.

The rack as well as the video unit (colour monitor) must be connected to an *interference free* mains supply of 220V 50-60 Hz +/- 1 Hz or 110V 50-60 Hz +/- 1 Hz.
The connection to a 110V mains supply requires some adjustments to be made. Details are given on the following pages.

Permissible voltage variations: - at 220V +10%, -15%
                                       - at 110V +10%, -15%

Power consumption: - rack   100VA
                                       - colour monitor 80VA
                                       - b/w monitor   15VA

The required connection cables must meet $3 \times 2.5mm^2$.
The appropriate plug is delivered with the unit.

Socket for
mains supply

NB7900
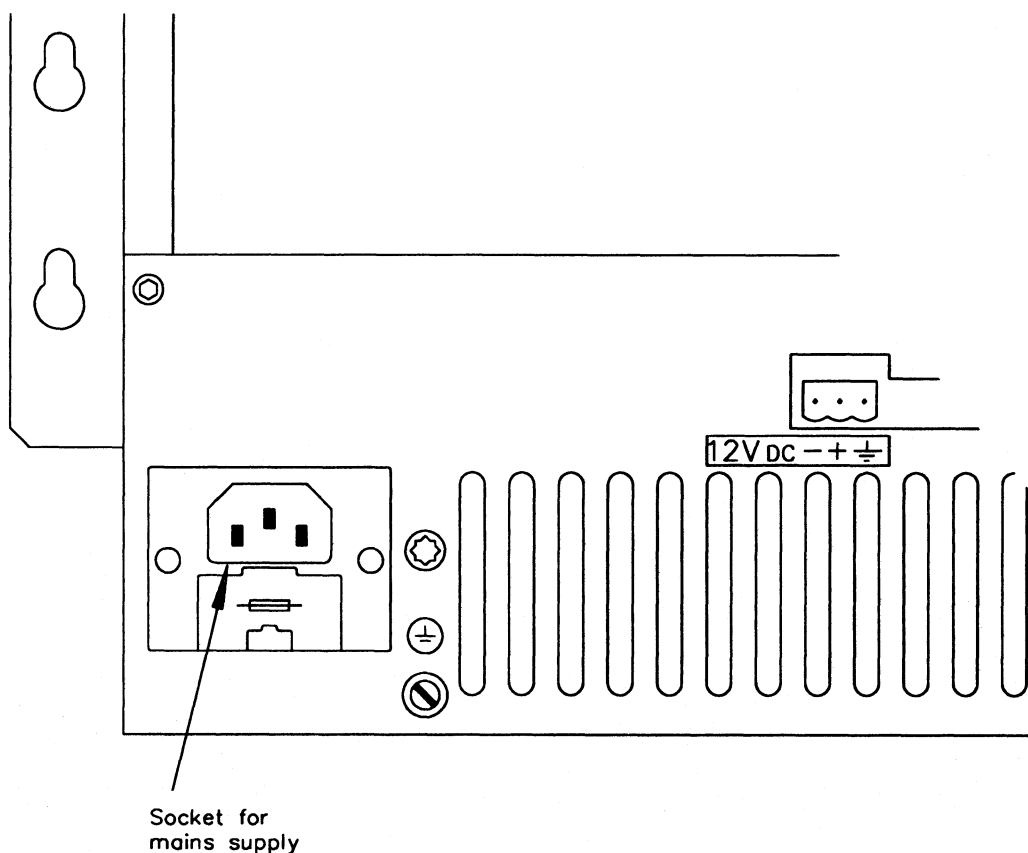
Fig. 4.1.1.-1   Detail of control showing socket for mains supply (7, 15 and 20 slots)

4.1.2.   The colour monitor

Socket
for mains supply

NB7625
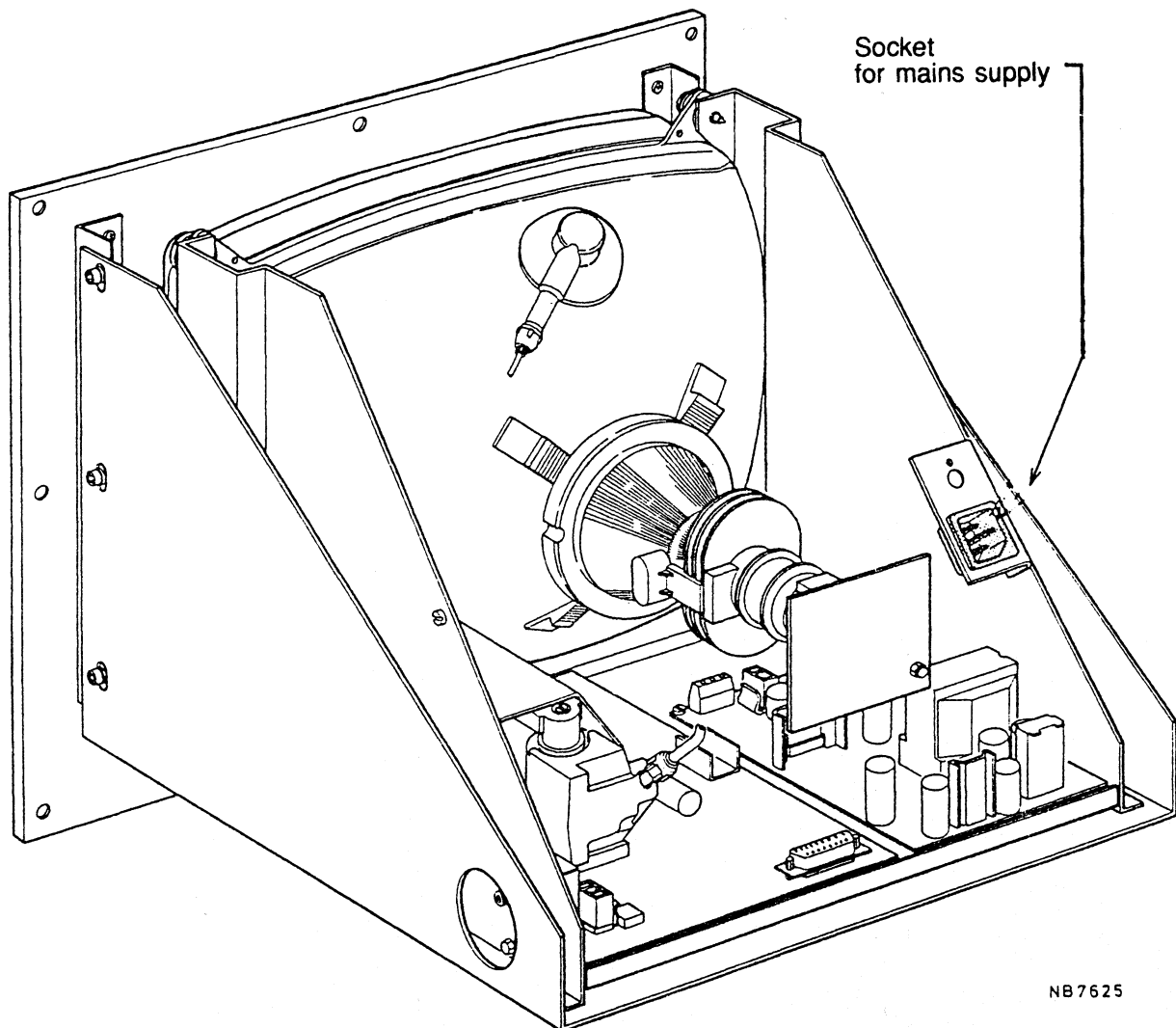
Fig 4.1.2.-1   **Rear view of 14' colour monitor showing socket for mains supply.**

4.2.   MAINS SUPPLY 110V

**MAKE SURE THAT THE UNIT IS DISCONNECTED FROM THE MAINS SUPPLY BEFORE ADJUST-
MENTS ARE MADE.**

4.2.1.   7 slots rack

The adjusment must be made on the power supply board.
To gain acccess to this board remove the two screws marked A in figure 4.2.1.-1. and slide the power supply compartment forwards over 10 cm and remove the plug indicated B in figure 4.2.1.-2. Now slide the compartment further.

Remark: The slide guides are not present over the full travel. When the unit is halfway it is locked.
Lift the unit and slide it further forwards to remove the unit from the rack.
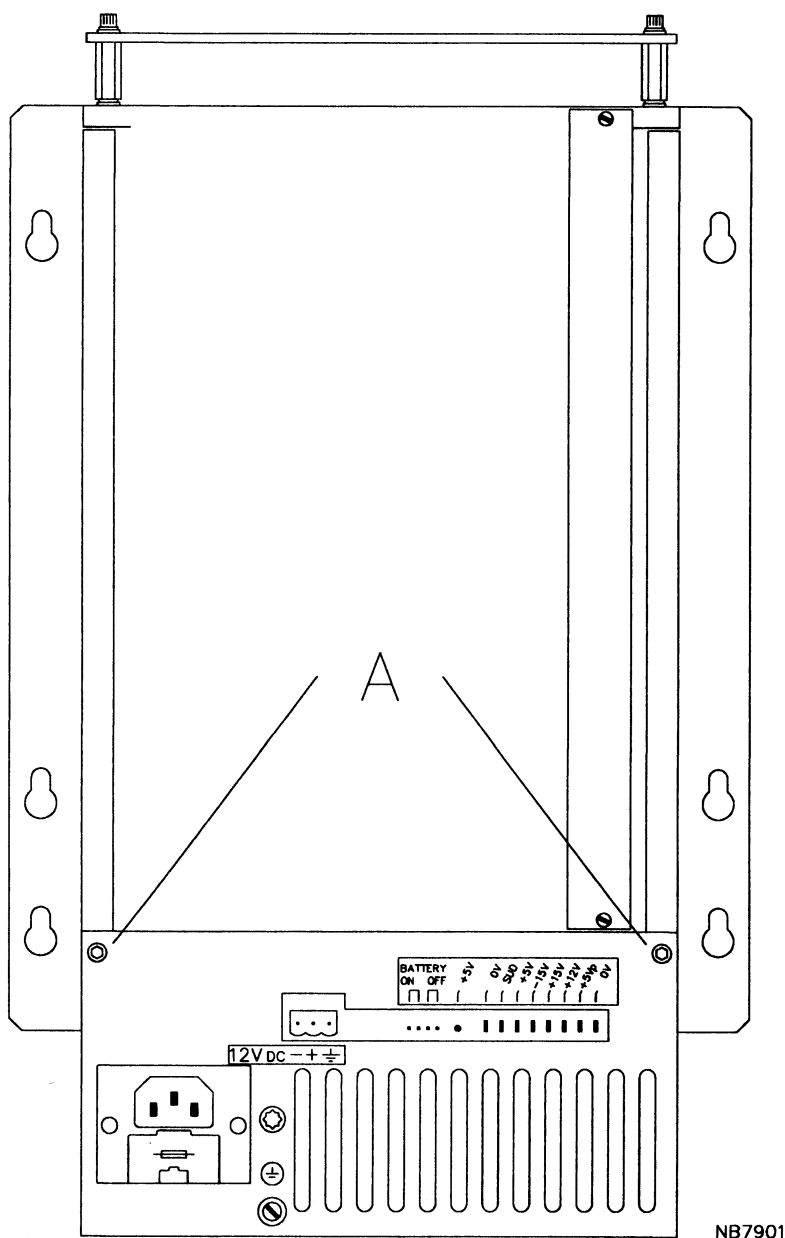
Fig. 4.2.1-1   Removing the power supply compartment.

To gain access to the power supply board:

- Remove the screws marked A in figure 4.2.1.-2.
- Remove the two plugs from the socket marked A.
- Lift the plate upwards by holding the fan.
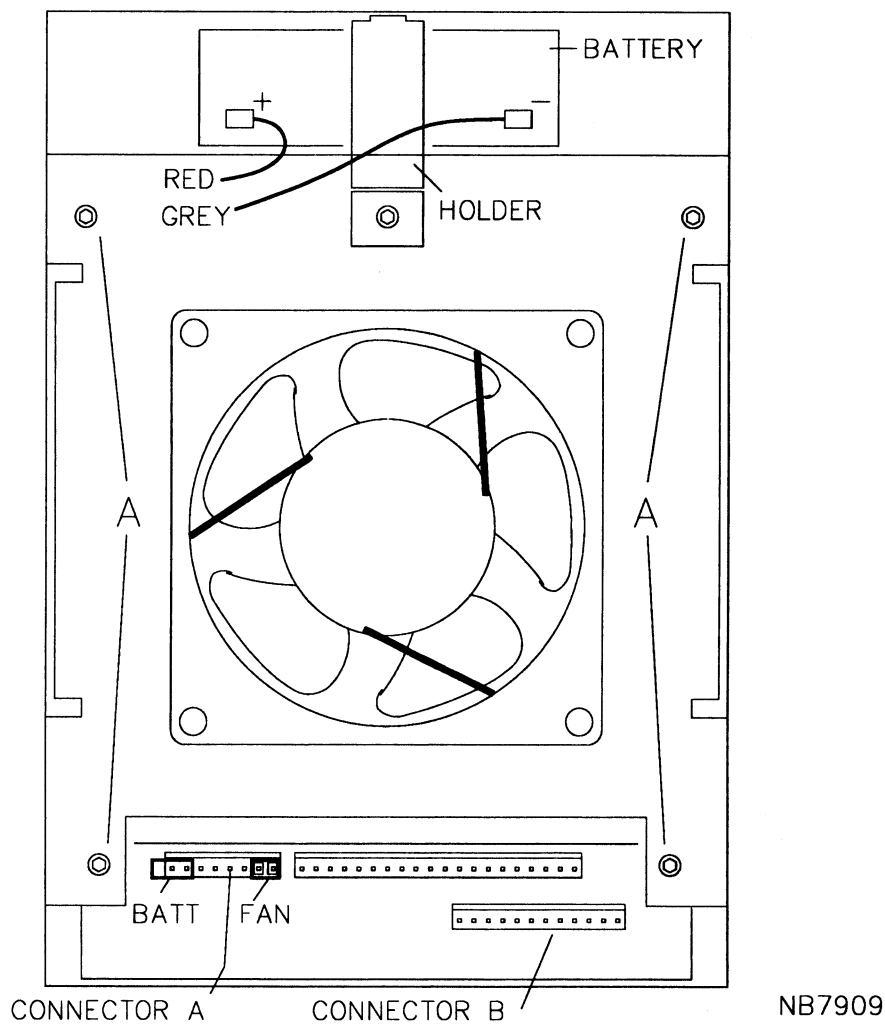
Fig. 4.2.1.-2. **Gaining access to the power supply board.**

The adjustment is made by placing the jumper in position 110V as indicated in figure 4.2.1.-3.

NB 7903
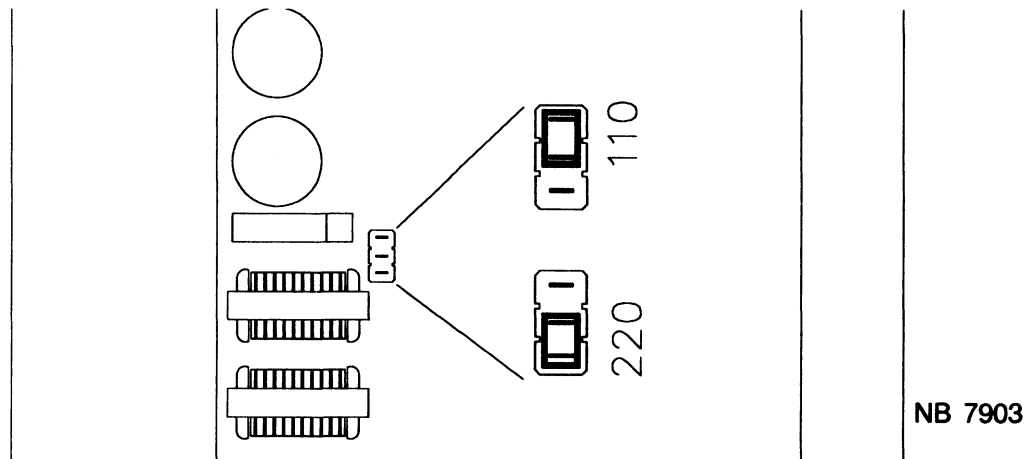
Fig. 4.2.1.-3.   Adjustment to the mains voltage.

To assemble the power supply compartment, place the fan plate matching the four nuts. Take care of the wires going to the small PC-board.
Replace the plugs as indicated in figure 4.2.1-2, the wires from the battery come to the left (mark the key slot) and the wires from the fan come to the utmost right of socket B.

Make sure that the new adjusted voltage is marked on the rack by placing the 110V sticker under the mains inlet.
The sticker is delivered with the control.

4.2.2.   110V mains supply for 15 and 20 slots rack

MAKE SURE THAT THE UNIT IS DISCONNECTED FROM THE MAINS SUPPLY BEFORE ADJUSTMENTS ARE MADE.

The adjustment is done on the power supply board. It is accesible via the opening in the left side of the rack, see figure 4.2.2.-1.

Place the plug in the right position for 110V with the use of a set of pliers as shown below.

CONNECTOR

220V   110V

NB7904
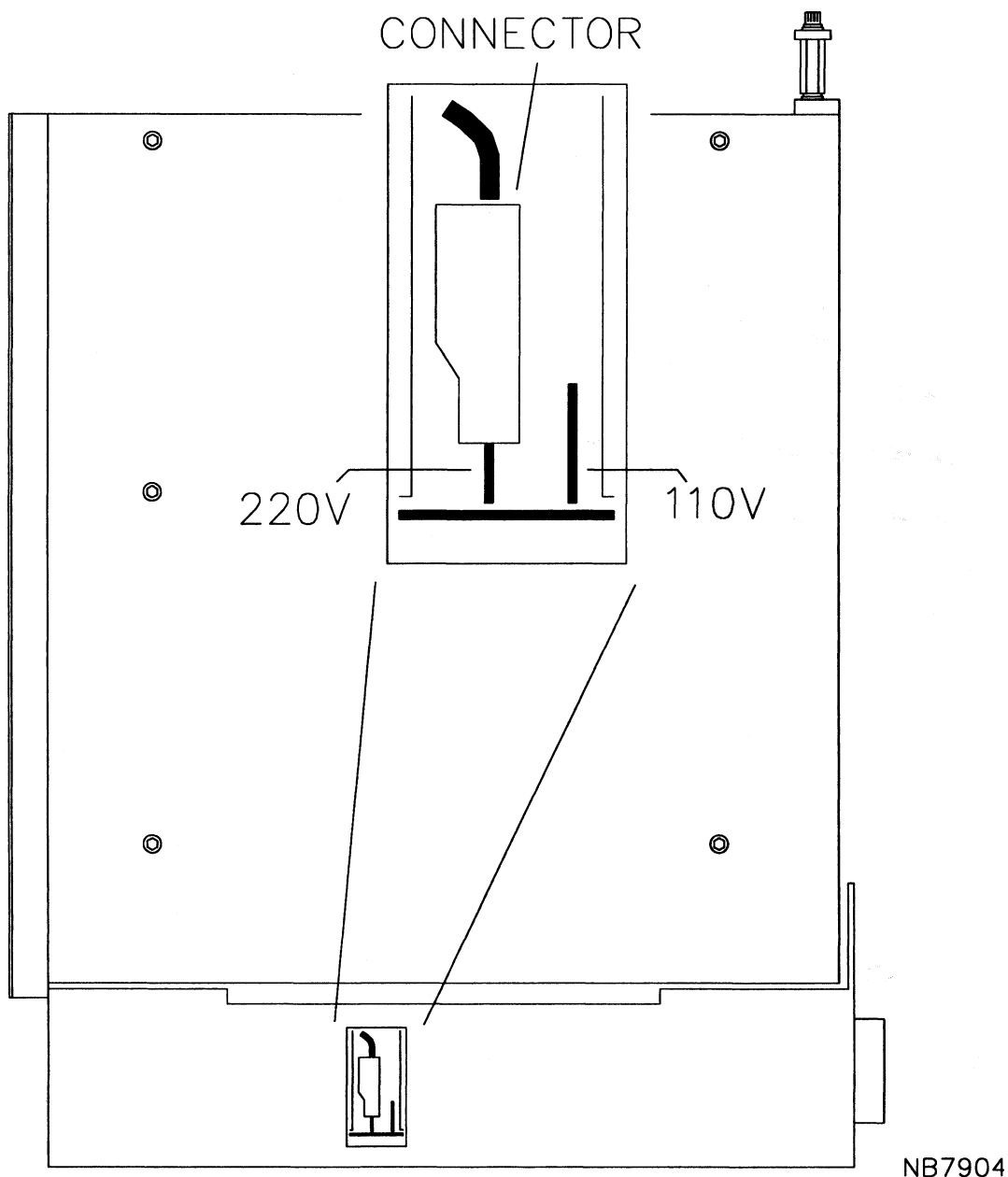
Fig. 4.2.2.-1   Setting the power supply to the mains voltage

Make sure that the new adjusted voltage is marked on the rack by placing the 110V sticker under the mains inlet.
The sticker is delivered with the control.

4.2.3. 110V mains supply for the colour monitor
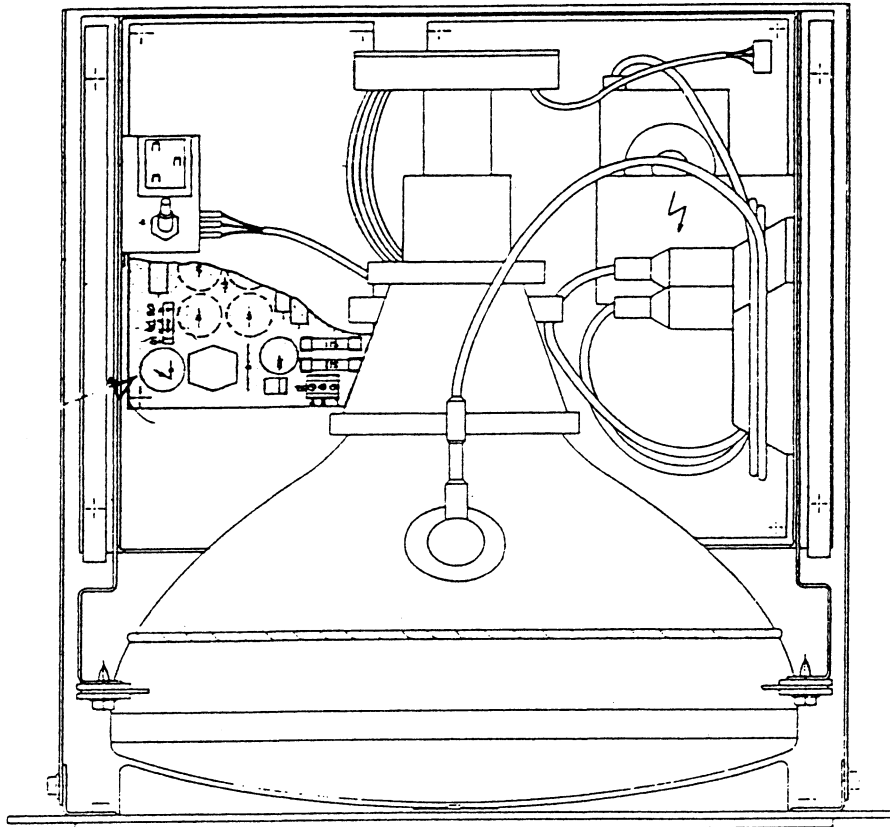
Fig 4.2.3.-1. Top view of 14" colour monitor.
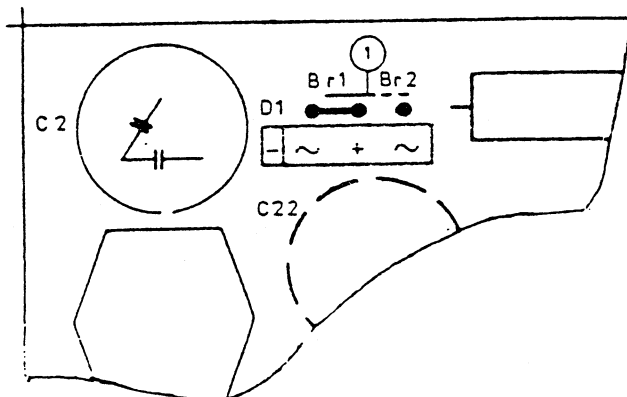
Fig. 4.2.3.-2.
Wire link for 220V mains supply
(Wire link Br 1)
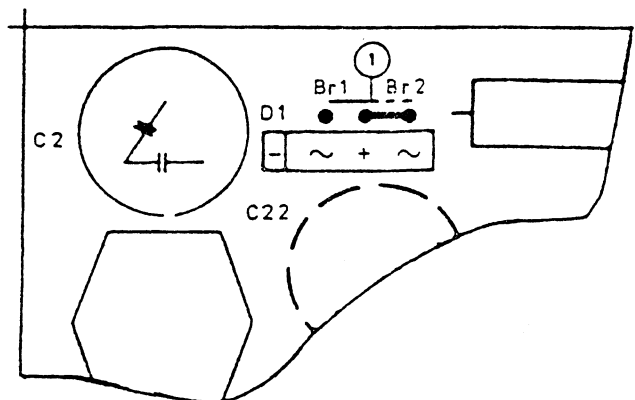
Fig. 4.2.3.-3.
Wire link for 110V mains supply
(Wire link Br 2)

4.3. POWER SUPPLY FOR THE MONOCHROME MONITOR

4.3.1. Distance up to 1 meter

Connection cable to socket VIDEO
on CONTR TELET MOD, including
conductor for 12V dc supply
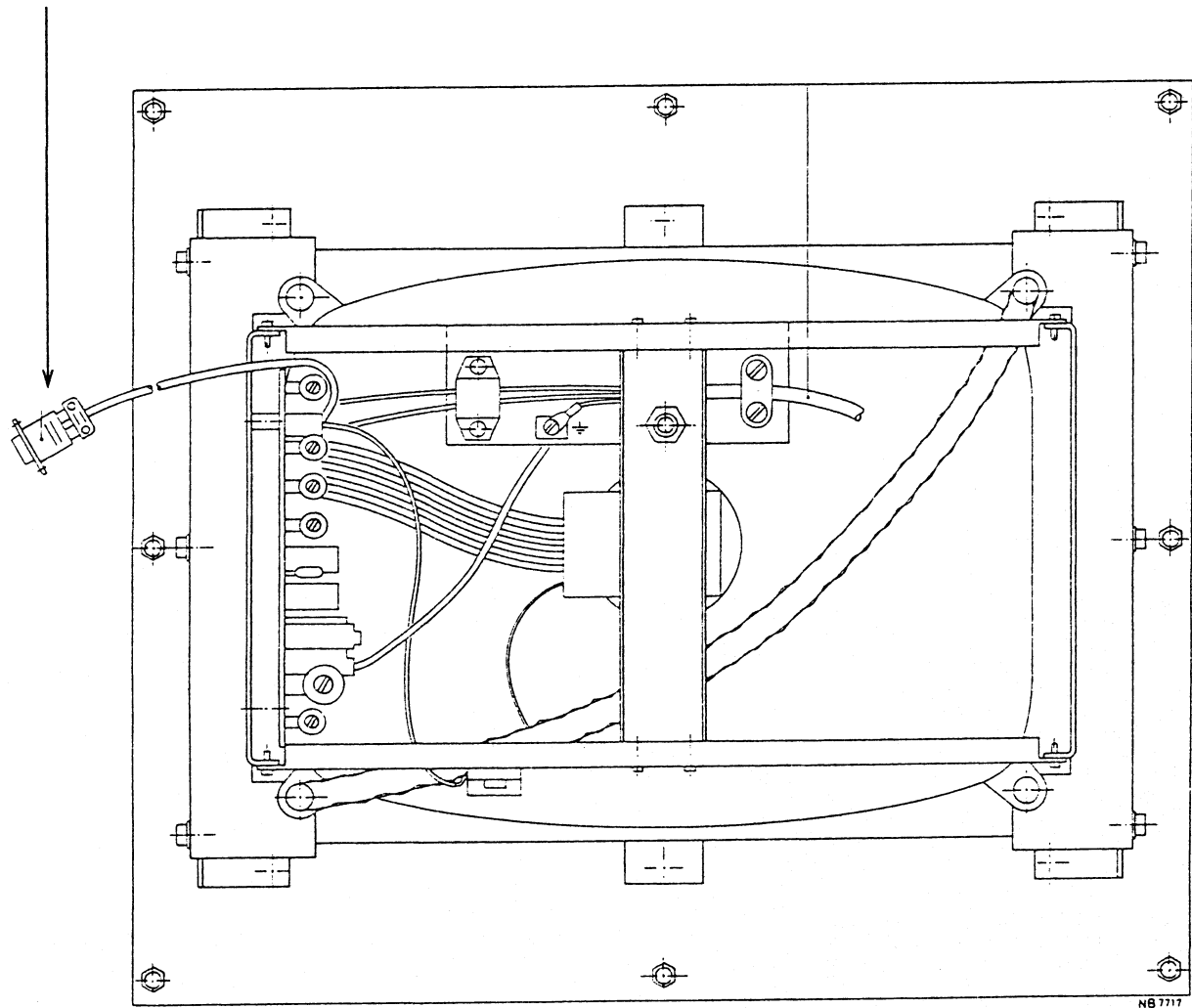
Fig. 4.3.1.-1.   Rear view of 12" monochrome monitor.

The 12V dc supply for the monochrome monitor is fed via the normal connection cable with the 9-way plug to the control unit.

This option is used when the cable between the monochrome monitor and the rack is not longer than 1 meter.

4.3.2. Distances from 1 to 10 meters

Connection cable to socket VIDEO
on CONTR TELET MOD, see section 5.1.2

Connection of separate
3-core 2.5 mm2 cable
for 12V dc supply.

NB 7507
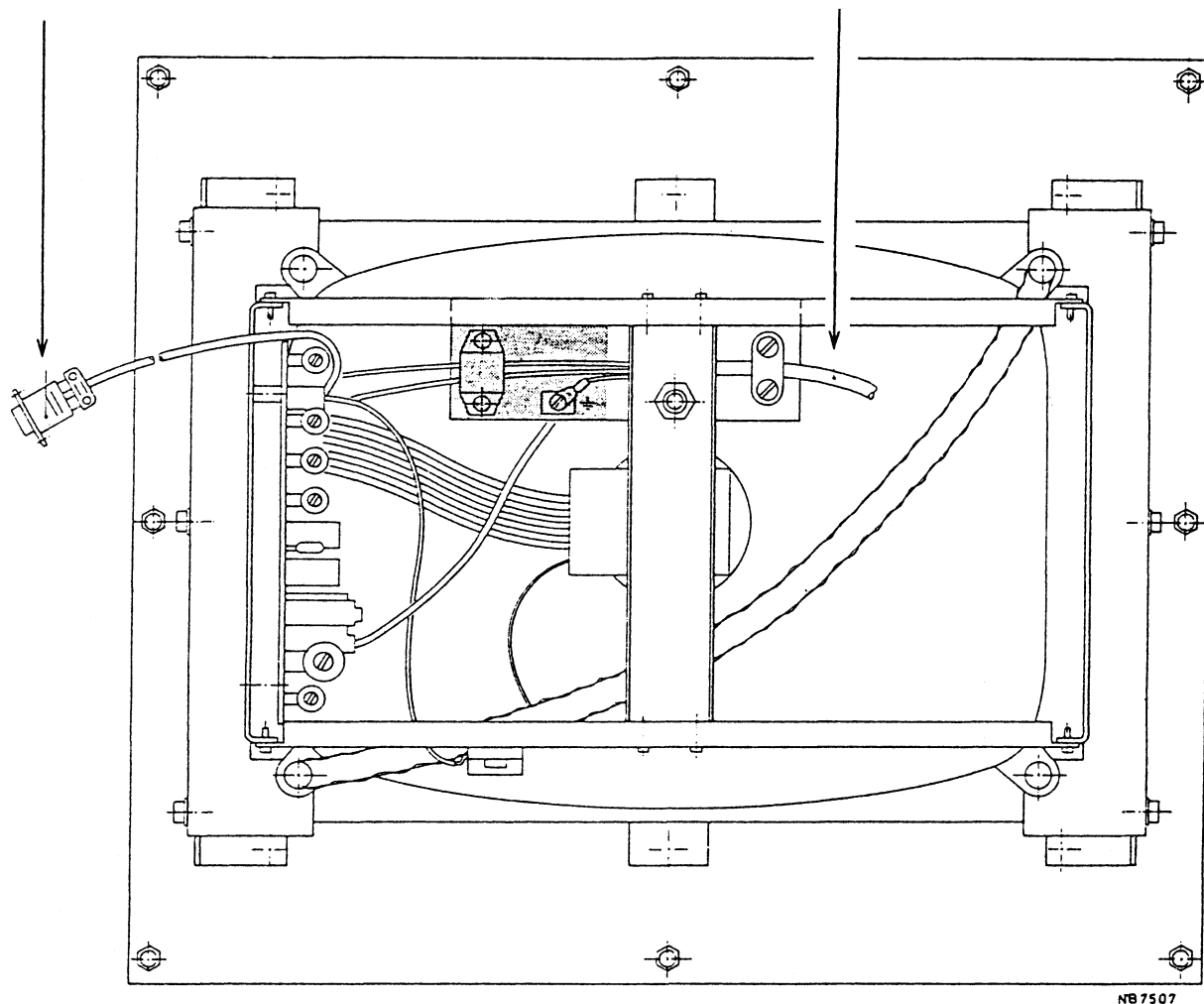
Fig. 4.3.2.-1   **Rear view of 12" monochrome monitor showing power connection.**

GREY

BLACK

BLUE

BROWN

0̸V

+12V

NB 7389

EARTH TERMINAL

Remove blue and brown wires to avoid a supply loop.
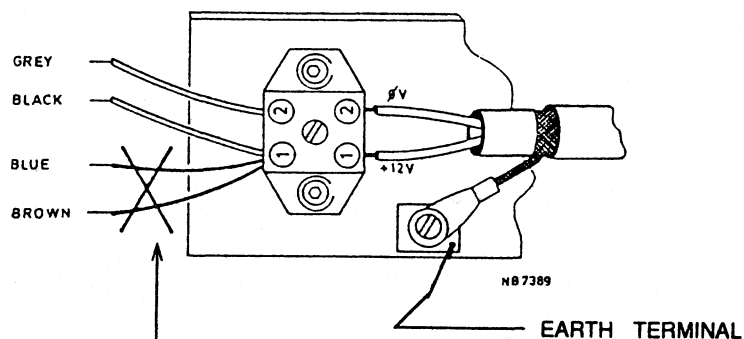
Fig. 4.3.2.-2   **Detailed rear view showing power connection.**

The 12V dc supply for the monochrome monitor is fed via a separate shielded 3-core cable (2.5 mm^2) of suitable length (max. 10m) to the connection terminal at the front of the power supply unit, see figure 4.3.2.-4.

This option is only to be used when the connection cable between monitor and rack is between 1 and 10 meters.

Wire the plug as described in figure 4.3.2.-3.

The plug is delivered with the unit.

NB7906
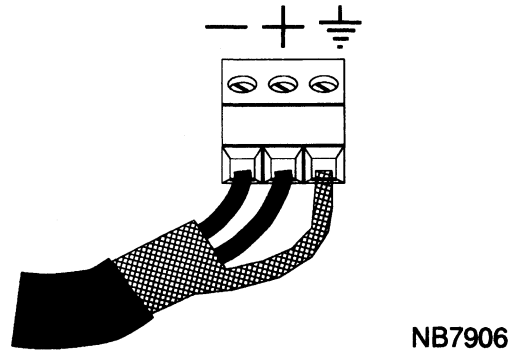
Fig. 4.3.2.-3   **Connection of the 12V dc power supply for the monochrome monitor on the rack side.**

CABLE PULL RELIEF
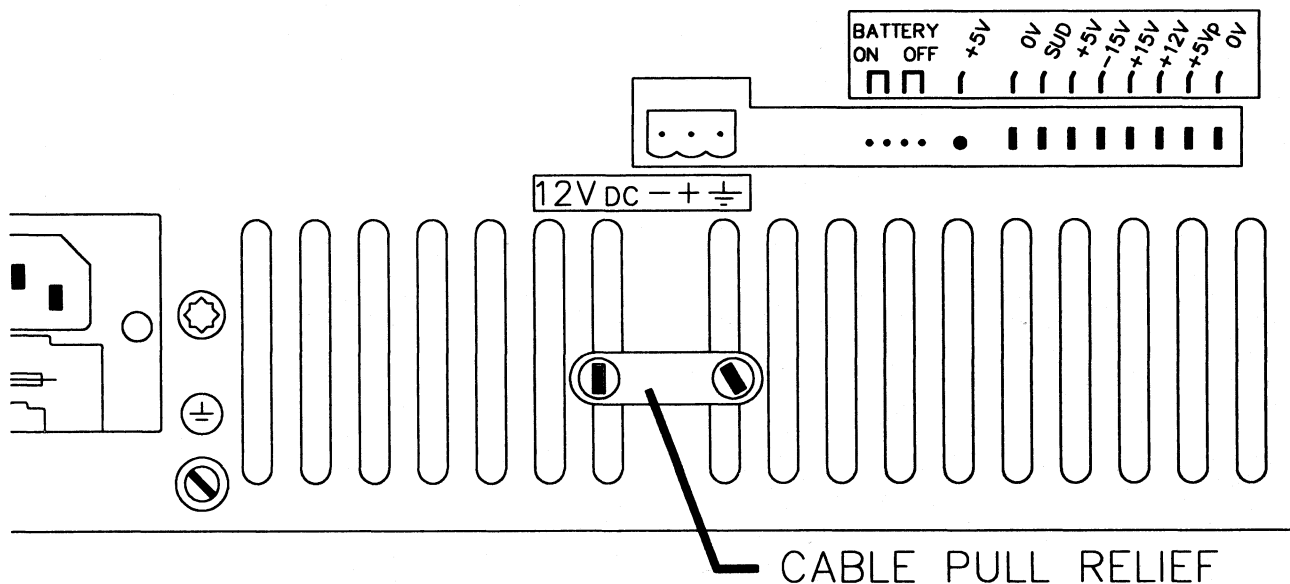
Fig. 4.3.2.-4   **Location of the 12V dc power supply socket for the monochrome monitor.**

Make sure to use the cable relief.

4.4. BACK UP POWER SUPPLY

The CNC is equipped with a back up power supply (in the form of a battery) as a standard provision. It prevents programs from getting lost during power-off for a duration of 1000 hours at most.

In order to be sure that a new battery is charged properly it is neccessary to have the control switched on for 24 hours at least after first commissioning or after replacing the battery.

The battery is charged automatically by the power supply module. The CNC may be switched on contineously, there is no danger of overcharging the battery.

Switching from mains supply operation to back up supply operation is effected automatically via the power supply module.

Precautions when using lead accumulators with gastight cells

Gastight cells should never be exposed to fire or opened by force.

They may explode or release toxic material.

Do not short circuit the cells; they may be overheated.

Gastight cells should not be permanently connected to the circuitry.

Though the cells are provided with safety valves to avoid explosion, the reliability of the valves can be unpredictably affected when exposed to fire.

Replace the battery four years after the production date indicated on the battery.

The cells contain Potassium Hydroxide (KOH) as an electrolyte.

4.4.1 Switching on (7,15 and 20 slots rack)

NB7907
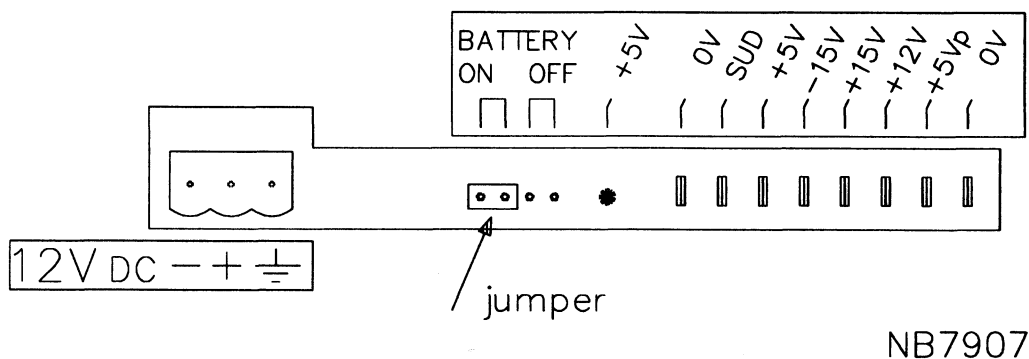
Fig. 4.4.1.-1   Switching on the back up power supply.

The back up power supply is put into use by setting the jumper in the postion battery "ON", as illustrated in the figure above.

4.4.2. Exchanging the battery

Exchanging the battery of the 7 slots rack

The back up battery is located in the power supply compartment.
Removing this compartment from the 7 slots rack is described in section 4.2.1, but do not remove the fan plate, however remove the battery holding strip.

Pull the grey and red wires from the battery, take the battery out and put a new one in.
Connect the wires again, making sure that the grey wire is connected to the -pole and the red one to the +pole. Mount the battery holder again.
Mount the control panel again, plugging in the connector at the back and securing the board to the control unit with the same screws.
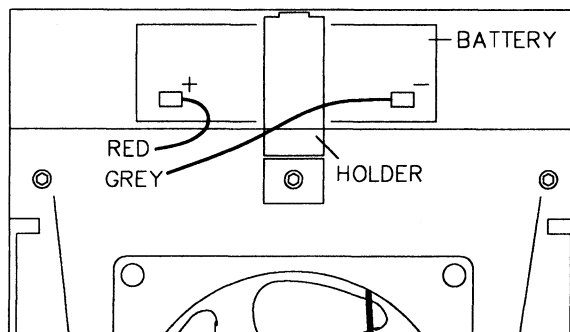
Fig. 4.4.2.-1 Location of the battery of the 7 slots rack

Exchanging the battery of the 15 and 20 slots rack

Removing the power supply compartment from the 15 slots rack is done by removing the screws marked A in figure 4.4.2.-2 and sliding the unit forward over 10 cm.

Removing the power supply compartment from the 20 slots rack is done by removing the screws marked A in figure 4.4.2.-3 and sliding the unit forward over 10 cm.
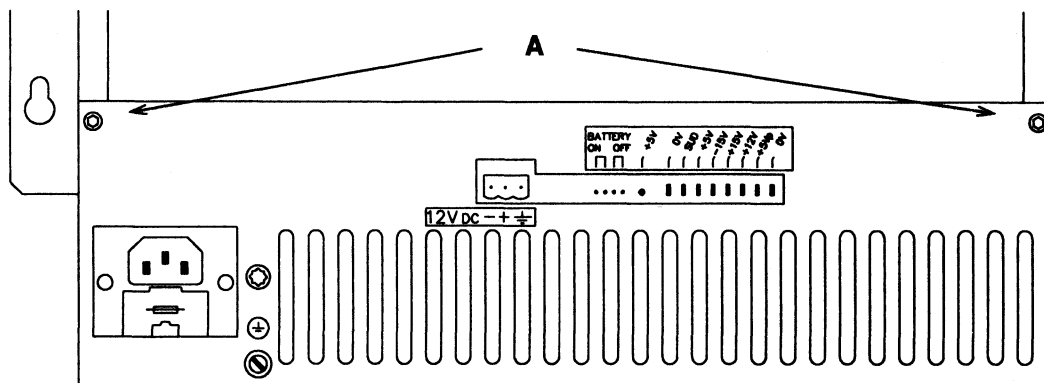
Fig 4.4.2.-2   Removing the power supply compartment from the 15 slots rack.
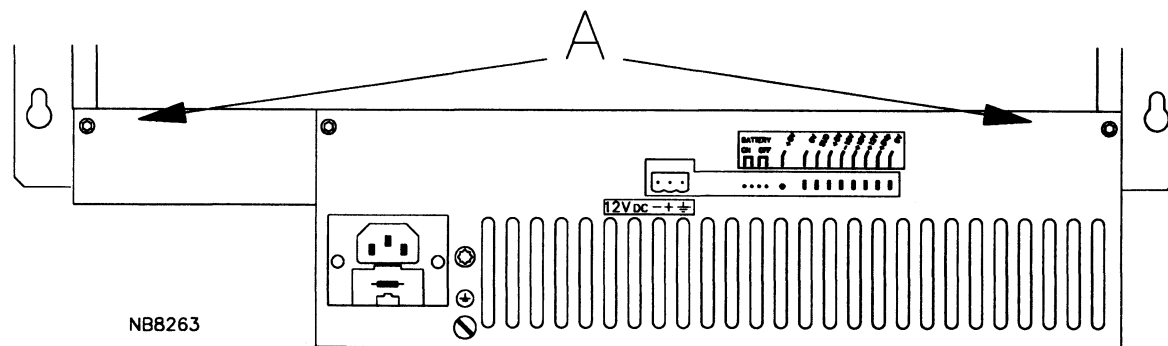
Fig 4.4.2.-2   Removing the power supply compartment from the 20 slots rack.

Remove the plug from connector B and the two thick wires from connector C.
Slide the unit further forwards.

Remark: The slide guides are not present over the full travel, when the unit is halfway it is locked.
            Lift the unit and slide it further forwards to remove the unit from the rack.
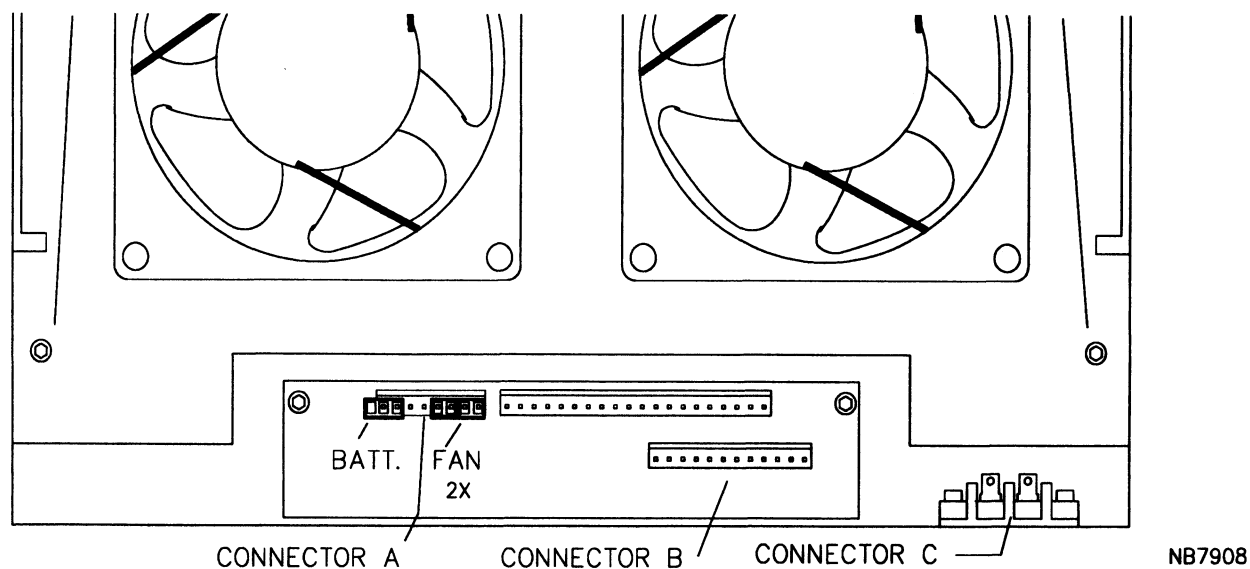
Fig 4.4.2.-3.   Location of the connectors B and C.

Pull the grey and red wires from the battery, take the battery out and put a new one in.
Connect the wires again, making sure that the grey wire is connected to the -pole and the red one to the +pole. Mount the battery holder again.

Mount the control panel again, plugging in the connector at the back and securing the board to the control unit with the same screws.
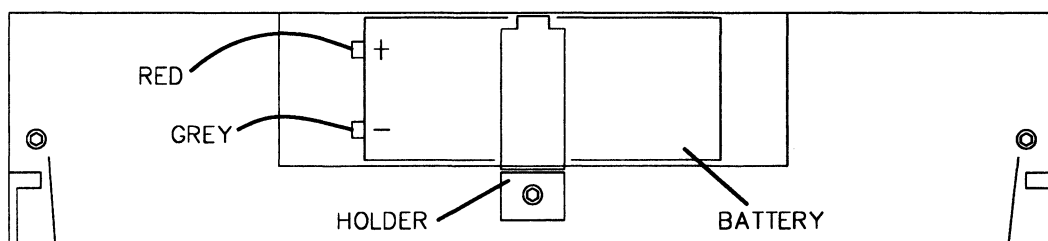
Fig. 4.4.2.-4   **Location of the battery of the 15 and 20 slots rack**

Remark: *Pay extra attention replacing the two wires on connector C. They must match the colours of the wires at the bottom of the connector. If connected wrong, the entire control is defective beyond repair!*

Some CNC's are delivered with a blue wire instead of a red wire for the plus connection.

4.5. LOCATION OF THE FUSES

4.5.1. Location of the fuses of the 7, 15 and 20 slots rack.

NB7902
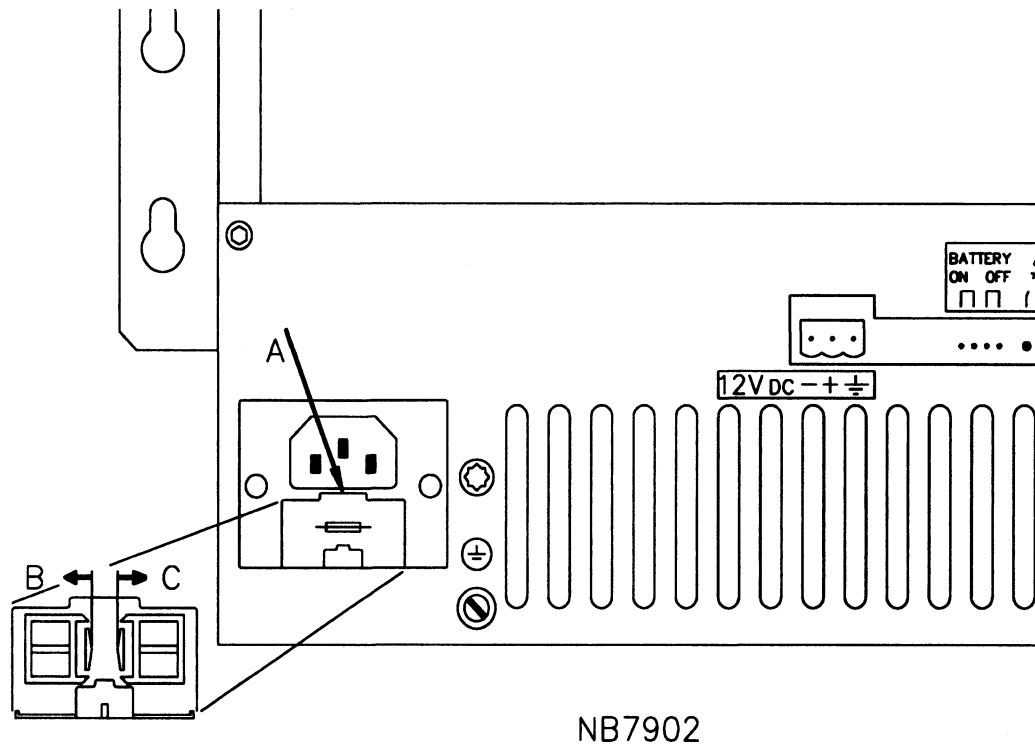
Fig. 4.5.1.-1   Location of the mains fuses.

To replace the fuse, remove the mains plug from the socket.
- Insert a screwdriver in slot A (figure above) and pull cap to the front.
- Push the lips B and/or C in the direction of the arrows (figure above) and pull the fuse holder out of the socket.

The fuses are for the 7 slots rack slow acting 250V/2A, and for the 15 slots rack slow acting 250V/3, 15A. and for the 20 slots slow acting 250V/3,15A.

4.5.2. Location of the power supply module in the 7 slots rack

Gaining access to the power supply board is described in section 4.2.1.

NB7903
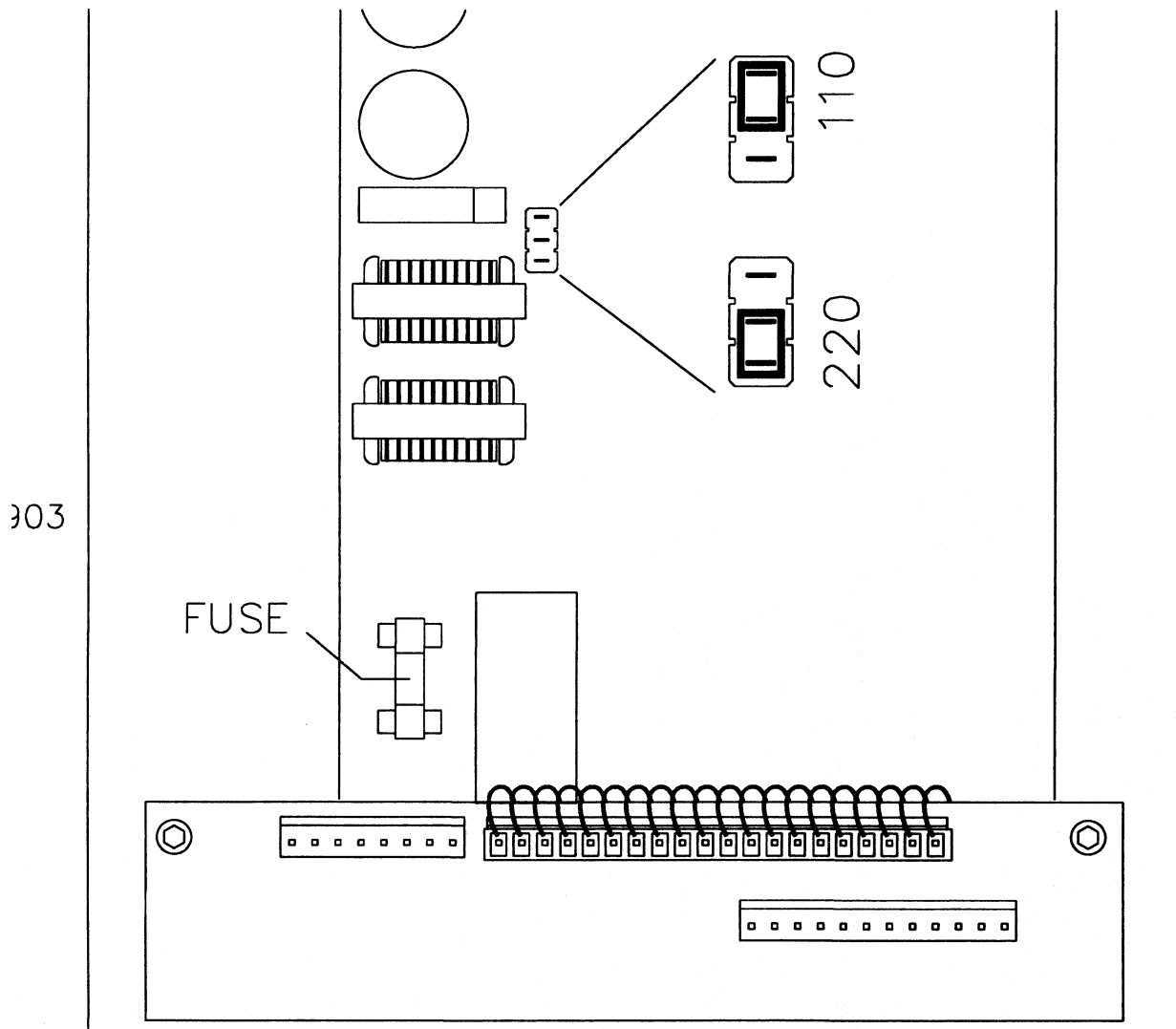
Fig. 4.5.2.-1   Location of the power supply fuse, 7 slots rack.

The power supply module of the 7 slots rack is protected by means of a fuse slow acting 250V/2.5A.

4.5.3. Location of the power supply module in the 15 slots rack

See section 4.4.2 for removing the power supply compartment.
Remove four screws marked A in figure 4.5.3.-1 and remove all plugs from connector A. The wires from the battery are protected against faulty positioning by means of a key slot.
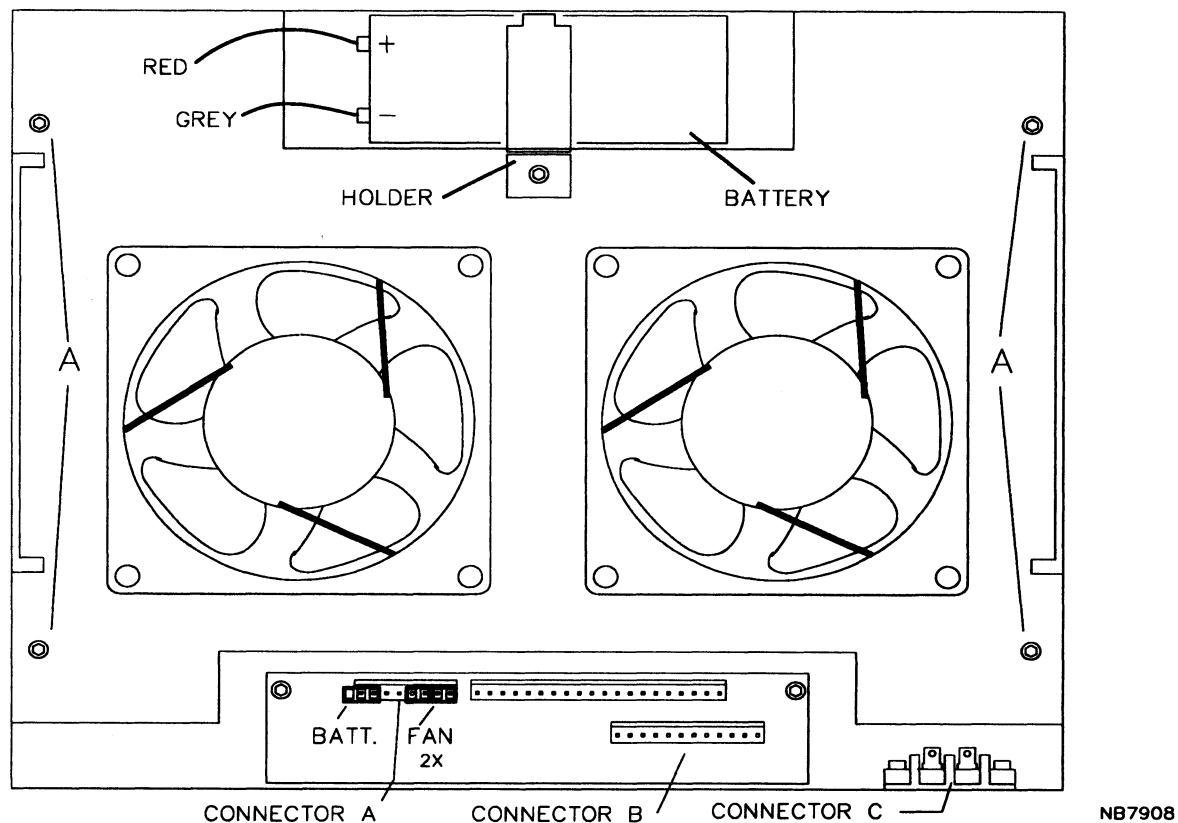
Fig. 4.5.3.-1   **Gaining access to the power supply board.**

The power supply module of the 15 slots rack is protected by means of a fuse (6.35 x 32 mm) slow acting 250V/4A.
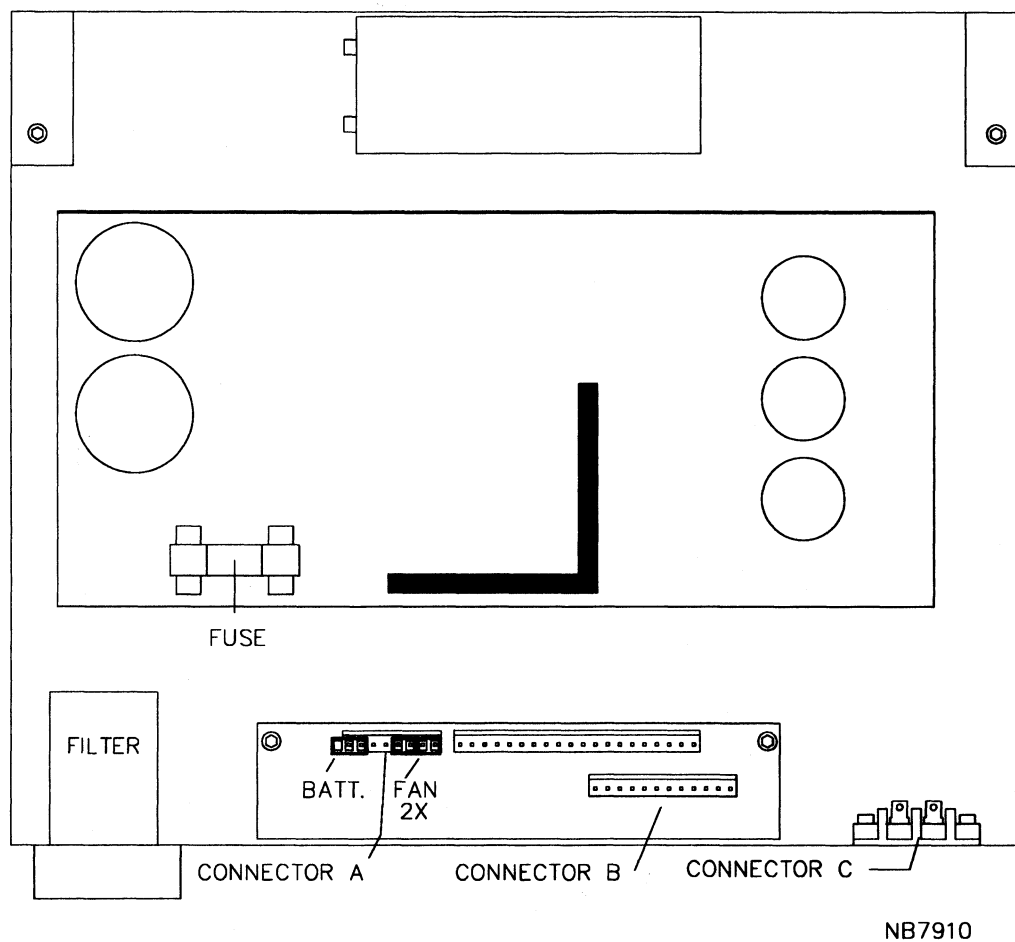
Fig 4.5.3.-2.   **Location of the power supply fuse, 15 slots rack.**

4.5.4. Location of the fuses of the power supply of the 20 slots rack

See section 4.4.2. for removing the power supply compartment.
Remove the four screws marked A in figure 4.5.4.-1 and remove all plugs from connector A. The wires from the battery are protected against faulty positioning by means of a key slot.
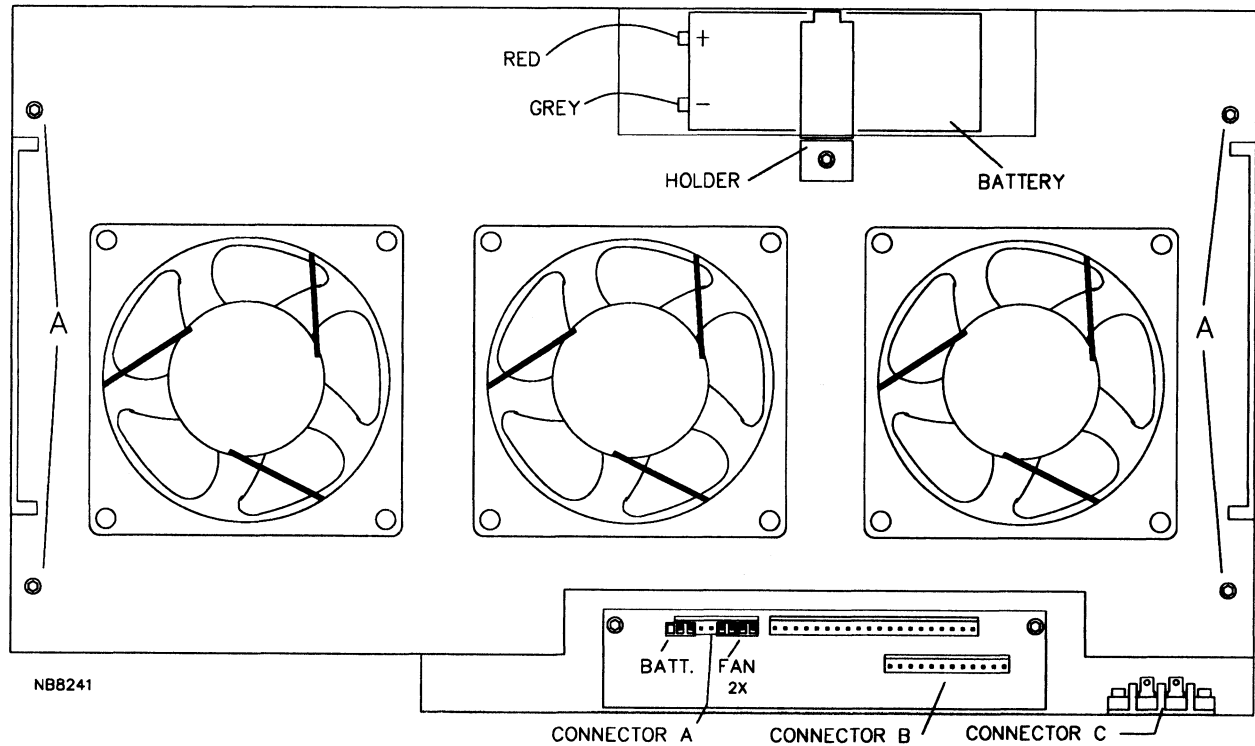
Fig. 4.5.4.-1   Gaining access to the power supply board

The power supply module of the 20 slots rack is protected by means of a fuse (6.26 x 32 mm) slow acting 250V/4A.

FUSE

FILTER

BATT. FAN 2X

CONNECTOR A   CONNECTOR B   CONNECTOR C

NB7910
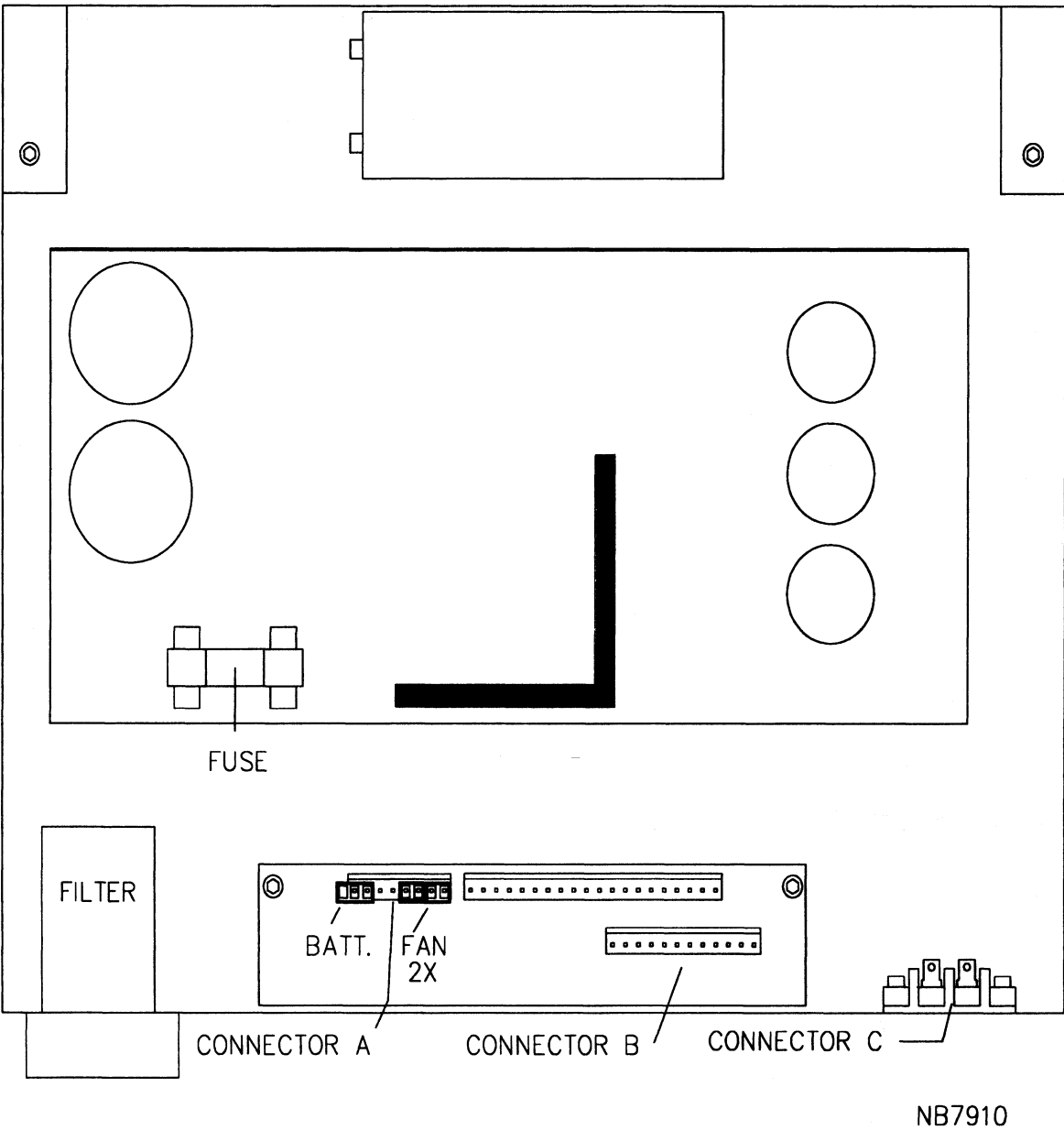
Fig. 4.5.4.-2   **Locating of the power supply fuse, 20 slots**

4.5.5. Location of the fuses in the colour monitor
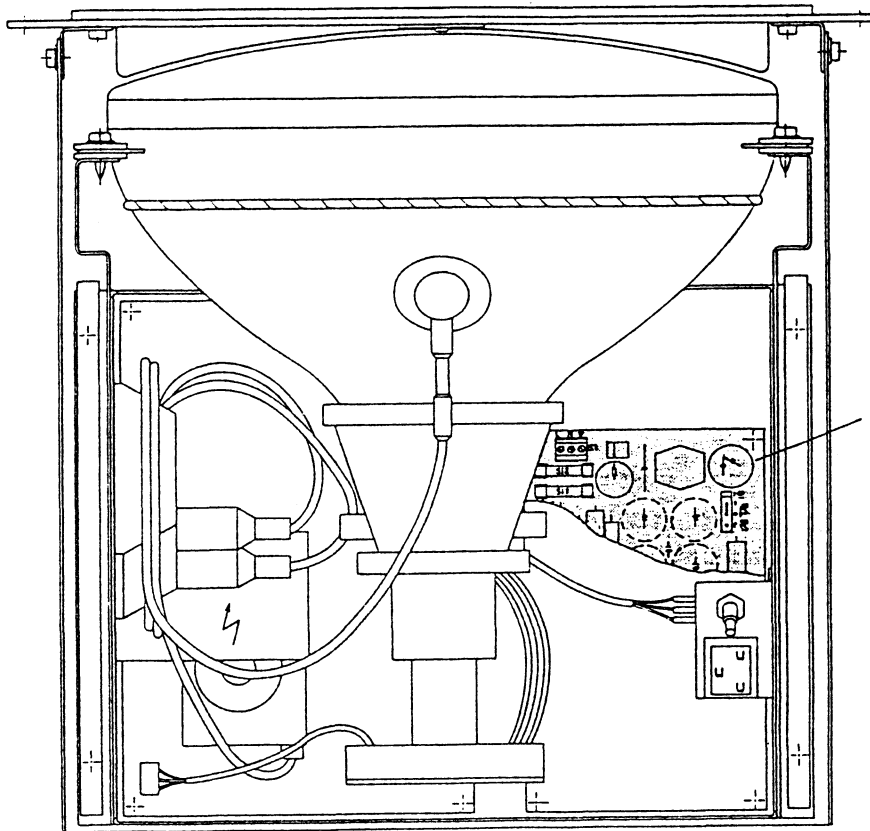
Fig. 4.5.5.-1 **Location of the power supply board of the colour monitor.**

Fuse 1.6A slow acting.

In case of supply faillure check and, if necessary, replace the fuses.
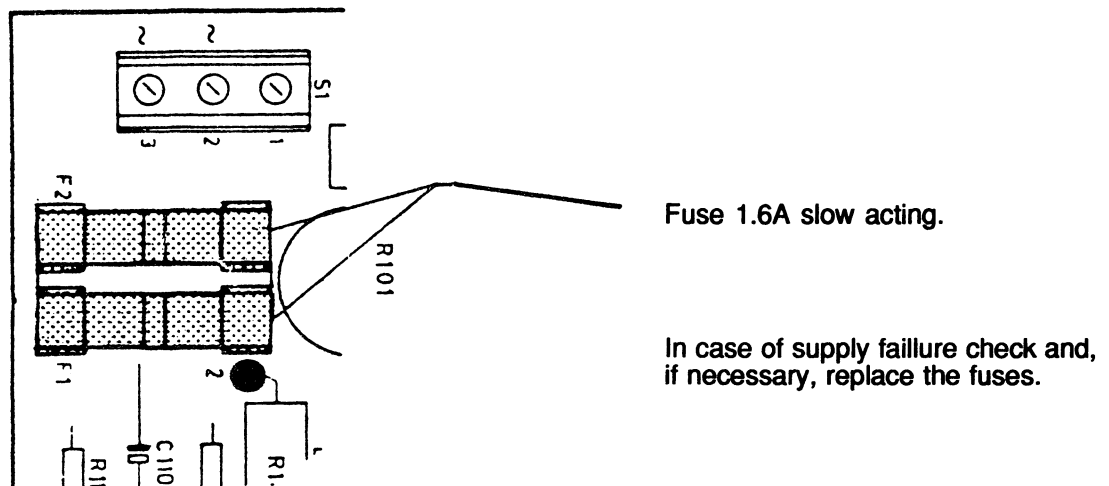
Fig. 4.5.5.-2. **Location of the fuses.**

4.5.6. Location of the fuses in the monochrome monitor
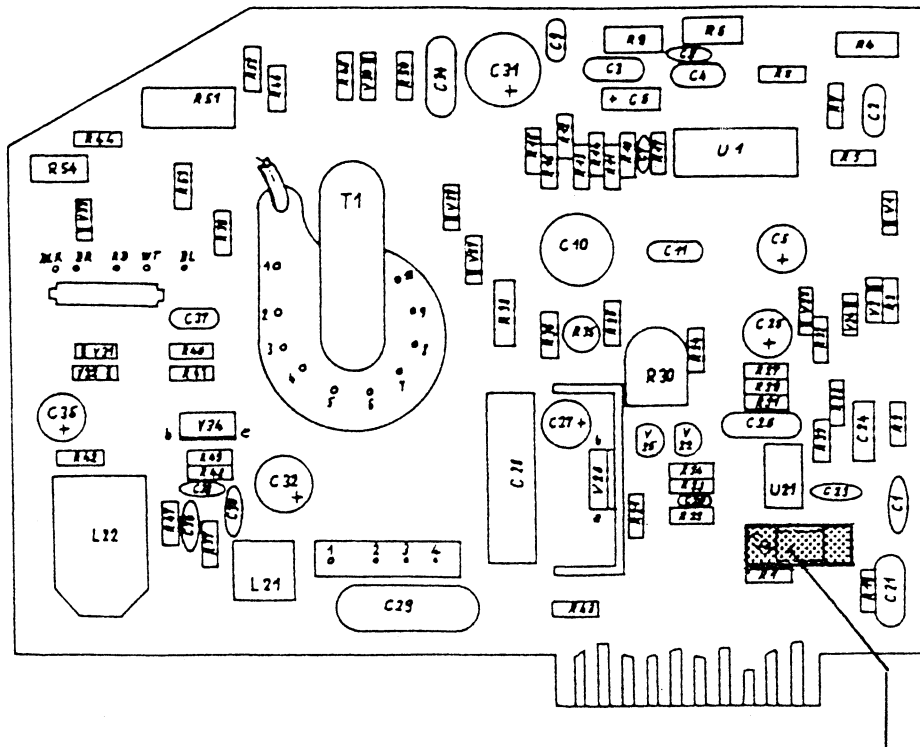
Fuse 1.25A slow

Fig. 4.5.6.-1 Location of the fuse on the printed circuit board of the 12" monochrome monitor.

4.6. POWER SUPPLY MEASURING POINTS

The dc voltages used by the CNC can be measured on the points indicated in figure 4.6.-1.

The measuring points are protected against short circuiting.

As a quick reference the +5V is forseen with a LED. This green LED is ON when the 5V of the power supply is in good working order.
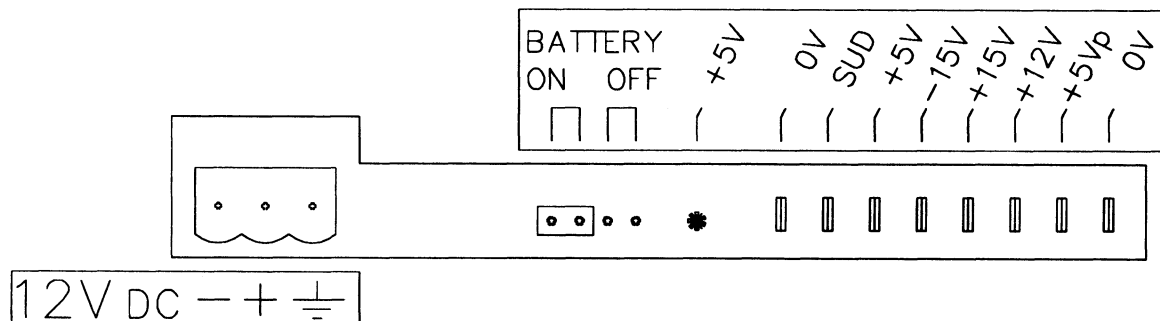
NB7911

Fig. 4.6.-1 **Measuring points on the power supply unit.**

5. CONNECTIONS TO CONTROL TELETEXT MODULE

The connection to the control teletext module consists of two cables. At first, a cable to connect a black and white or a colour monitor to the 'VIDEO' output of the control teletext module. The second cable is to connect the flat panel to the socket 'EXT.CONTROL', and/or to connect the external jog module when this option is available (see section 11.2 for connection of the external jog module). For instructions on wiring the plugs, see paragraph 3.2.
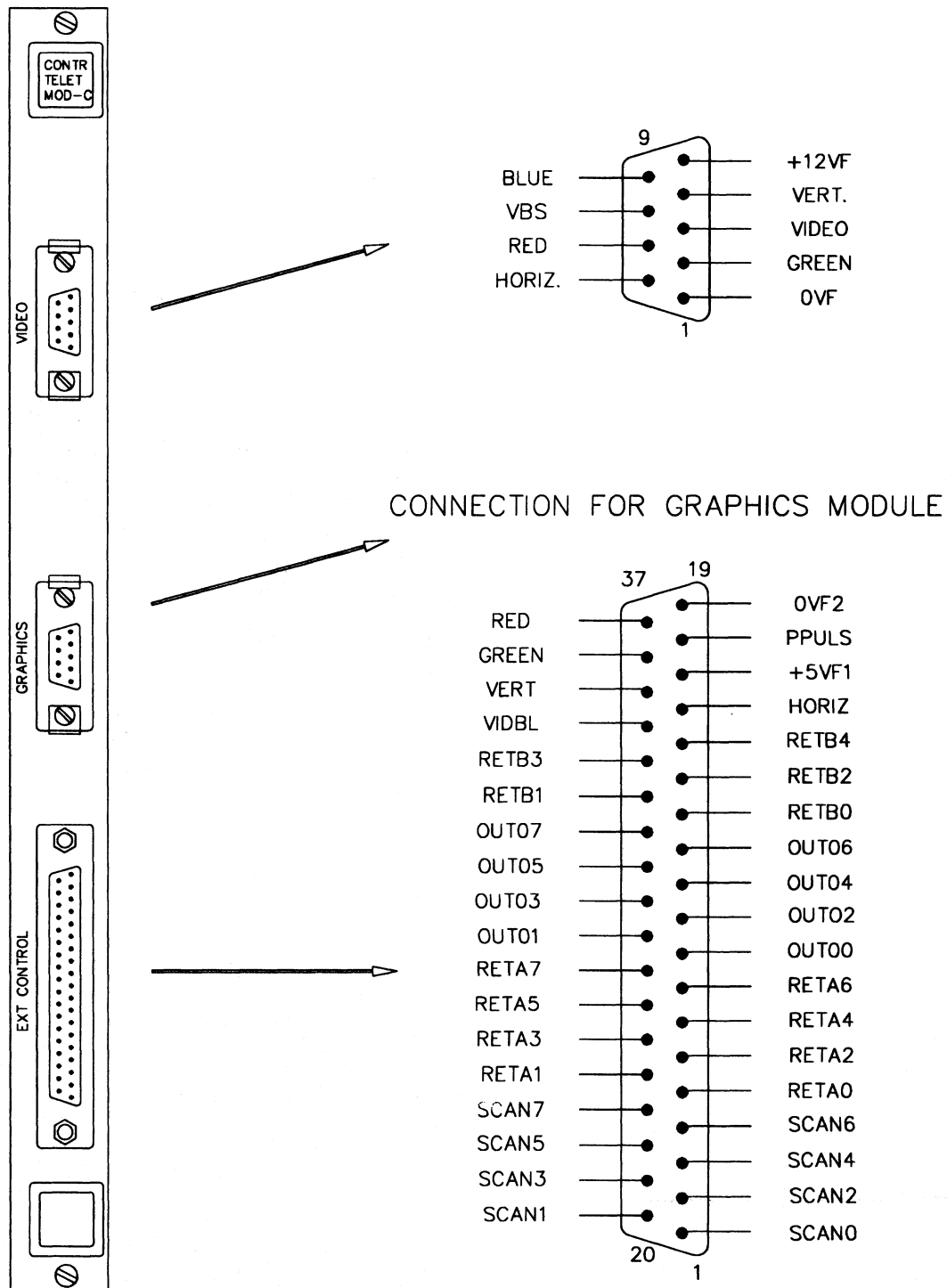
Fig. 5.-1   Connection to the control teletext module.

5.1. CONNECTING THE MONITOR

The monitor (black and white or colour) is delivered with an extension cable of 1 meter to connect the control teletext socket 'VIDEO' and the socket at the back of the monitor. If this cable is too short an extra extension cable can be used to make the connection. Chapter 5.1.1. describes the extra extension cable for the colour monitor (10 or 25 meters), and chapter 5.1.2. for the black and white monitor.

5.1.1. Connecting the colour monitor

NB7626
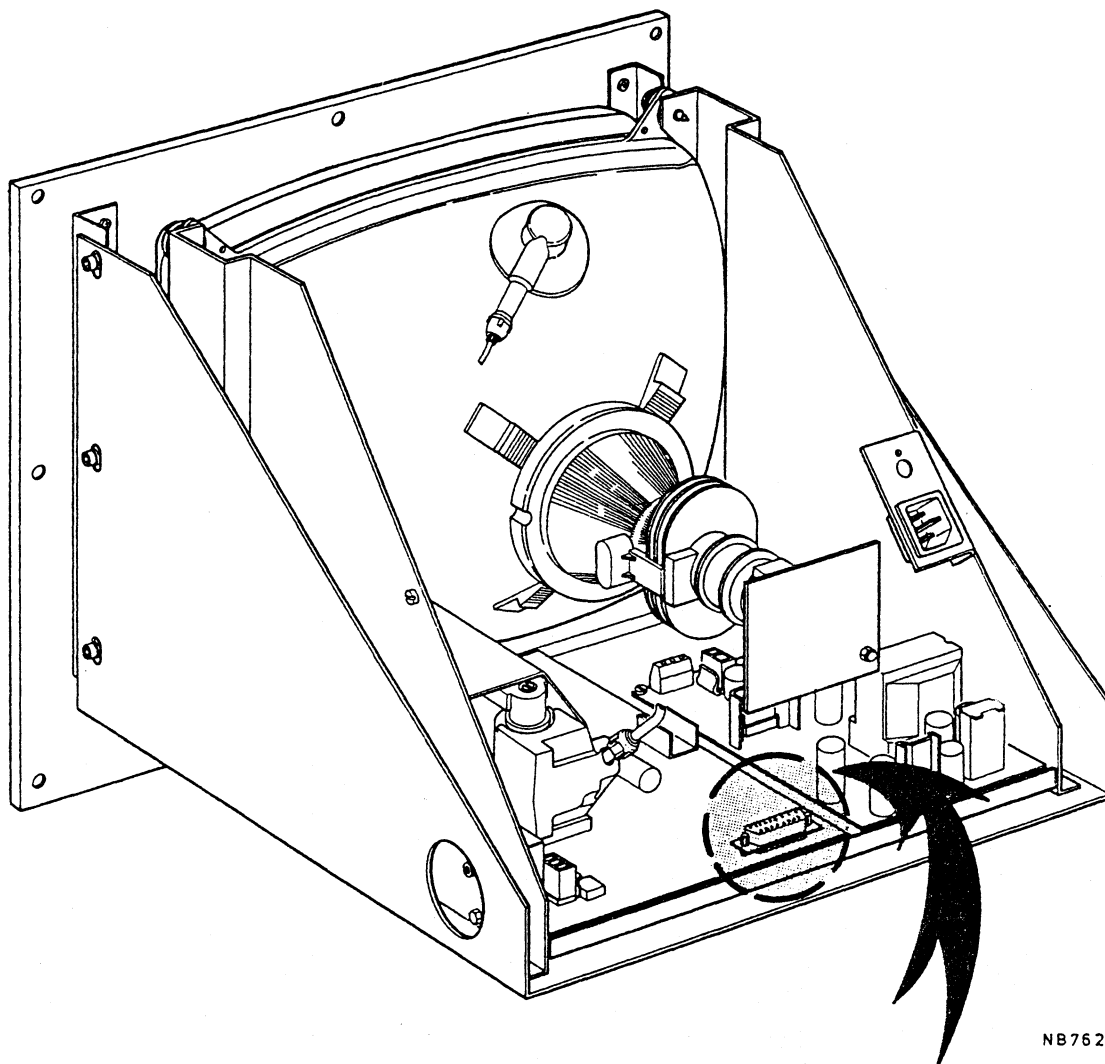
To be connected to the socket VIDEO
of the CONTR TELET MOD of the control unit

MAKE SURE THAT THE PLUG IS SCREWED SECURELY TO THE SOCKET.

Fig. 5.1.1.-1   Rear view of the 14" colour monitor showing the video connector

EXTENSION CABLE UP TO 10 METERS

Connecting the colour monitor at a distance of up to 10 meters, via the cable of one meter and an extension cable of maximum 9 meters.
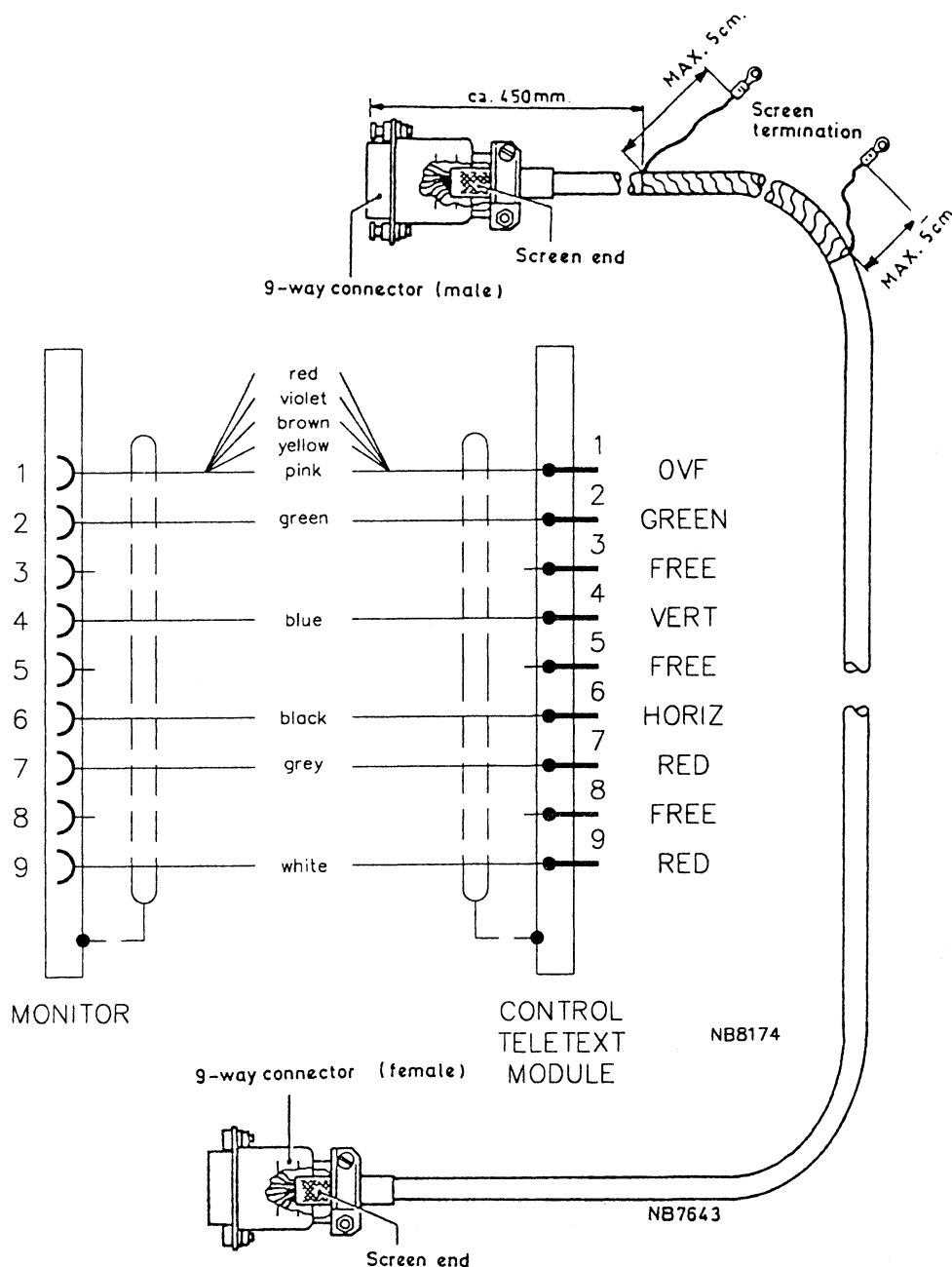
Note:
5 conductors are soldered to both pins nr. 1 (see figure). To this end, the 5 wires are stripped over 1 cm, the cores twisted together and soldered to the pins. The cable to be used is a 5 x 2-core computer cable, Philips ordering number 0712 220 04052.

Fig. 5.1.1.-2 Connection diagram extension cable (up to 10 meters)

EXTENSION CABLE UP TO 25 METERS

For connecting the colour monitor at a distance of up to 25 meters, the cable of 1 meter delivered with the monitor is not used, but only an extension cable with a maximum of 25 meters.
The connections for this extension cable are shown below.

MONITOR (15p)		CONTROL TELETEXT MODULE (9p)
3	red / violet / brown / yellow / pink	1 OVF
4	green	2 GREEN
		3 FREE
11	blue	4 VERT
		5 FREE
13	black	6 HORIZ
6	grey	7 RED
		8 FREE
2	white	9 BLUE

NB8313
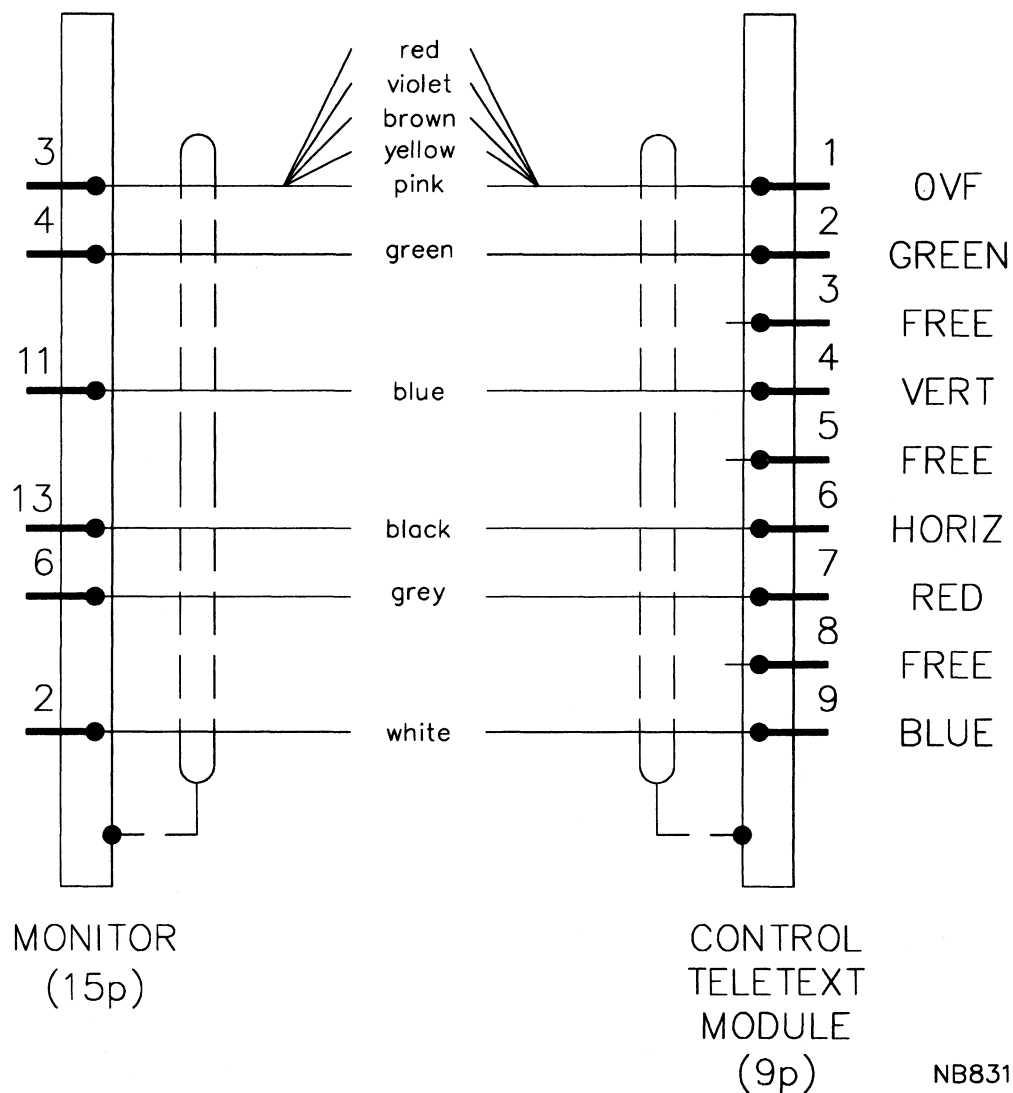
Fig. 5.1.1.-3   Connection diagram extension cable (up to 25 meters)

The construction method is the same as for the extension cable up to 10 meters, and also the same computer cable can be used.

5.1.2. Connecting the monochrome monitor
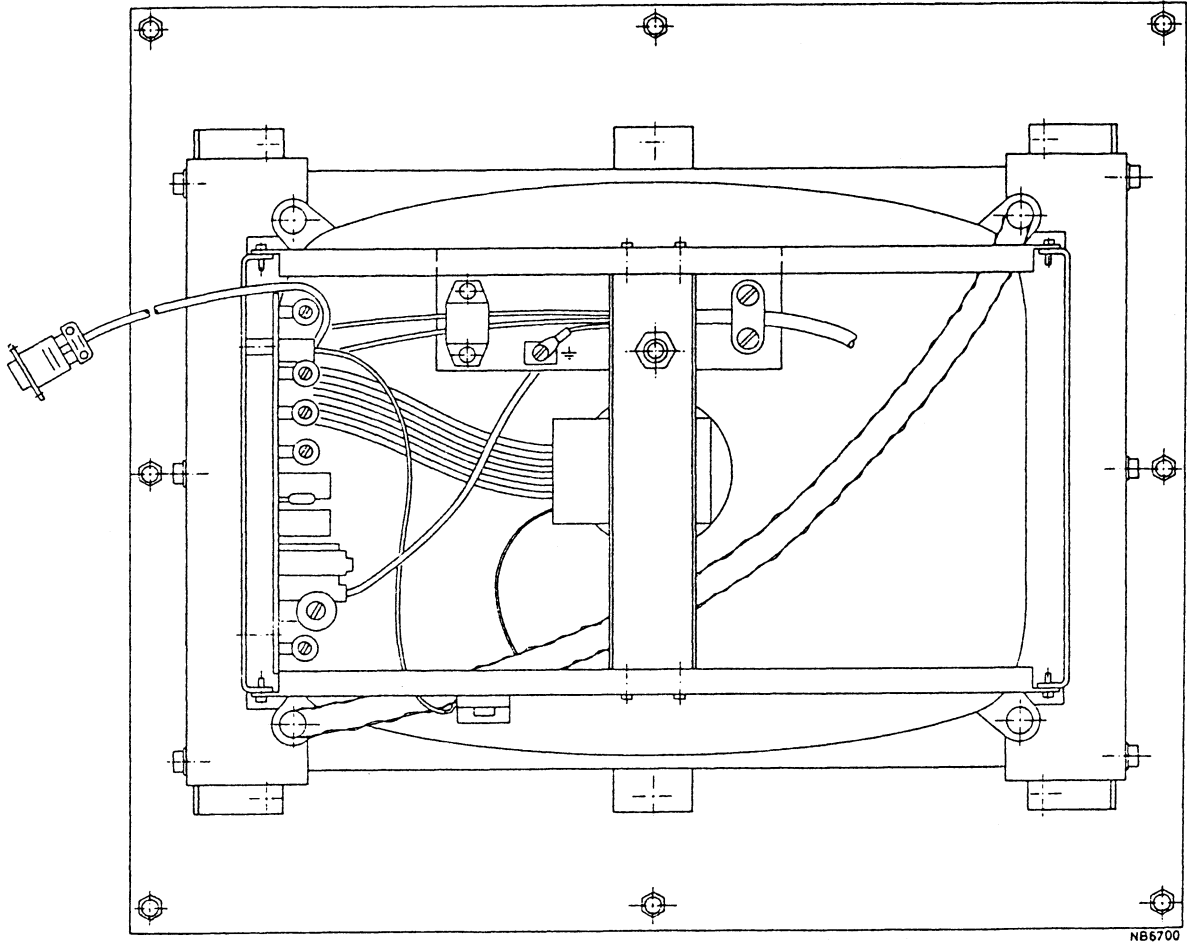
Fig. 5.1.2.-1   Rear view of 12" monochrome monitor FIMI 12GP with connection cable

Connecting the monochrome monitor at a distance of up to 10 meters via the cable of 1 meter and an extension cable of maximum 9 meters.
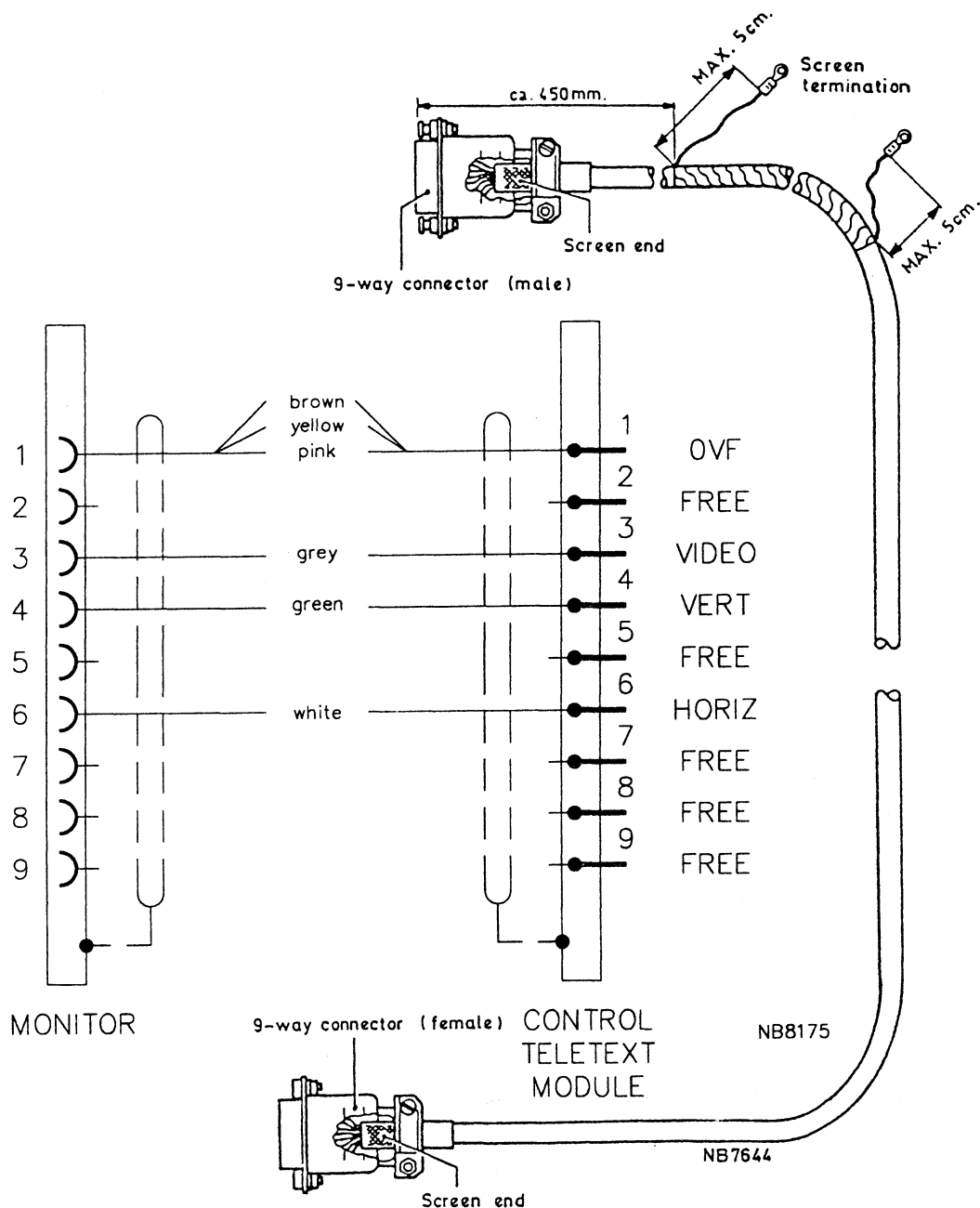
Note:
3 conductors are to be soldered to both pins nr. 1 (see figure). To this end, the 3 wires are stripped over 1 cm, the cores twisted together and soldered to the pins. The cable to be used is 5 x 2-core computer cable, Philips ordering number 0712 220 04052.

Fig. 5.1.2.-2   **Extension cable for monochrome monitor 12" FIMI 12GP. Connection diagram.**

5.2. CONNECTING THE FLAT PANEL

A connection cable must be made between the socket on the flat panel and the socket 'EXT.CONTROL' on the control teletext module. The cable is a 18 core screened cable and must not exceed 25 meters.

The connection diagram of this cable is shown in figure 5.2.-1.

When fitting this cable to the 37-way plug of the control teletext, *do not wire pins 16, 34, 35, 36 and 37* of this plug, otherwise this will cause interference to the control.

37-way plug	Signal	25-way plug
1	SCAN0	19
2	SCAN2	21
3	SCAN4	23
4	SCAN6	25
5	RETA0	3
7	RETA2	1
8	RETA4	16
13	RETA6	5
20	RETB0	17
20	SCAN1	7
21	SCAN3	9
22	SCAN5	11
23	SCAN7	13
24	RETA1	2
25	RETA3	14
26	RETA5	18
27	RETA7	4
32	RETB1	15
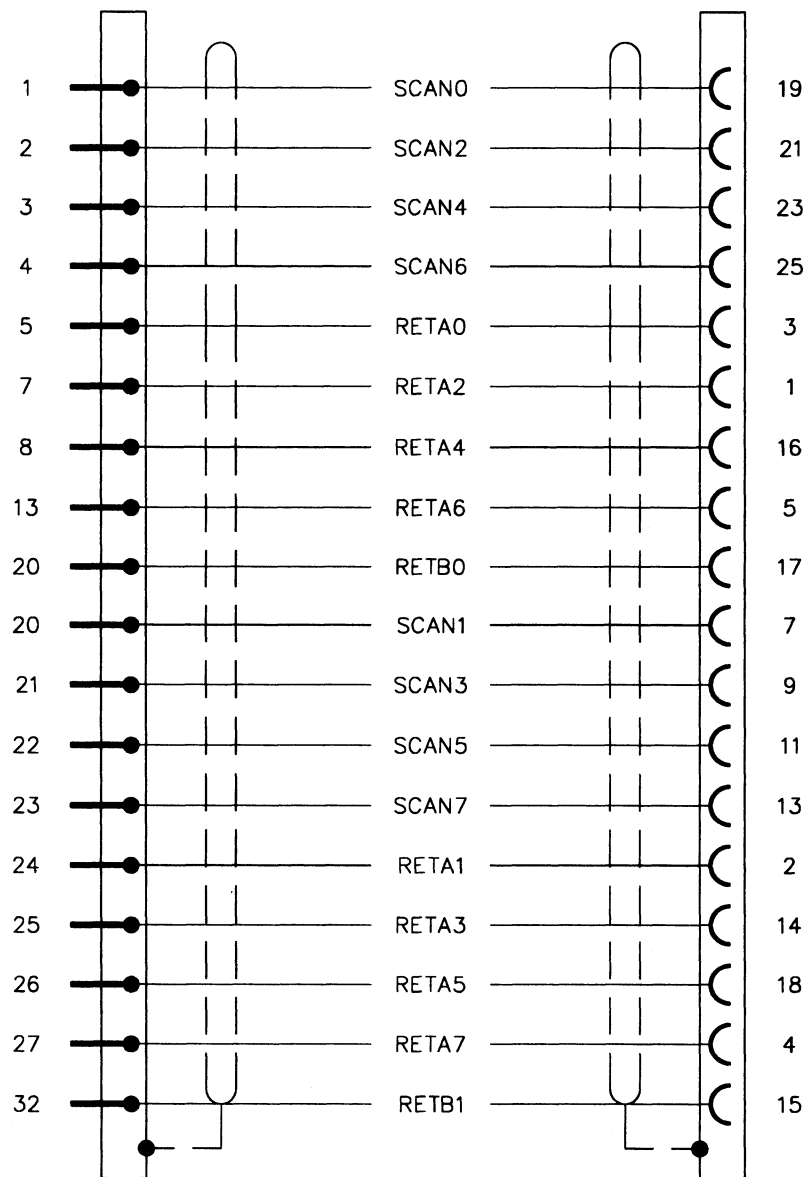
NB8176

37-way connector to 'EXT.CONTROL' socket on contr. tel. mod.

25-way connector to control panel

Fig. 5.2.-1 **Connection diagram for cable to connect the control**

6.   CONNECTIONS TO THE DRIVE MODULES

6.1.   MEASURING SYSTEMS

The CNC3460/3560 allows two types of measuring systems to be connected: at first, incremental rotary measuring system meeting the specification given in this section (6.1.1) and secondly, the Philips PE 2580/20 or PE 2580/30 linear measuring system (6.1.2).
The type of measuring system to be employed in the different axes depends on the machine tool requirements.
For the integration of the rotary measuring system into the machine tool and for the wiring of the connector on the transducer side refer to the relevant manufacturer's documentation.
Instructions on integration and wiring of the Philips PE 2580 measuring system can be found in the relevant installation manual, provided with the NC system.
The measuring systems are connected to the sockets X3 and X4 labelled MEAS SYST on the pc boards designated LS/LS DRIVE MOD, LS/RM DRIVE MOD, RM/RM DRIVE MOD and RM DRIVE MOD in the control unit; see figure 6.-1 below.
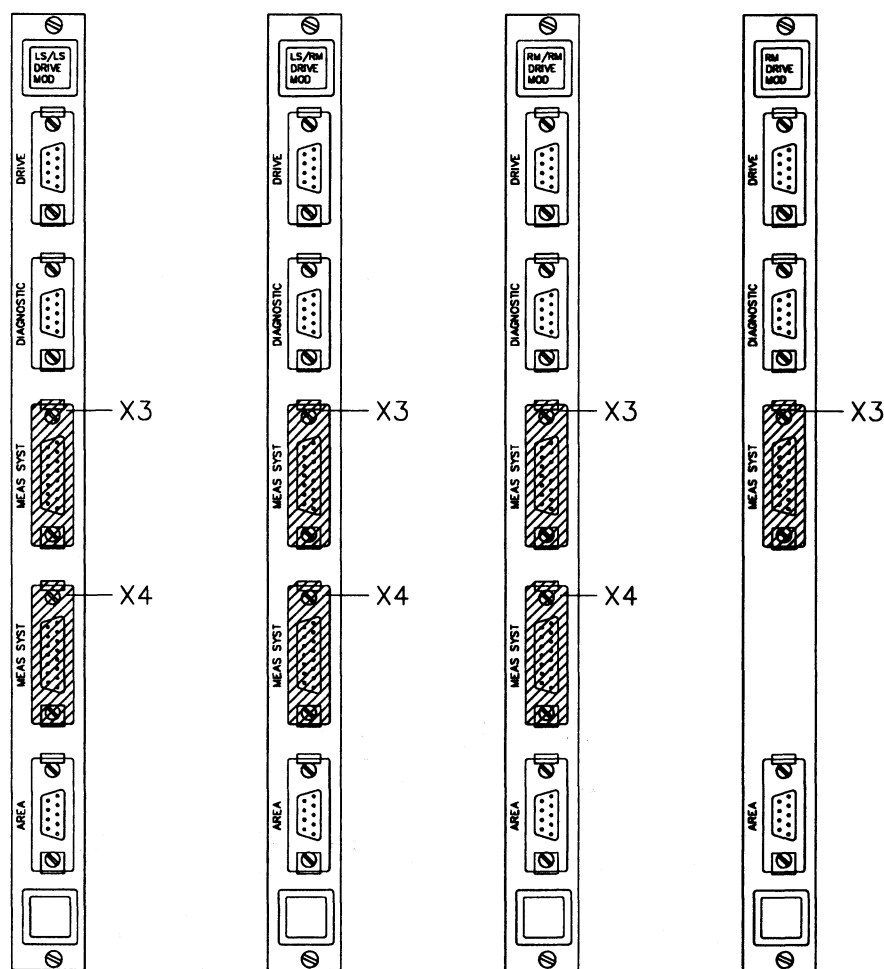
NB7950

RM/RM DRIVE MOD
X3 = Rotary measuring system
X4 = Rotary measuring system

LS/RM DRIVE MOD
X3 = Philips lin. measuring system
X4 = Rotary measuring system

LS/LS DRIVE MOD
X3 = Philips lin. measuring system
X4 = Philips lin. measuring system

RM DRIVE MOD
X3 = Rotary measuring system

Fig. 6.-1   Measuring system connections

6.1.1. ROTARY MEASURING SYSTEM

6.1.1.1. Inputs

The inputs for the rotary transducer on the measuring system boards in the control unit are suited for S00/S90 signals. A separate input is provided for a positive or negative going marker signal.
The rotary transducers must be supplied from the control unit's power supply. Both +5V and +12V are available.

Input signal characteristics:

Voltage	Logic "1"	Logic "0"	I sink
+12V	+12V min. +9V	0V max. +3V	16mA
+ 5V	+ 5V min. +3.5V	0V max. +1.5V	16mA

Maximum frequency of S00/S90 signals: 100 kHz
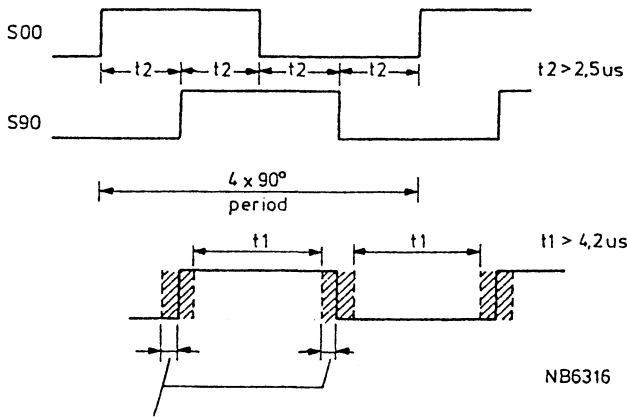
Maximum jitter on signal edges: ± 15°

Fig. 6.1.1.1.-1 Wave form of S00/S90 signals

Marker pulse

The marker pulse should be a positive or negative going pulse having a minimum duration of 100 us. On delivery, the relevant input is set for a positive going pulse. Negative going pulses require the pc board concerned to be removed from the control unit and set accordingly. For a LS/RM module the setting consists in transposing the wire link from position "a" to position "b", as detailed in figure 6.1.1.1.-2.
Before extracting a pc board from or inserting it into the control unit always be sure that the latter is disconnected from the mains supply.

NB8164

Measuring system connected to	Positive going pulse (standard)	Negative going pulse
X4	W3 a	W3 b
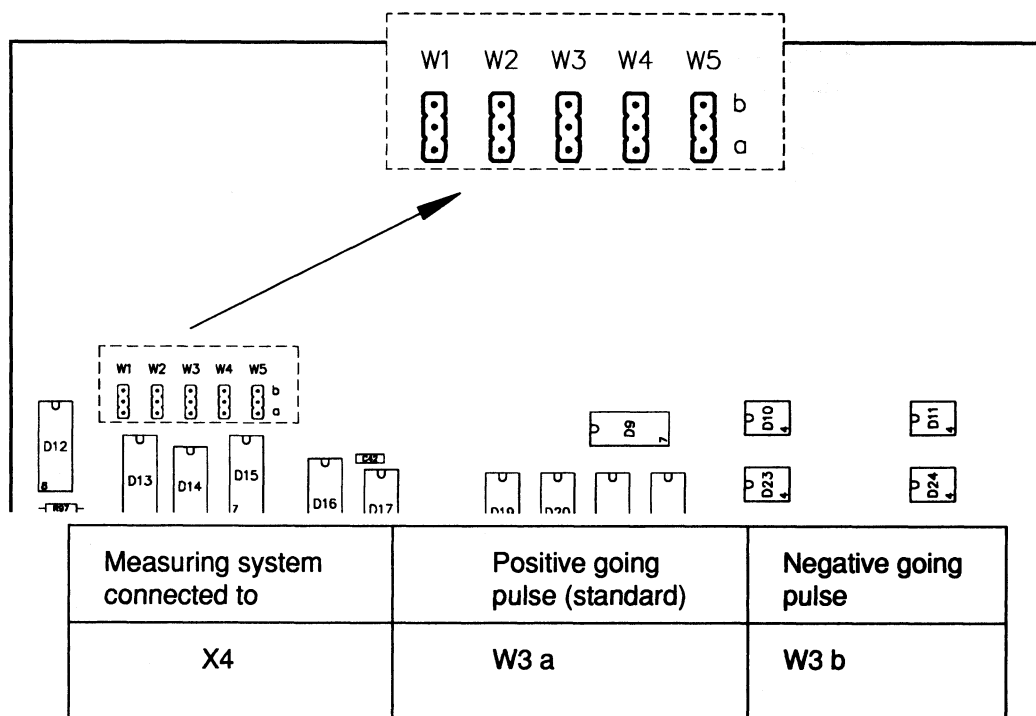
Fig. 6.1.1.1.-2   Changing the input polarity of the marker pulse on the LS/RM module

In case of a RM/RM module 4022 226 3645 or a RM module 4022 226 3656 the setting is done by transposing a jumper for the relevant axis as shown in figure 6.1.1.1-3.

NB8170

Measuring system connected to	Positive going pulse (standard)	Negative going pulse
X3	W6 a	W6 b
X4	W3 a	W3 b
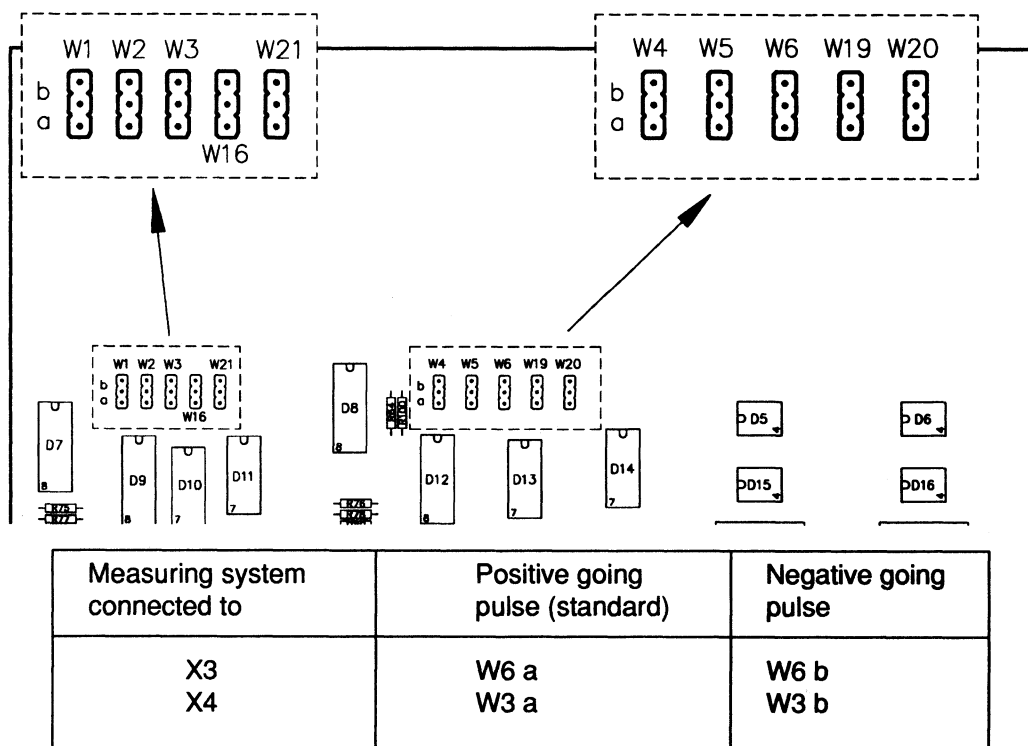
Fig. 6.1.1.1.-3   Changing the input polarity of the marker pulse on the RM/RM module or RM module

Alarm input

Each rotary measuring system is equipped with a phase detector which will give an alarm to the processor when the S00/S90 signals are not correct.

If the rotary transducer is equipped with an alarm output, it is possible to connect this alarm also. If the alarm input is not used set jumper W5 in position A (position at delivery).
To obtain alarm input without cable breakage detection leave jumper W5 in position A. This position is suitable for three state alarm outputs. To obtain alarm input with cable breakage detection set jumper W5 in position B.

NB8164

PC Board LS/RM Drive Module	W5a Alarm not used or Alarm input only	W5b Alarm + cable breakage detection
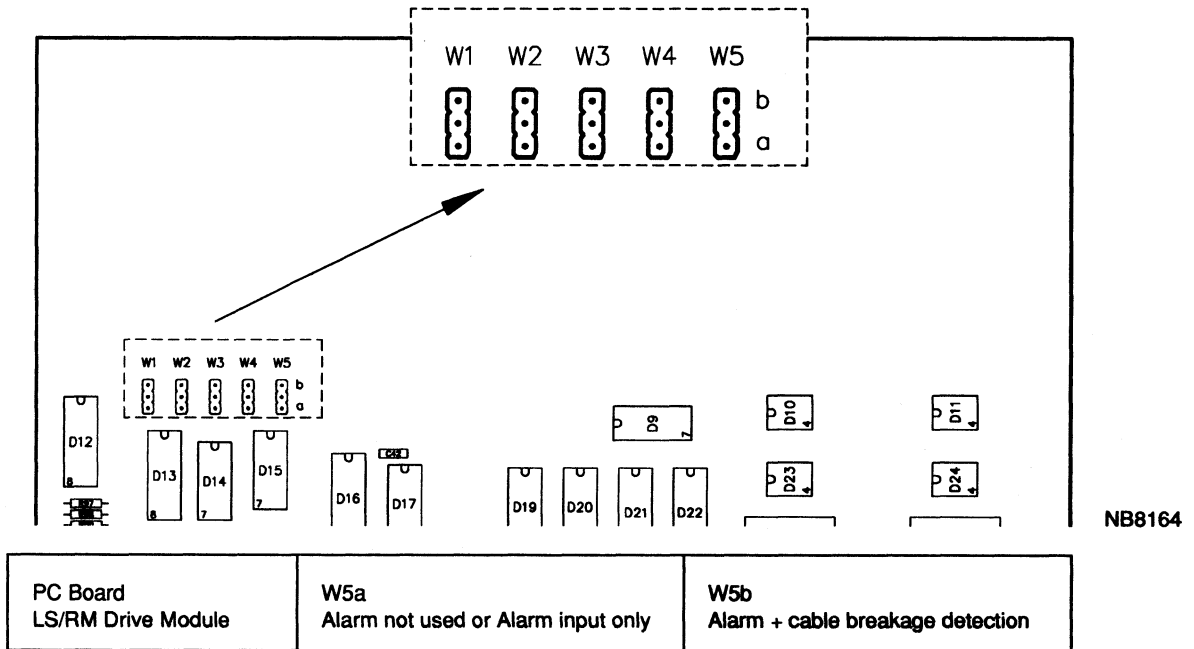
Fig. 6.1.1.1.-4   Alarm selection on the LS/RM module

NB8180

PC Board RM or RM/RM Drive Module	W5a Alarm not used or Alarm input only	W5b Alarm + cable breakage detection
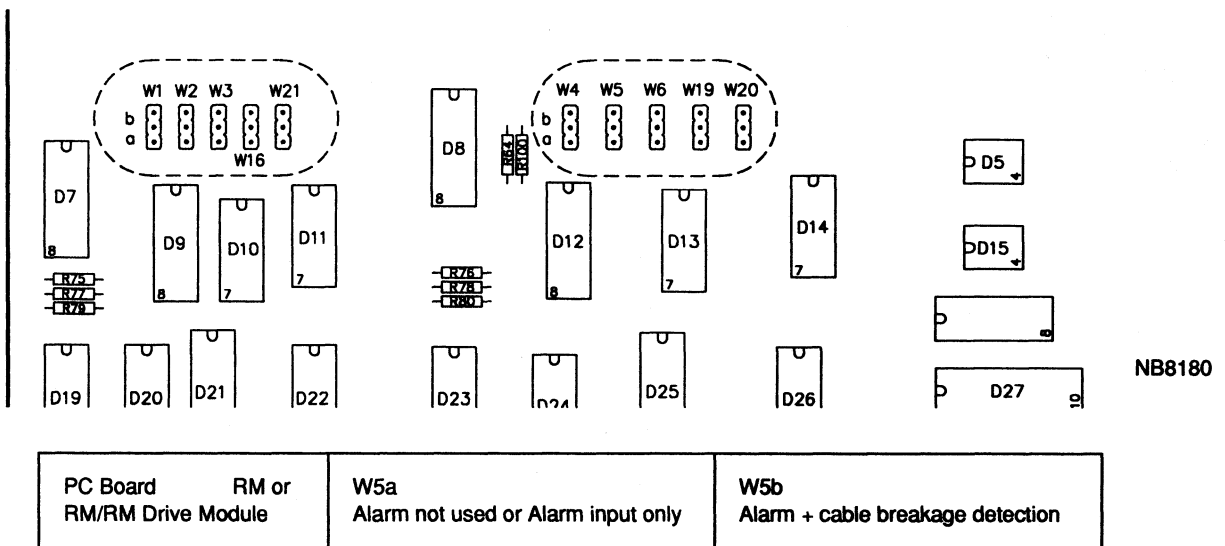
Fig. 6.1.1.1.-5   Alarm selection on the RM/RM module or RM module

It is also possible to use the alarm input as a cable breakage/presence detection. This cable breakage detection can be done in two ways:

1.   Externally by means of a 5V or 12V voltage supplied by the rotary measuring system
2.   Internally by means of a 5V or 12V voltage supplied to the alarm input inside the connector at RM module side. In this case the detection is restricted to the removal of the connector.

If the 12V supply is used for the cable breakage detection a resistor of 681 ohm must be connected between the 12V supply and the alarm input.
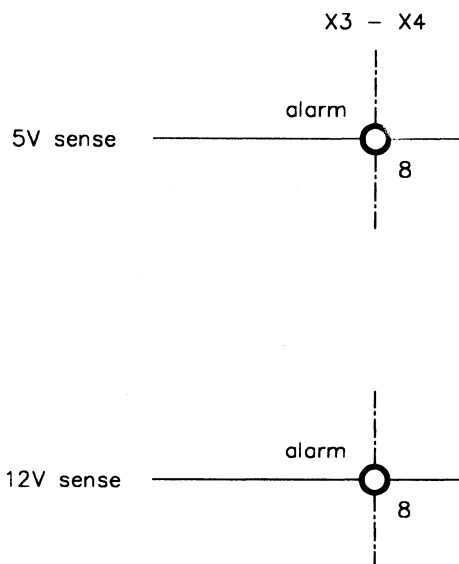
NB8163

Cable breakage detection
externally
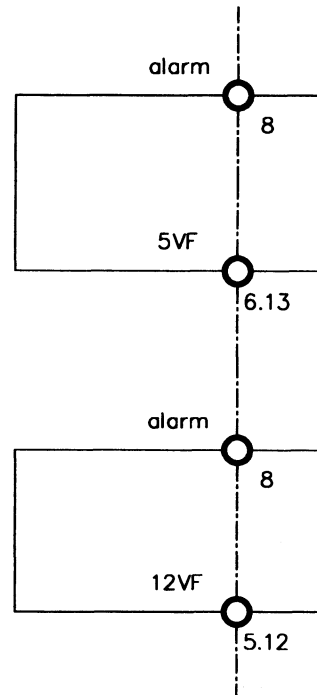
Cable presence detection
internally

6.1.1.2.   Supplying the rotary transducer

The rotary transducer can be supplied from the control unit's power supply. Following supply voltages are available:

+12V, max. load per axis 300mA
+ 5V, max. load per axis 300mA

The selection of the transducer's supply voltage is made on the plug; see figure 6.1.1.4.-1 and 6.1.1.4.-2. Since the RM-module can deliver up to 300 mA per axis, the supply wires in the cable must be thicker (0.75 mm^2) than the other wires (0.14 mm^2).

6.1.1.3.   Resolution of the rotary transducer

The resolution is given by the formula: $(A \times B) : C$

where

A =   number of S00/S90 pulses per revolution of the transducer; refer to manufacturer's specification

B =   multiplication factor (1, 2 or 4); to be assigned to machine constants 203/253/303/353/403/453 (axes) and 503 (spindle)

C =   transmission: the displacement of the machine slide in microns per revolution of the transducer (machine tool specification)

6.1.1.4.   Connection cable from the rotary transducer

Wiring instructions for the connection cable and the associated plug are given in section 3.2. The cable is screened and contains $2 \times 3 \times 0.14 \text{ mm}^2 + 2 \times 0.75 \text{ mm}^2$ wires; it can be ordered from Philips with ordering number 0722 491 00001.

Figure 6.1.1.4.-1 and 6.1.1.4.-2 show the wiring-diagrams to connect respectively 5V and 12V rotary transducers with the RM drive-module.
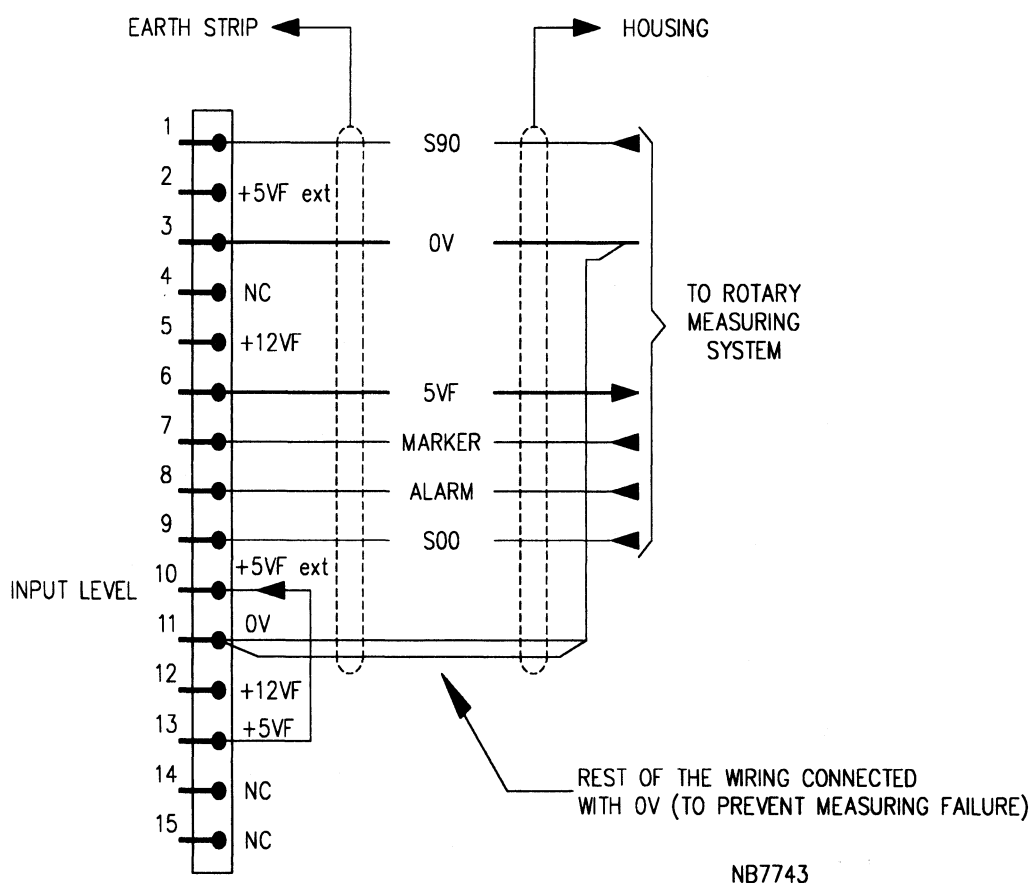
Fig. 6.1.1.4.-1   Wiring diagram of cable for 5V rotary transducers
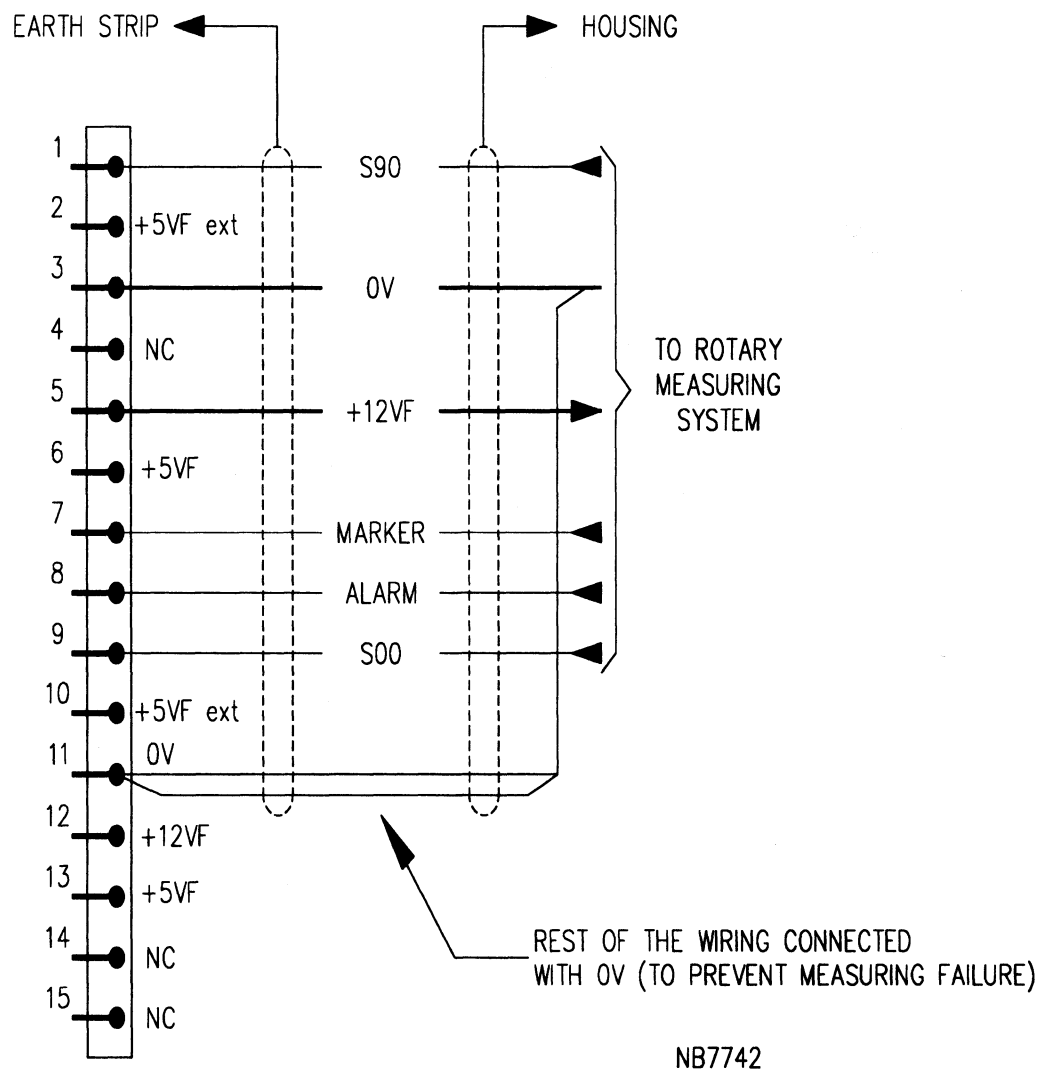
Fig. 6.1.1.4.-2   **Wiring diagram of cable for 12V rotary transducers**

6.1.2. PHILIPS LINEAR MEASURING SYSTEM

When applying linear measuring systems, the CNC 3460 requires the Philips PE2580/20 or PE2580/30 linear transducer to be employed. Wiring instructions on fitting the 15-way plug to the cable are given in section 3.2. The 15-way plug is provided with the control system, the 7-way plug with the transducer.

SCREENED 7 CORE CABLE
ORDERING NUMBER DT 0712 187 09001
SCREENING CONNECTED TO HOUSING
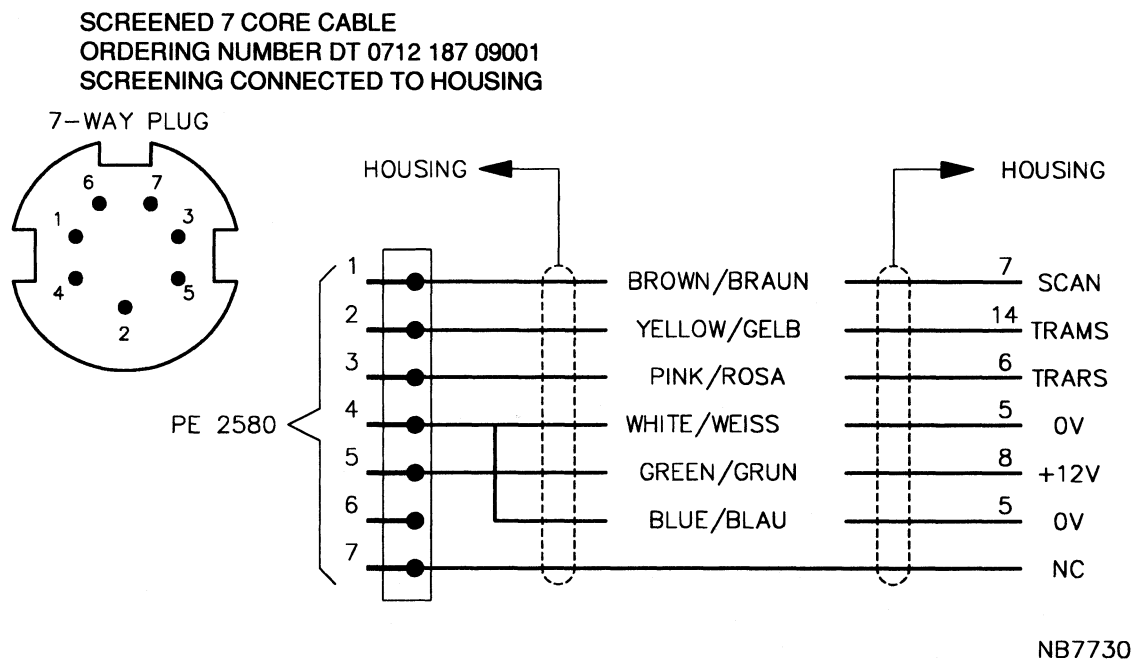
Fig. 6.1.2.-1   Wiring of cable to connect linear transducer PE2580/20

SCREENED 7 CORE CABLE
ORDERING NUMBER DT 0712 187 09001
SCREENING CONNECTED TO HOUSING
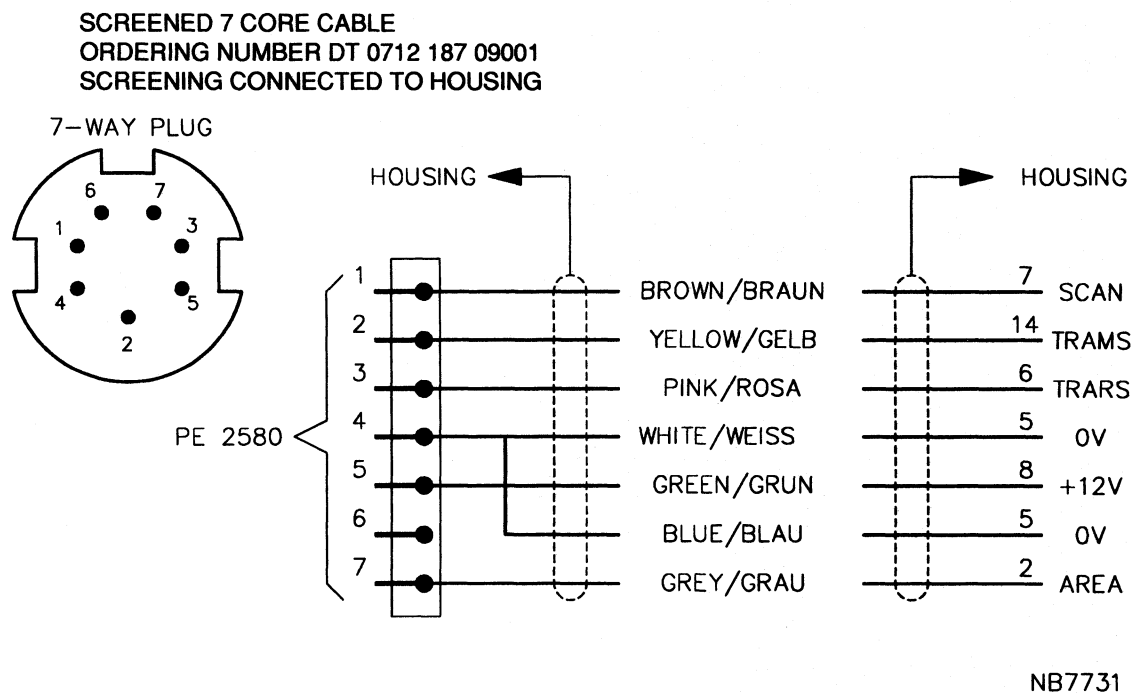
Fig. 6.1.2.-2   Wiring of cable to connect linear transducer PE2580/30

6.2. ANALOG OUTPUT SIGNALS

The control system can provide analog output signals for closed loop servo drive system for 3 axes at most.
The respective servo drive-systems must have *DIFFERENTIAL* inputs.

The characteristics are:
>Output voltage : +10V ... 0V ... -10V
>Output current : 5 mA max.

For the adjustments see machine constants:

- MC 215 to 220 for the first axis	- MC 365 to 370 for the fourth axis
- MC 265 to 270 for the second axis	- MC 415 to 420 for the fifth axis
- MC 315 to 320 for the third axis	- MC 465 to 470 for the sixth axis

Note: Rapid traverse rate should be achieved at +9V / -9V input voltage on the velocity loop amplifier.

The analog output signals are available on the sockets DRIVE (X1) of the PC boards LS/LS DRIVE MOD, LS/RM DRIVE MOD, RM/RM DRIVE MOD and RM DRIVE MOD; see fig. 6.2.-1.
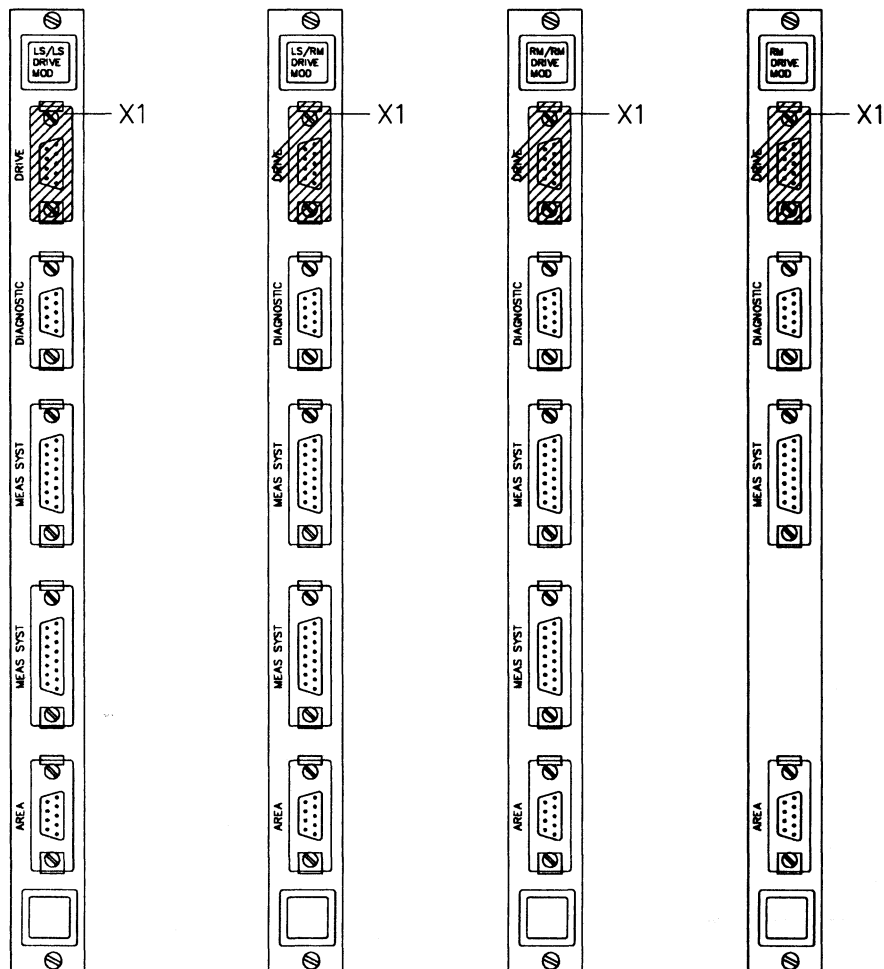
NB7951

Fig. 6.2.-1   **Analog outputs (X1) on drive modules**

Wiring instructions on the connection cable for the analog output signals are given in section 3.2.

Note: The analog outputs may *never* be short-circuited, otherwise the drive modules will fail.
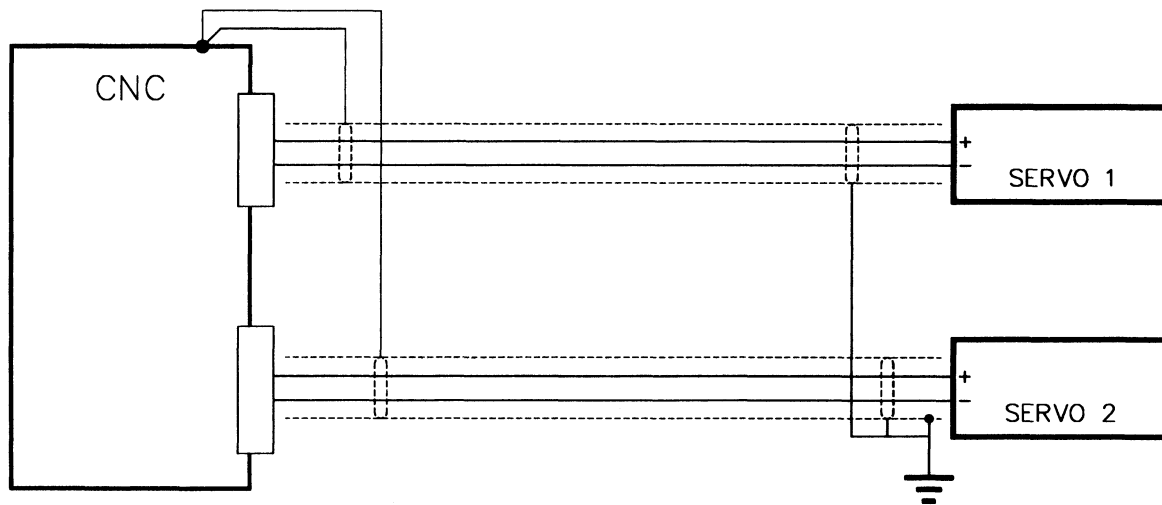
NB6321

Fig. 6.2.-2   **Cabling for differential inputs**

The figure below shows the wiring-diagram to connect the analog drive-signals with the servo-drives on the machine-tool.

The cable, 4 x 0.14 mm^2 with screen, can be ordered from Philips with ordering number 0712 187 03005.
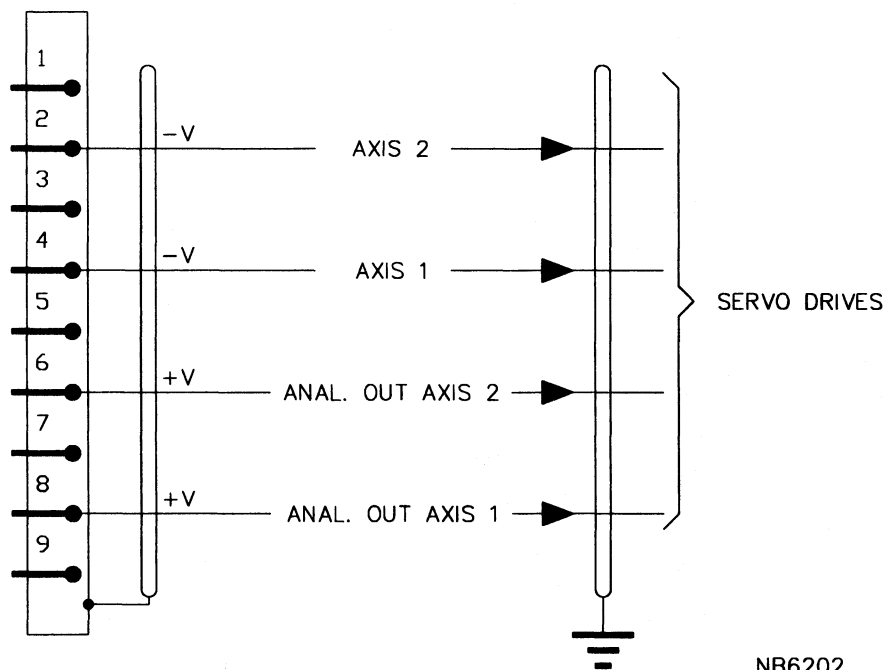
NB6202

Fig. 6.2.-3   **Wiring of 9-way plug on cable for analog output signals**

6.3.   MACHINE REFERENCE POINT INPUT SIGNALS

6.3.1.   General information

The NC system allows the machine reference point to be automatically searched for, provided this point has been constructionally defined. The search procedure has been described in the installation manual for the linear transducer PE 2580.

When employing a rotary transducer, this information may be helpful in understanding the principles of transducer and area switch alignment for automatic reference point search.

The machine tool has to be equipped with cams at the end of the slides. The cam should overlap the emergency limit switch.

The minimum spacing between the front end of the cam and the limit switch amounts to 32 mm. The cam causes a micro switch, referred to as area switch, to be actuated. At the start of the reference point search a displacement along the axis towards the cam at "search velocity" (RPF feed) is performed (defined by machine constants 231, 281, 331, 381, 431 and 481). When the cam opens the micro switch, the direction of displacement is reversed. Now a movement in opposite direction takes place at search velocity (RPF feed), until the switch closes.

When the switch closes, again the direction of displacement is reversed, but the movement in the opposite direction is at "creep speed" defined by machine constants 232, 282, 332, 382, 432, 482 (RPF creep speed), until the switch opens again.

When the switch opens again, the actual position of the axis is reset to zero and the movement at creep speed will continue until the first marker pulse is given in order to reset the axis or preset it to a given position, defined by machine constants 233, 283, 333, 383, 433 and 483 (RPF zero offset).

The positions of the software limit switches in the axis concerned are now established. Note that the software limit switches are effective only if the reference points in all axes have been established.

When jogging operations are performed before the positions of the software limit switches are established, the cams and the micro switches operate as safeguarding devices and will stop any movements in the axis, so that the emergency switches will not be actuated. For that reason it is recommended that a cam be fitted at the other end of the machine slide as a precaution.

The Philips linear measuring system generates a marker pulse at a fixed position within every 635 um pitch.

When a rotary transducer is employed, a marker pulse is generated at a fixed position once per revolution.

Area switch characteristics:

When the area switch is *closed (not actuated)*, the current drawn is 16 mA.

The maximum length of the connection cable (0.34 mm^2) is 100 m.

Axis related machine constants:

1.   RPF direction (direction in which reference point search starts)

       MC 230 axis 1   +1 or -1 direction, forward or backward
       MC 280 axis 2
       MC 330 axis 3
       MC 380 axis 4
       MC 430 axis 5
       MC 480 axis 6

2.   RPF feed during reference point search

       MC 231 axis 1   max. 32 m/min, in increments of 0.1 mm/min
       MC 281 axis 2   min. value 0.1 mm/min
       MC 331 axis 3
       MC 381 axis 4
       MC 431 axis 5
       MC 481 axis 6

3. RPF creep speed during marker pulse search

       MC 232 axis 1   max. 10 m/min, in increments of 0.1 mm/min
       MC 282 axis 2   min. value 0.1 mm/min
       MC 332 axis 3
       MC 382 axis 4
       MC 432 axis 5
       MC 482 axis 6

4. RPF zero offset, preset value of reference point relative to machine datum point

       MC 233 axis 1   offset value in increments
       MC 283 axis 2
       MC 333 axis 3
       MC 383 axis 4
       MC 433 axis 5
       MC 483 axis 6

5.   Positions of software limit switches

       MC 235   the positive distance in increments
       MC 236 }axis 1   the negative distance in increments
       MC 285
       MC 286 }axis 2
       MC 335
       MC 336 }axis 3
       MC 385
       MC 386 }axis 4
       MC 435
       MC 436 }axis 5
       MC 485
       MC 486 }axis 6

Note: All distances are relative to the reference points in the axes.

6.3.2.   Connecting the AREA-switches

The area switches are connected to the sockets AREA (X5) on the pc boards LS/LS DRIVE MOD, LS/RM DRIVE MOD, RM/RM DRIVE MOD or RM DRIVE MOD in the control unit.
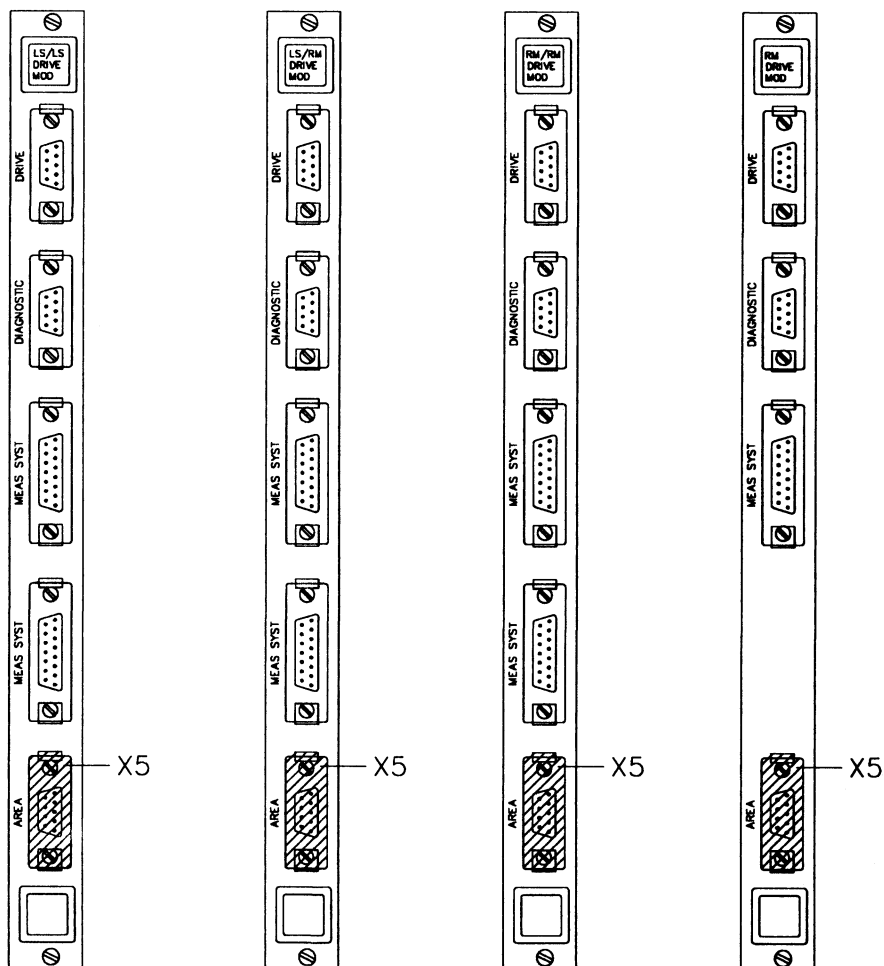
NB7952

Fig. 6.3.2.-1   Position of AREA-switch sockets (X5) on drive modules

Figure 6.3.2.-2 shows how the plugs must be wired to get the correct connection of the AREA-switches with the RM/RM drive module.
Figures 6.3.2.-3 and 6.3.2.-4 show the diagrams for respectively the LS/RM and LS/LS drive module.
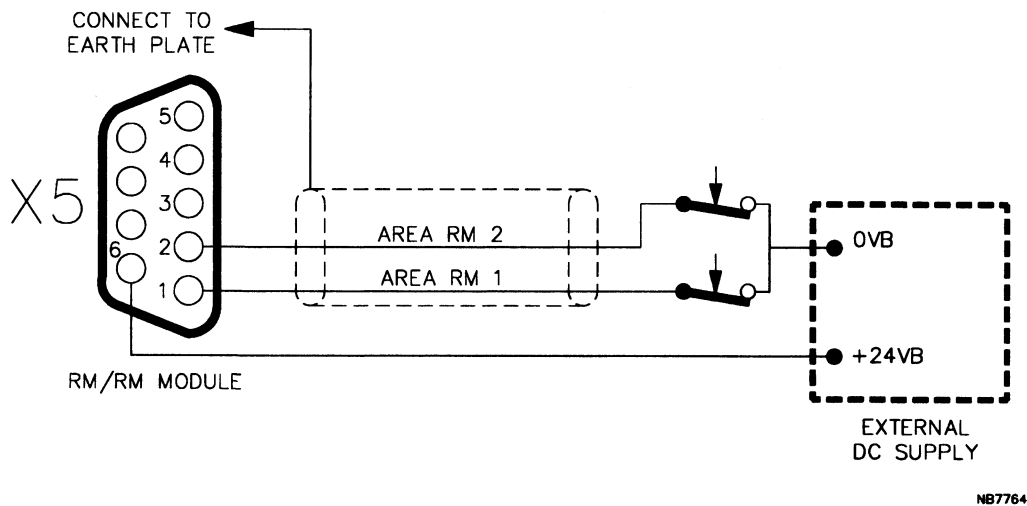
Fig. 6.3.2.-2   Wiring of the 9-way plug on cable from AREA-switch for the *RM/RM* drive module
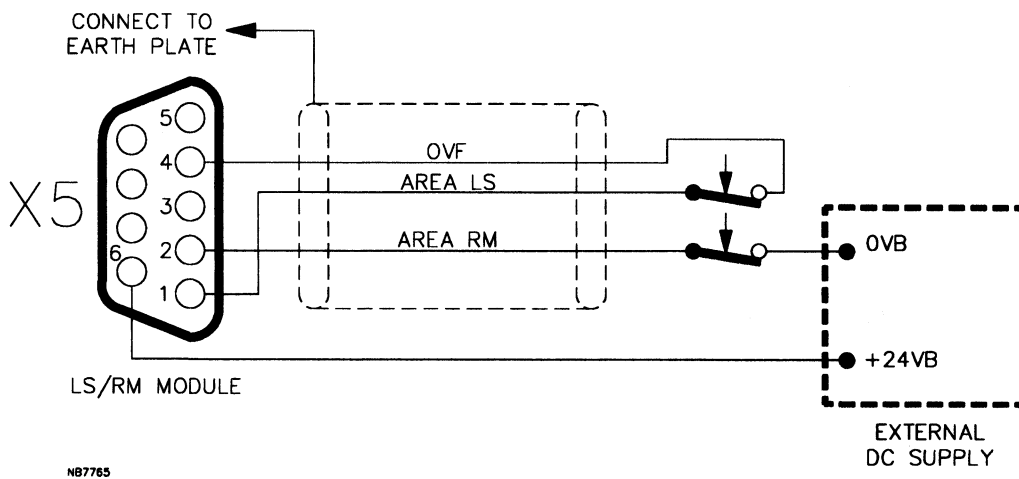
Fig. 6.3.2.-3   Wiring of the 9-way plug on cable from AREA-switch for the *LS/RM* drive module

CONNECT TO
EARTH PLATE

OVF

AREA LS 2

AREA LS 1

X5

LS/LS MODULE
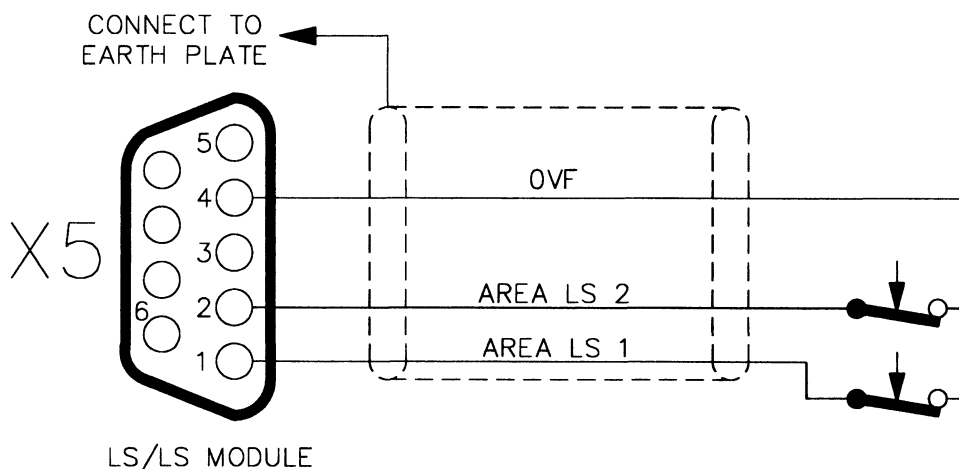
NB7766

Fig. 6.3.2.-4   Wiring of the 9-way plug on cable from AREA-switch for the *LS/LS* drive module

6.3.3.   Setting-up the reference point

The zero point (reference point) can be set at any pitch of the scale. A signal (area signal) must be generated at that pitch to signify this location.

The area signal is generated from a micro switch activated by a cam placed at the appropriate position opposite to it. The switch then sends this signal to the CNC.

In order to do this, the signal processing circuitry regards a part of the pitch as a "safe area" and a part as a "forbidden area" for the area signal to be generated. In this way the following reference point will always be captured within the same pitch (figure 6.3.3.-1) and not carry-over to the next pitch, 635 um later.

Care must be taken however, so that the area signal is generated approximately in the middle of the safe area. So a repeatability tolerance of 0.3 mm of the area switch is allowed.

PITCH OF SCALE (635 um)

SAFE AREA

FORBIDDEN AREA

REFERENCE POINT

NB7776

ACTIVATE AREA SIGNAL
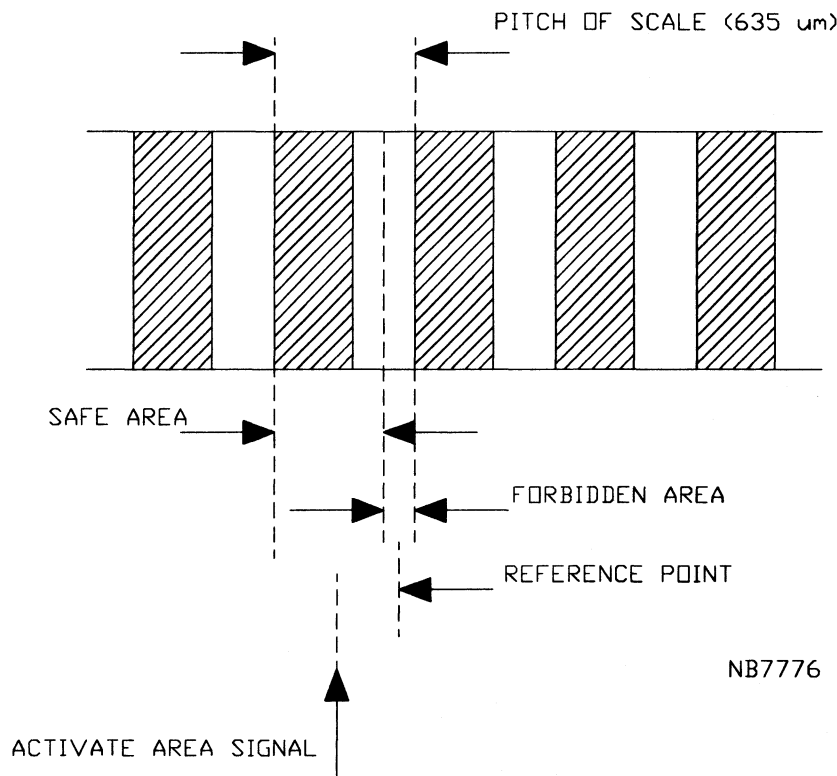
Fig. 6.3.3.-1   Reference point

The switch and cam should be installed in the approximate position required for the zero point to be made. The cam should be left loose and have sufficient play to allow this adjustment. The scales and transducers should be installed and alligned correctly and all components wired before this adjustment can be made.

To obtain the correct position of the area cam, perform the RPF-search for the relevant axes. As soon as the switch hits the cam for the second time the value under actual in the display is reset to zero and starts counting. Observe the value shortly before the display is preset to the offset value. The display should have been counted to a value between 250 and 300 increments. The switch is then in the safe area. Is the value larger or smaller, the cam must be shifted over a distance equal to the displayed distance minus 275 increments. The sign of the results gives the direction of displacement. Repeat the operation till the are cam is in the correct position.

6.3.4.   Connecting the Measuring probe

The switch of the measuring probe is connected to pin 1 or 2 of the socket X5 (AREA-switch input) of the measuring-system, depending on the assignment in machine constant MC840. Usually the AREA-switch input of the spindle drive-system is taken. See figures 6.3.2.-2, 6.3.2.-3, or 6.3.2.-4 for the connection diagram. The measuring probe switch must be normally closed.

6.4. JUMPER SETTINGS ON DRIVE MODULES

Before extracting a pc board from or inserting it into the control rack always be sure that the CNC system is switched off.

Module type 4022 226 . . . . . axis on module	LS/LS 3681 . 1	2	LS/RM 3691 . 1	2	RM/RM 3645 . 1	2	RM 3656 . 1
REFERENCE POINT marker + area area only				W01 a b	W04 a b	W01 a b	W04 a b
AREA SWITCH normally closed normally open				W02 a b	W05 a b	W02 a b	W05 a b
MARKER PHASE not inverted inverted				W03 a b	W06 a b	W03 a b	W06 a b
AXIS TYPE linear or rotary main spindle with thread cutting				W4 a b	W19 a b	W16 a b	W19 a b
ALARM INPUT not used ext. alarm with cable breakage detection				W5 a a b	W20 a b	W21 a b	W20 a b
ADDRESSING jumper only	W06		W07		W14		W14

Notes:

1.-   REFERENCE POINT

If the transducer is not able to give a marker signal, the measuring system can be set to AREA-only. In this case the accuracy depends on the repeatability of the area switch.

2.-   AXIS TYPE

For milling machines this jumper must always be set to linear or rotary.

3.- ADDRESSING

The addressing of the modules 4022 226 3681, 4022 226 3691, 4022 226 3645 and 4022 226 3656 is done respectively by the following jumpers: W6, W7 and W 14, and is **not** depending on the position of the module in the rack.

card number	axes	jumper W6, W7, W14
1	1 - 2	1
2	3 - 4	2
3	5 - 6	3
4	7 - 8	4

4.- READ MODE

Select RT when using a 6, 8, 10 or 12 MHz CPU, or select RD when using a 5 MHz CPU.
For this software version this jumper must always be set to RT.
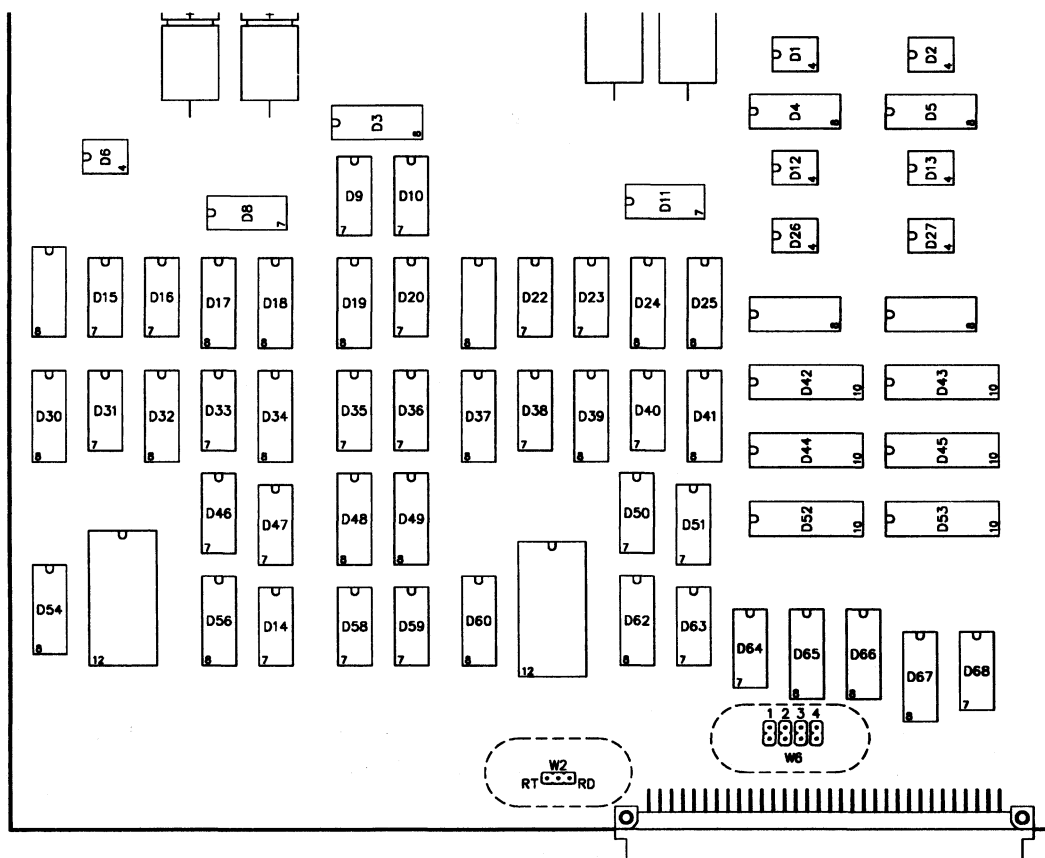
N88178

Fig. 6.4.-1   Jumpers on LS/LS module 4022 226 3681
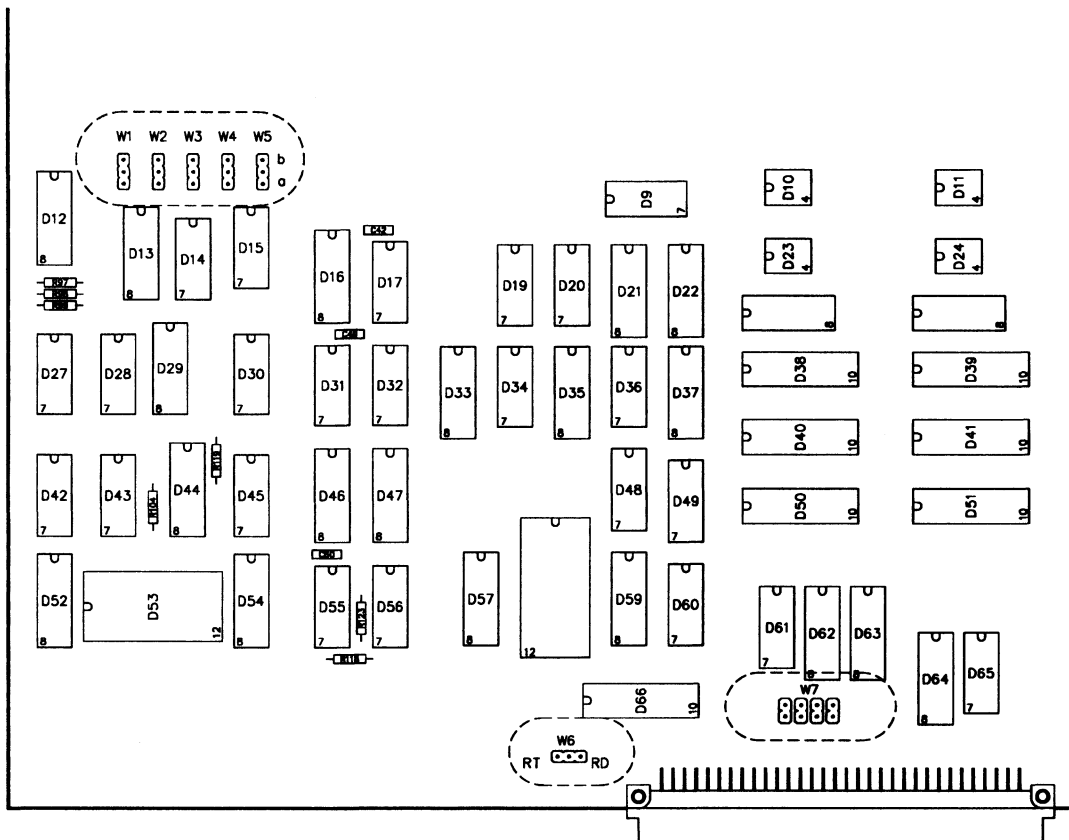
Fig. 6.4.-2   Jumpers on LS/RM module 4022 226 3691
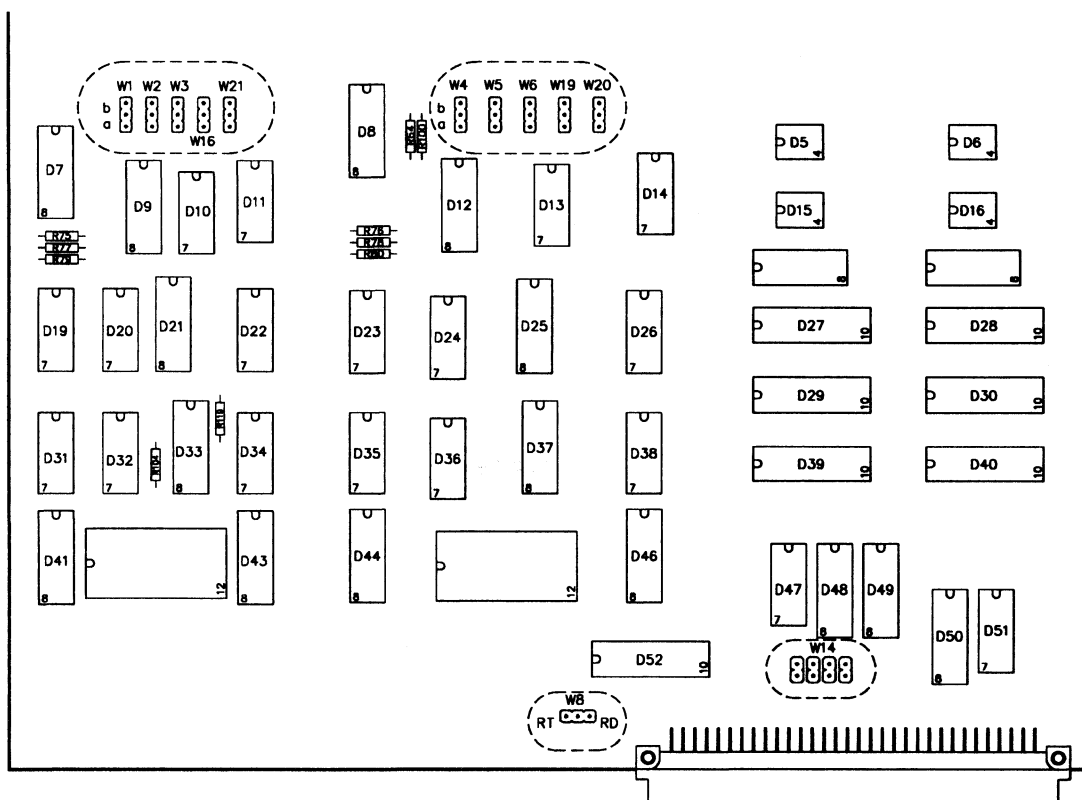
Fig. 6.4.-3   Jumpers on RM/RM module 4022 226 3645
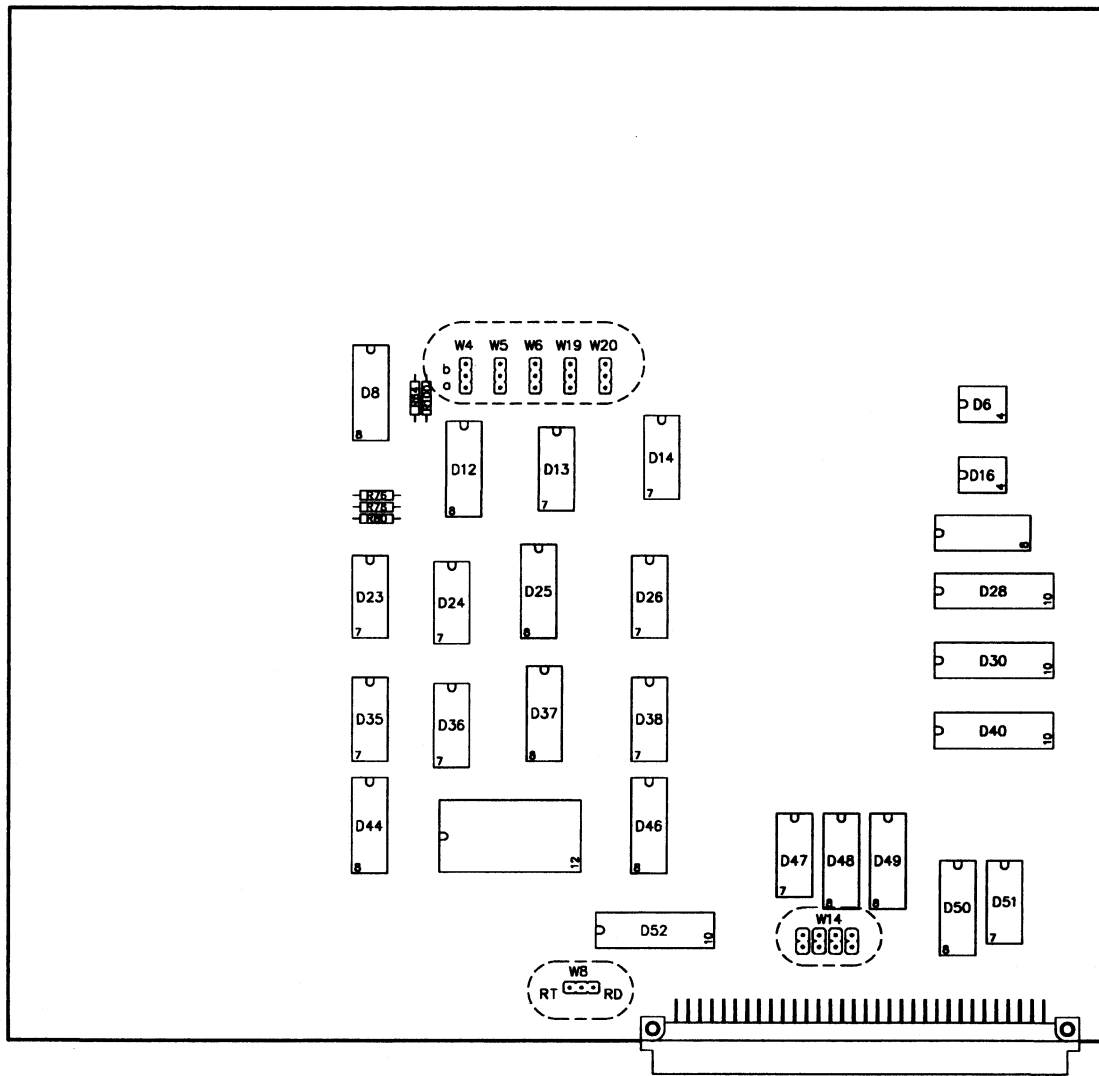
NB8181

Fig. 6.4.-4   Jumpers on RM module 4022 226 3656

7. CONNECTION TO THE INPUT AND OUTPUT MODULES.

7.1. GENERAL

The control rack can obtain up to 2 I/O modules for the 7 slots version, up to 4 I/O modules for the 15 slots version, and a maximum of 6 I/O modules for the 20 slots version.

The number of available boards depends on the application and is determined by the IPLC program.

Each board provides 32 inputs and 32 outputs for digital data.

The input and output signals are also depending on the IPLC program and are therefor not described in this manual. Information about the I/O configuration must be obtained from the author of the IPLC program.

7.2. CHARACTERISTICS OF THE I/O MODULES

7.2.1. General information

Select jumper in position 1 for the first I/O module.
Select jumper in position 2 for the second I/O module, etc.
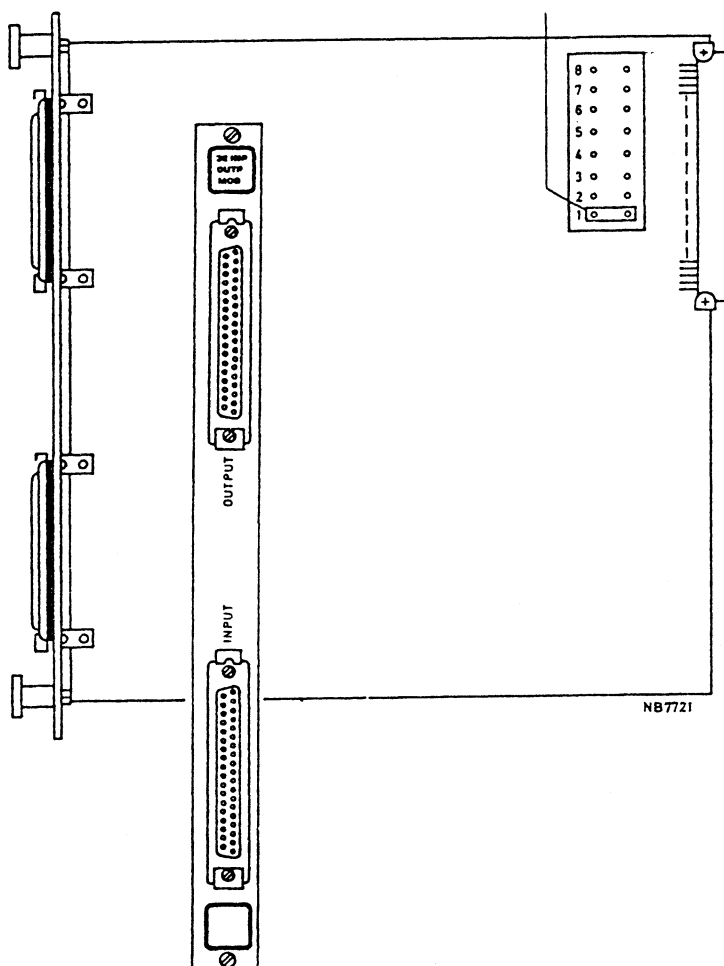
Fig. 7.2.1.-1   Front and side views of 32-Input/Output Module

7.2.2. Input characteristics

The 32 inputs are solid state circuits in accordance with IEC550.
  Their characteristics are:
  Nominal external voltage: +24V (min. 22V/max. 26V)
  Ripple: 4V maximum (between 22V and 26V)
  Logic 1: 22V - 26V
  Logic 0: 0V - 1V
  Load: 15 mA per input
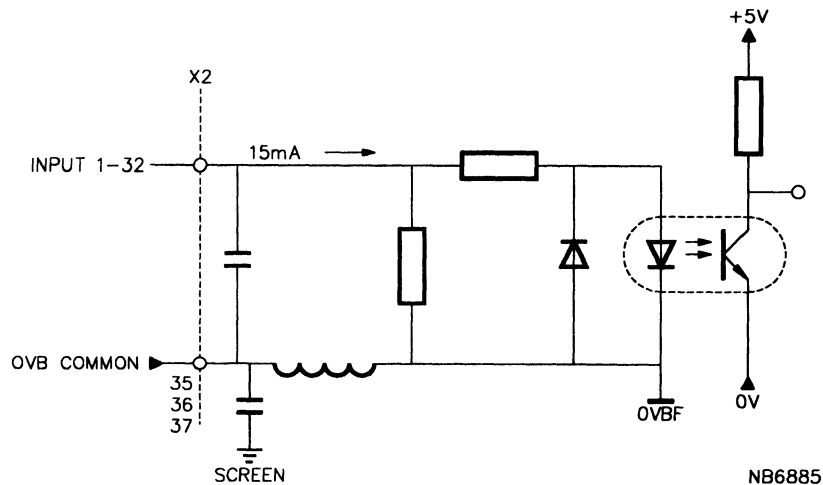
Fig 7.2.2.-1   Input configuration.

7.2.3. Output characteristics

The 32 outputs are solid state circuits in accordance with IEC550.
  Their characteristics are:
  Nominal external voltage: 24V (min. 22V, max. 26V)
  Ripple: 4V maximum (between 22V and 26V)
  Logic 1: 22V - 26V
  Logic 0: 0V - 1V
  Maximum load: 100mA

Outputs are *not protected* against short-circuits.
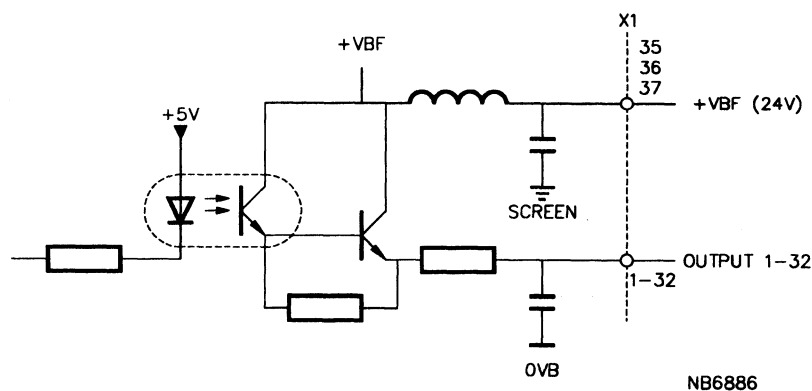
Fig. 7.2.3.-1   Output configuration

Note: The 24V dc supply voltage will be provided by the machine tool manufacturer.

8. CONNECTIONS TO THE GRAPHIC MODULE

The 2-planes graphic module is connected to the control teletext module as indicated in figure 8 -1.

The 8-planes graphic modules are interconnected to each other and to the control teletext module as indicated in figure 8.-2.

To be able to make a hard copy of the screen in graphics mode a printer with a centronics interface can be connected to the socket marked "PRINTER" on the 2- planes graphic module or the 8-planes graphic module-A.

The wiring of a centronic interface cable is shown in figure 8.-3.

Note that the colour information is not available via this printer connection.
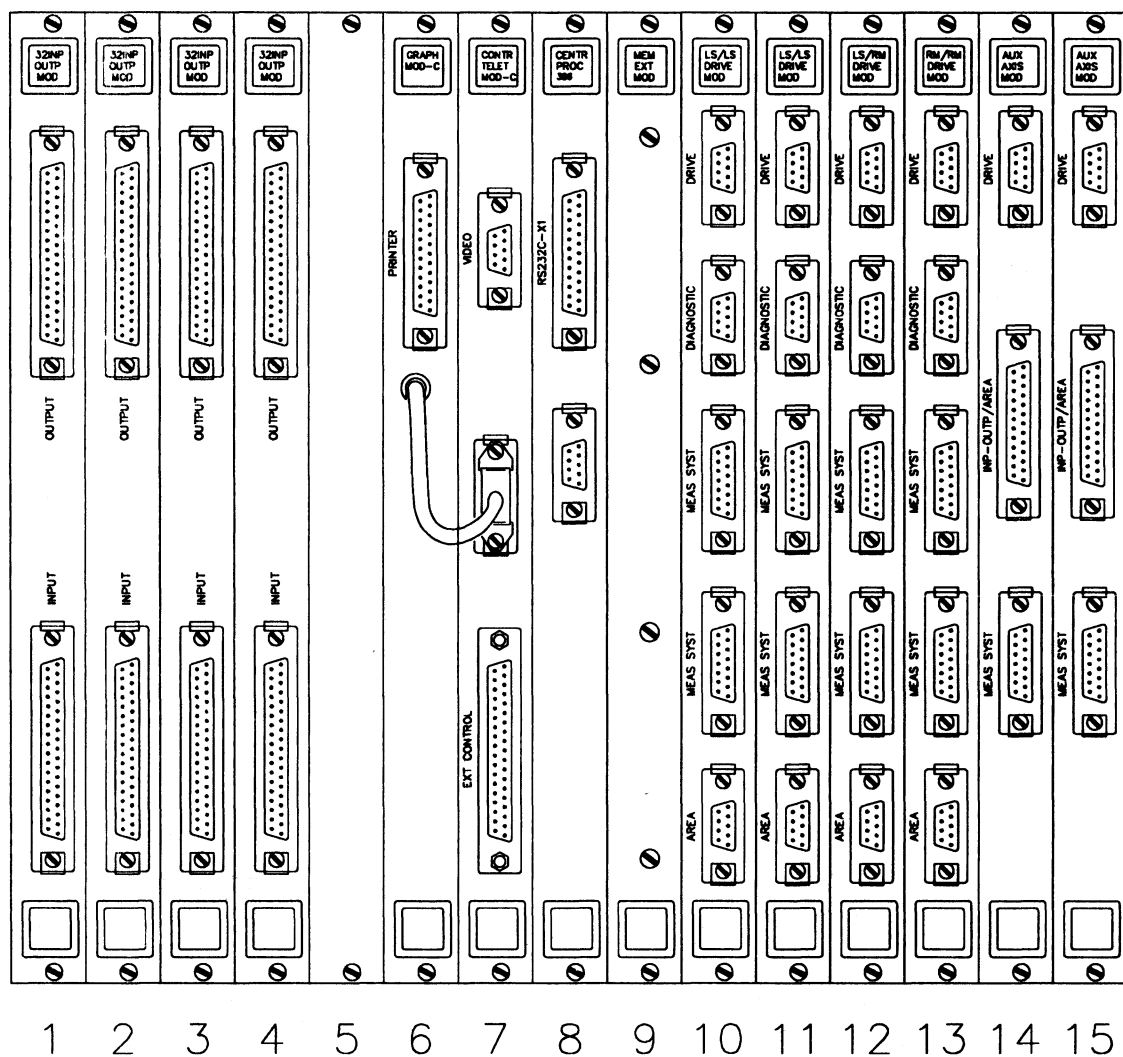
NB8255

Fig. 8.-1. Connection of the 2-planes graphic module
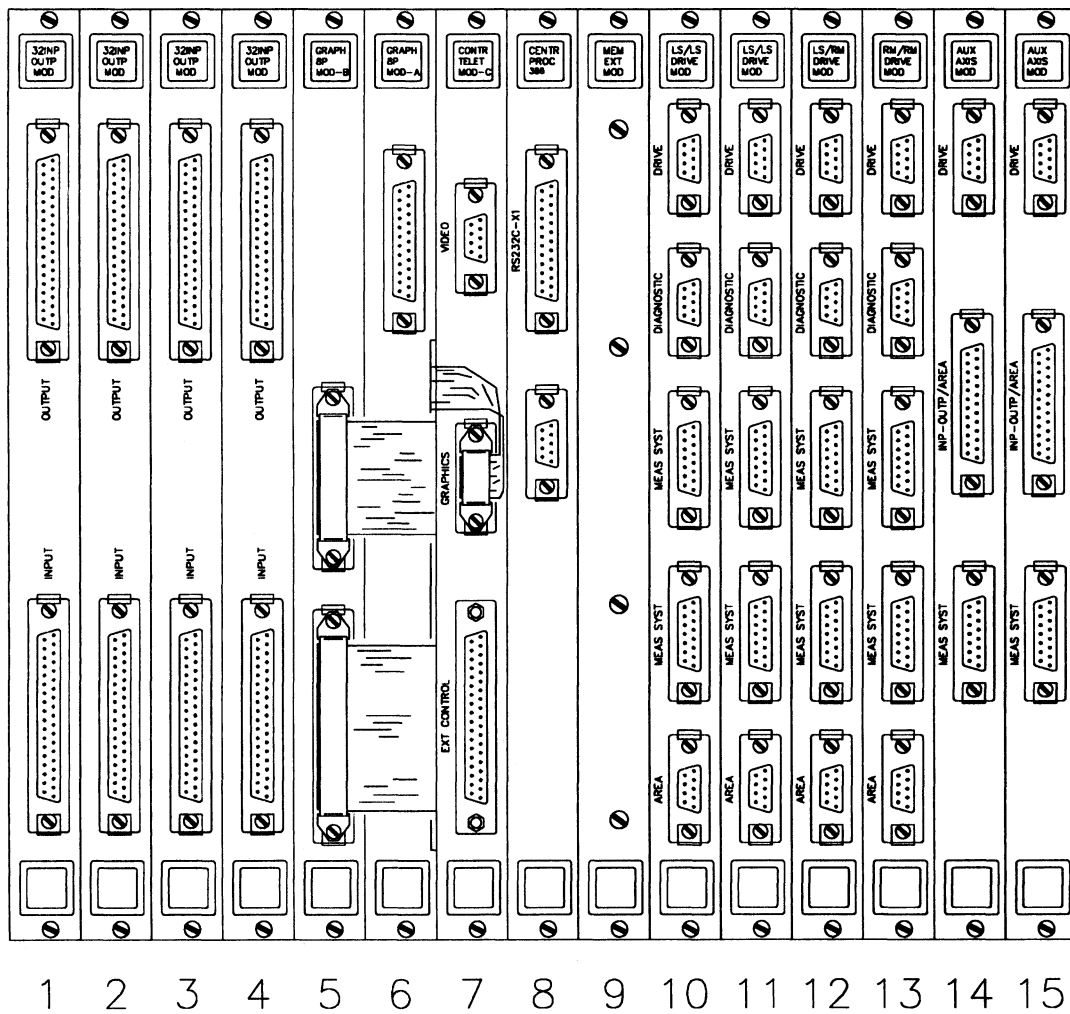
Fig. 8.-2. Connection of the 8-planes graphic modules

CABLE SCREEN

25p male plug
(CNC side)

25p (CNC side)		36p (printer side)
(0V)	12	26+27 (0V)
(0V)	11	24+25 (0V)
(0V)	10	22+23 (0V)
(0V)	9	20+21 (0V)
DATA 8	8	9 DATA 8
DATA 7	7	8 DATA 7
DATA 6	6	7 DATA 6
DATA 5	5	6 DATA 5
BUSY	17	11 BUSY
DATA 4	4	5 DATA 4
ACKNLG	16	10 ACKNLG
DATA 3	3	4 DATA 3
DATA 2	2	3 DATA 2
STROBE	14	1 STROBE
DATA 1	1	2 DATA 1

25p male plug
(CNC side)

36p male plug
(printer side)
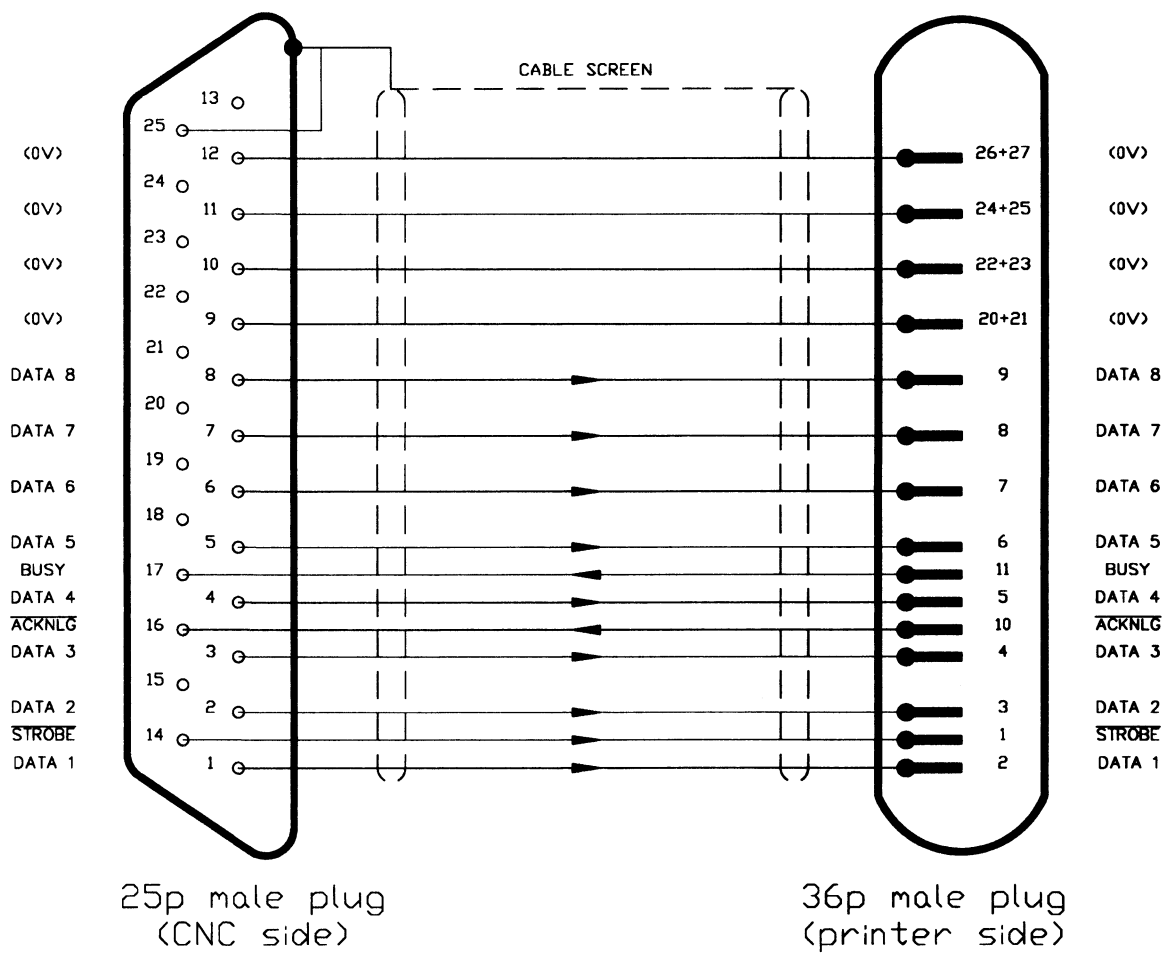
NB8116

Fig. 8.-3. Centronics interface signals

9. CONNECTIONS TO THE CENTRAL PROCESSOR MODULE

The central processor module is provided with a RS232C (X1) and RS449 (X2) interface for data communication. Connector X1 is also provided with a connection for optical communication via a glassfibre cable.

For detailed information on communication protocol and specification of the connections see the interfacing manual.

CONNECTOR X1 (RS232C)

PIN	NAME	INPUT/OUTPUT
2	TxD	I
3	RxD	O
4	RTS	I
5	CTS	O
6	DRS	O
7	0VF	
9	-15V	
10	+15V	
11	+5VP	
15	RxD OPTO	I
18	+12V	
19	0VF	
20	DTR	I
21	+5V	
22	TxD OPTO	O
25	-12V	

CONNECTOR X2 (RS422C) (also named RS449)

PIN	NAME	INPUT/OUTPUT
1	CTS	O
2	DSR	O
3	TxDA	I
4	RxDA	O
5	0VF	
6	RTS	I
7	DTR	I
8	TxDB	I
9	RxDB	O

Fig. 9.-1.   Signals at connectors X1 and X2

9.1   Description of connector X1 for optical devices

The connector X1 can also be used for optical data communication via a glass-fibre cable. On this socket a special plug (Thomas & Betts) can be connected which converts the electrical data into optical data and visa versa.
The connections for this plug are the points 7, 15, 19, 21 and 22.
Because the Thomas and Betts plug has another connection configuration then the RS232 connection on the Central Processor Unit an interconnection cable is needed, for connections see figure 9.1.-1.
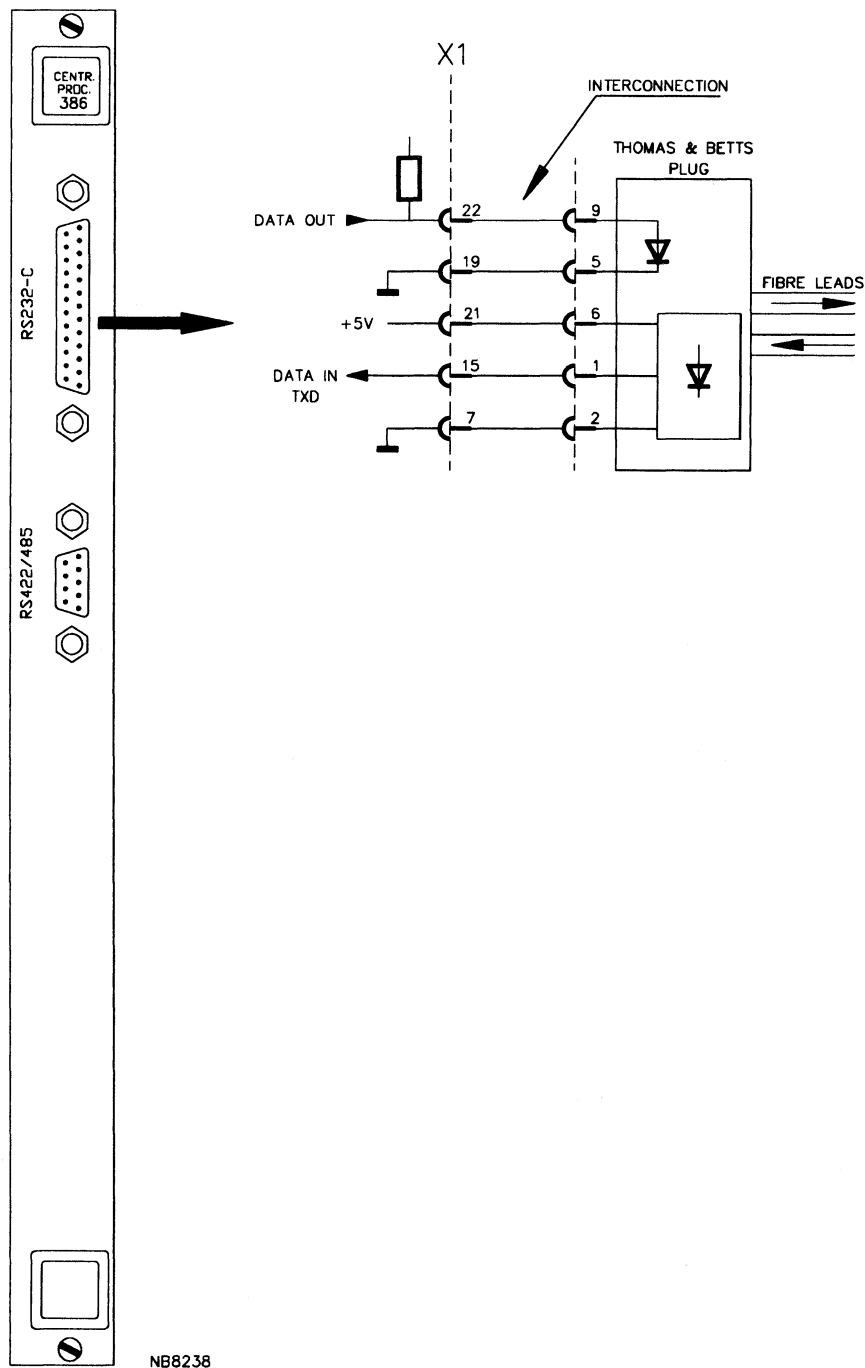
Fig. 9.1.-1 Schematic diagram showing the optical link connection

10. ERROR CODES

If the CNC detects an error it shows an error code in the upper right of the screen display. Depending on the error, the control also takes action to prevent the error from causing damage.
The three most common types of action which is taken by the CNC are:

INTERRUPT:

Interrupts generated during program execution cause the line >INT< to be shown in the screen display. The CNC is forced into Intervention mode - all axis and spindle drive analog outputs are reset (that is, go to zero volt), all digital outputs that anticipate axis or spindle motion are reset, reference-point finding is stopped, search is stopped.

OPEN LOOP:

Open loop action does not result in any indication in the screen display. All analog outputs for the axes and spindle are reset - motion is stopped, but position is maintained. All servo control loops are opened.

R-POINT:

R-point action does not result in any message in the screen display until after the error has been cleared when the line >REF. POINT< is shown to indicate that a reference point search must be performed for all axes. R-point always occurs together with one of the two actions above.

The exact seriousness of each error is give by its "class", which defines what actions are taken by the CNC. These classes are from A (most serious) to H (least serious), as detailed below.
The class of each error is given in the error code list after the error message between round parenthesis ().

Class A: System Errors

These errors are caused by a serious failure of the CNC hardware or its system software.

The CNC halt all axis movement and spindle rotation. It will show an error description in the display as well as an error code.

The CNC can only be cleared by switching it off and re-starting it.

Mostly the error announcement returns after switching on the control again. In this case the local customer support centre is to be contacted.

Class B: Checksum Errors

These errors are caused by a failure in RAM or PROM memory. In some cases the memory failure can also cause errors of the other classes.

Interrupt, open loop and R-point actions will result from checksum errors.

The CNC can be cleared by the Clear button in Manual mode, but the error code will reappear in the screen display.
Some checksum errors disappear after editing the relevant memory.
When the checksum error does not disappear the memory hardware is defective, contact the local Customer Support centre.

Class C: Position-loss Errors

The CNC is unable to find the current values for one or more axis positions.

Interrupt, open loop and R-point actions will result from positionloss errors.

The CNC can be cleared by the Clear button in Manual mode.

Class D: Emergency Stop Errors

These errors are caused by conditions which require that the analog output voltages go to zero immediately.

Open loop and R-point actions will result from emergency stop errors.
There is no loss of position.

The CNC can be cleared by the Clear button in Manual mode.

Class E: Stop-execution Errors

These errors are caused by conditions that require that the executions of the part program is halted.

Interrupt action will result from stop-execution errors.

The CNC can be cleared by the Clear button in Manual mode.

Class F: Programming Errors

These errors are generally caused by programming errors that are detected before the block containing the error is executed.

No positive action will result from a programming error, but in Auto mode the execution of the program will be stopped at the end of the block preceding the block with the error.

The CNC can be cleared by the Clear button in Manual or the Teach-in mode.

Class G: Operating Errors

These errors are caused by the operator making an incorrect keyboard input while editing or while in Teach-in or Manual Data Input modes.

No positive action will result from an operating error, but responses to operator keyboard inputs will be stopped.

The CNC can be cleared by the Clear button in any operation.

Class H: Warning Errors

Certain error conditions can be detected in an early stage (for instance soiling of the measuring system or memory overflow) and are displayed as warnings. The operator is given the opportunity to remove the warning condition before it results in a serious error.

No CNC actions result from a Warning error.

Remark:

If more than two errors are detected at the same time only the two most serious errors will be given priority and are displayed.

AXIS ERRORS:

Note that in the axis error codes below n can be replaced by X, Y or Z, etc. for the axis in question.

n01: LINEAR MEASURING SYSTEM PRE-ALARM (H)

Measuring system voltage is reduced by deposit of dirt at the measuring system. The measurement is still correct.

Solution: 1) Cleaning the measuring scale
2) Cleaning the measuring window of the transducer

n02: MEASURING SYSTEM ALARM (C)

Measurement is not reliable because the measurement system voltage is too small.

Solution: 1) Cleaning the measuring scale
2) Replace the measuring scale (if the scale is damaged)
3) Replace transducer

n03: MEAS. SYSTEM POWER SUPPLY FAILURE (C)

Measuring system power supply failure.

Solution: 1) Check power supply unit
2) Check the regarding drivecard

n04: MAX. FOLLOWING DISTANCE TOO LARGE (C)

The following distance is more than 110% of the maximum following distance, defined in machine constants MC215 or MC217 for the first axis and the relevant machine constants for the other axes.

Possible cause: Defect in the servo system (amplifier, motor or measuring system)

n05: SOFTWARE LIMIT SWITCH TRIPPED (E)

Axis passed the software limit switch.

Solution: 1) Clear error message
2) Jog the axes from the end switch with the softkeys

n06: MOTION FASTER THAN RAPID (E)

The calculated traverse speed is more than the maximum allowed traverse speed for this axis (defined in machine constant MC355 for the first axis).

Solution: Clear error message and coprrect the program

n07: DRIFTING OUTSIDE NO-MOTION WINDOW (D)

The axis position is shifted from the "commanded position" over a distance larger than specified in the no-motion window because of external influences, such as drift in the servo system.

Solution: 1) Check the servo system
2) With very heavy machining, it can be necessary to expand the no-motion window

n08: MOTION FASTER THAN MAX. FEEDRATE (E)

The programmed feedrate is more than the maximum feedrate (regarding machine constant MC356 for the first axis).

Solution: -   Clear the error message
              -   Change the program

n09:   DYN. FOLLOWING DISTANCE ERROR TOO LARGE (C)

The following distance deviates too much from the standard course.
The cause can be a failure in the servo system (amplifier, motor or measuring system) or a mechanical fault.

n10:   LINEAR CORRECTION VALUE > 200 INC. (C)

There are more than 200 increments for correction values in the machine constants.

n11:   LINEAR CORRECTION DISTANCE < 100 INC. (C)

The distance between two correction points is less than 100 increments.

n12:   CYC. CORRECTION VALUE > 200 INC. (C)

There are more than 200 increments for cyclic correction value in the machine constants for this axis.

n13:   CYC. CORRECTION DISTANCE < 100 INC. (C)

The distance between two correction points is smaller than 100 increments.

DATA IN/OUT ERRORS:

D01:   READING TM/MC DURING EXECUTION (G)

The tool memory or the machine constants memory cannot be accessed during an execution of a part-program, as this can lead to hazardous situations. Data I/O is aborted immediately.

D02:   NO BLOCK NUMBER READ (G)

The block number is missing on the data carrier.
(Carriage Return encounter without reading a N-address.)

D04:   CNC CANNOT STORE DATA (INTERNAL ERROR) (G)

During data entry the CNC detected an error in the file while storing it. This fault is not caused by an error on the data carrier or in the data connection but can be caused for example by a short voltage drop in the power supply of the CNC.

Solution: Try to delete the block in which the error occurs or delete the complete program and reload the program.

D05:   TOO MANY CHARACTERS IN A BLOCK (G)

There are more then 128 characters in a block.

Solution: Subdivide the block into more blocks

D06:   BLOCKNUMBER ALREADY EXISTS (G)

The blocknumber is already existing in the partprogram.
With MC 796 the blocknumber check can be switched off to allow multiple blocknumbers in a program (MC 796=1).

Solution: Correct the program.

D07:   PREWARNING MEMORY OVERFLOW (H)

Warning: Memory overflow imminent.

Solution: 1) Delete another not used existing program so that   memory space will be available for the transmitting program, after that clear the error message (D07),and the transmission will be continued
              2) If many programs must reside in memory it can be possible to enlarge the memory capacity, for that consult the machine tool builder.

D08:   PROGRAM MEMORY OVERFLOW (G)

Program memory overflow (there is no space enough in the memory to store the transmitted program).
This will cause an immediate abortion of the transmission.

Solution: See error D07.

D10:   PROGRAM NUMBER ALREADY EXISTS (G)

There is already a program with this identification number in the CNC-memory. This will cause an immediate abortion of the transmission.

Solution: 1) Delete or rename the resident program and repeat data entry.
              2) Rename the program on the data carrier.

D20:   UNKNOWN G-FUNCTION (G)

Read in G-function does not exist for this software version.

D25:   NR. EXT. PROGRAM CALLS > MC43 (G)

Number of external programs calls is more than specified by machine constant MC43.

D26:   EXT. CALL NR. ALREADY IN PE MEMORY (G)

External call of program number that is already in the external prgram call memory.

D27:   TOOL IS SPARE TOOL (G)

Tool number is already allocated as a spare tool.
A spare tool has already been allocated to the tool.
Only free spare tools can be allocated to free tools.

D29:   EXT. PROGR. CALL WITHOUT PROGRAM NR. (G)

External program call without allocated part program number.

D31:   ILLEGAL CHARACTER (G)

An illegal character is programmed (the character is not in the character set list as given in the data communication manual).

D32:   PARITY ERROR (G)

During data in a parity error is detected.

Possible cause: 1) wrong setting of machine constant (MC771)
                  2) communication failure
                  3) faulty information on data carrier

D33:   WORD LENGTH TOO BIG (G)

During data in an address is detected with too many characters.
For example: N10 G1234567890

D34:   DATA IN/OUT IS INTERRUPTED (G)

During data in/out the session is canceled with cancel-softkey.
This will cause an immediate abortion of the transmission.

D35:   PROGRAM IS SMALLER THEN 9000 (G)

During data in with protocol A, a program number smaller than 9000 is detected.

D36:   REQUESTED MEMORY SELECT NOT FOUND (G)

Requested memory code is not on the data carrier.

End Of Tape is detected and the relevant memory identifier is not encountered.

Possible cause: 1) Memory identifier has been omitted.
                  2) Faulty memory in CNC has been opened.

D37:   ILLEGAL DELIMITER AFTER ADDRESS (G)

Address has an illegal character in its data, for example:
N1G0   X100%

D38:   ADDRESS WITHOUT DATA (G)

Address without data detected, for example:
N1G0   X   Y100

D39:   BLOCK NUMBER >=9000 (G)

Blocknumber begins with a program identification number
(a blocknumber must be less then 9000).

D40:   EXPRESSIONS IN NON PARAMETER MODE (G)

Algebraic expressions are only allowed in parameter mode.

D41:   DATA IN/OUT PERIPHERAL NOT READY (G)

The DTR-line of the peripheral is not active, this causes an immediate abortion of the transmission.

Solution: 1)   Connect peripheral
                 2)   Switch on peripheral
                 3)   If appropriate change machine constant MC773 into 3

D42:   BAUDRATE ERROR (G)

Baudrate from peripheral or CNC is wrong or communication connection is bad.

D43:   NON-VALID MEMORY SELECT FOUND (G)

Memory identifier is unknown for this software version (address must be PM, CM, TM etc.).
This causes an immediate abortion of the transmission.

D44:   NON-EXISTING ADDRESS (G)

Address is unknown for this software version.

D45:   SIGN NOT ALLOWED IN THIS ADDRESS (G)

Minus sign (-) is not allowed in this address.

D46:   INVALID OR MULTIPLE EQUALS IN WORD (G)

Block contains multiple "=" signs for example N1   E1==10
or
block contains an address in which an equal sign is not allowed for example G=E10.

D47:   REQUESTED PROGRAM NR. NOT FOUND (G)

The requested program is not found, probably the program is not on the data carrier.

D48:   PROGR. IS INCH, NC IS METRIC OR VV. (H)

Warning: During read in inch/metric or metric /inch conversion takes place.

D49:   PROGRAM LOCKED (H)

Attempt to perform data I/O with a locked program.

Possible cause: 1) The requested program is locked by the lock attribute (softkey).
                2) The programs are locked by an external switch.

D50:   MEMORY IN USE, CAN NOT BE CHANGED (H)

Memory is currently in use by a function other than data I/O, this will cause an immediate abortion of the transmission.

Solution: Wait a moment and try again.

D51:   NUMBER OF PROGRAMS > MC85 (H)

The maximum number of programs is reached, entering new programs in ID-directory is not possible.
Machine constant MC85 contains the maximum number of programs.

Solution: 1) Delete a not used program from memory.
                    If necessary change machine constant MC85 (remark: memory will be cleared!)

D52:   TEMPERATURE MEMORY OVERFLOW (H)

Memory capacity of the temperature memory is exceeded.

D53:   NO TEMPERATURE DATA PROGRAMMED (H)

The temperature-word is not found in the program.

D60:   T-NUMBER ALREADY EXISTS (H)

The T-word is already existing in another block.

D61:   T-NUMBER < 3 DECADEN (H)

For FMS tool memory the tool data must contain spare tool information (T>=100).

D62:   OVERSIZED TOOL NEXT TO NORMAL TOOL (H)

The oversized tool is stored in a magazin position next to an occupied position.

Solution: 1) Choose another tool position for the oversized tool
                2) Remove the tool in the position next to the over-sized tool

D63:   NO EMPTY PLACE NEXT OVERSIZED TOOL (H)

A tool is stored in the magazin next to a oversized tool position.

Solution: 1) Choose another toolposition
                2) Remove the oversized tool

D80:   DNC: NOT SYNCHRONIZED (G)

DNC is out of synchronization.
When this message is reported, there is usually an error in the computer connected to the CNC; either hardware or the protocol is not correct.

D81:   DNC: BREAK DETECTED IN CONNECTION (G)

The CNC detects that the DTR connection to the computer is not active. The cause is usually a bad or disconnected cable or the connected computer is not switched on.

Solution: 1) Switch on computer
                2) This error does not occur when MC783 is 1
                   (MC determines whether or not the CNC is checking the signals RTS and DTR.
                   When MC=0, DTR is constantly checking for active level and the CNC only transmits
                   when RTS is active. When MC=0 both DTR and RTS are ignored.)

D82:   DNC: BAUDRATE ERROR (G)

Baudrate of CNC or computer is wrong.

Possible cause: 1) Bad communication
                2) Baudrate not correctly set on CNC or computer
                3) Parity error
                4) The DNC connection is subject to interferenc i.e. the connection is too long
                   or runs past an interference source

Possible solutions: 1) Set the baudrate correctl
                   2) Check the connection, particularly the handshaking signals, use an oscillograph, if
                     applicable

D84:   DNC: TIME OUT OCCURRED (H)

Time-out occurred in LSV/2 protocol.

Possible cause: 1) The DNC connection is subject to interference, i.e. the connection is too long or bad
                2) Wrong setting machine constant MC795 (MC795 specifies a delay time between
                   "reading" and "writing")

Remark: this error is usually related to other errors (like D82 and D85)

D85:   DNC: BLOCK CHECK CODE FAILED (H)

The CNC does not receive the correct checksum character, even after repeats.

Possible cause: 1) The DNC connection is subject to interference
                2) An error in the connected computer

D86:   DNC: DNC COMPUTER ABORTS TRANSFER (H)

The transmission has been aborted by the computer using a "T_BD"
message.

Possible cause: Format of the useful data is not identical to the format of the indicated memory

D100:   DNC: COMPUTER CANNOT STORE PROGRAM (H)

The DNC computer cannot store the transmitted program.

Possible cause: No write permission because file is in use, etc.

D101   DNC: PROGR. NOT FOUND ON MAIN COMP.(H)

The computer can not find the file requested by the CNC.

Possible causes: 1) Incorrect program requested by control.
                    2) Desired program not in indicated program inventory of master computer

D102:   DNC: COMPUTER CANNOT SEND PROGRAM (H)

The computer is not able to send the requested file although it does exist.

Possible cause: 1) The requested program is in use.

D103:   DNC: ID IN USE ON COMPUTER (H)

The computer cannot store the transmitted file using the given name, because this name is already existing.

D120:   MISSING "OPERATOR" (G)

The required operator for this expression is missing.

D121:   MISSING "OPERAND" (G)

The required operand for this expression is missing.

D122:   MISSING LEFT PARENTHESIS (G)

The left parenthesis in this expression is missing.

D123:   MISSING RIGHT PARENTHESIS (G)

The right parenthesis in this expression is missing.

D124:   "SYNTAX" ERROR IN EXPRESSION (G)

The syntax of the expression is not right.

D125:   E-PARAMETER IS CALCULATOR MODE (G)

In the calculator mode an E-parameter is used.

D126:   TOO MANY PARENTHESES IN EXPRESSION (G)

There are too many parentheses in this expression.

D127:   E-PARAMETER DIVIDE BY ZERO (G)

Value is infinitive because an E-parameter is divided by zero.

D128:   CONVERSION DEGREE-RAD. NOT ALLOWED (G)

Conversion from degrees to radials or vice versa is not allowed.

D129:   INTERMEDIATE RESULTS OUT OF RANGE (G)

Intermediate result is too big or too small.

D130:   EXPONENT SYNTAX ERROR (G)

This exponent is not allowed.

D131:   E-PARAMETER WITHOUT VALUE (G)

There is no value assigned to the used E-parameter.

D132:   CALCULATED VALUE TOO BIG (G)

The result of the calculation is too big.

D133:   CALCULATED VALUE TOO SMALL (G)

The result of the calculation is too small.

D134:   VALUE > MAXIMUM (G)

Programmed value for an address is more than the specified maximum value e.g. maximum is 100 and in the program is:
X101, X=101, X=100+1.

Solution: program the correct value

D135:   VALUE < MINIMUM (G)

Programmed value for an address is smaller then the specified minimum value.

D136:   "SYNTAX" ERROR (G)

In the program a syntax error is detected.

Possible cause: 1) In an address with an equal sign (=) not a digit, but an arbitrary character is pro-
                   grammed.
                2) In an address with an equal sign (=) more than three characters (maximum is 3) are
                   programmed.
                   Solution: remove the superfluous digits
                3) It is not allowed to use 4,5,6,7,8 or 10 for sensing temperature compensation memo-
                   ry.
                   Solution: program the correct sense values (1, 2, 3, 11, 12 or 13).
                4) In the toolspare there is no 'T' programmed after the equal sign (=).
                   Solution: change program, toolspare format   must be Txx = Tyy.

D137:   CHANGE NOT ALLOWED (G)

                1) A spare tool is allocated to T0: T0=TX
                   Solution: do not allocate to T0
                2) A spare tool allocated to a tool outside the tool changer or a spare tool outside the
                   tool changer allocated to a tool inside the tool changer.
                   For example: Number of tools=20 number of tool positions in tool magazin=10:
                   T9=T11 or T11=T8
                3) A not activated machine constant is entered.

D138:   G-FUNCTION NOT PERMITTED (G)

The programmed G-function in combination with the active modal G- functions is not permitted.

D139:   TOO MANY CHARACTERS IN A WORD (G)

An address has too many characters before or after the decimal point.

D140: DECIMAL POINT NOT ALLOWED (G)

In the address a decimal point is not allowed.

D141: ADDRESS NOT PERMITTED (G)

1) In a block a non-permitted address is programmed.

2) In tool spare memory Txx is programmed without allocation, e.g. Txx is programmed. Txx=Tyy must be programmed

3) Programmed spare tool is higher than the number of tools for example in machine constants is set: Number of tools=20 and T22 is programmed.

D142: COMMENT PARENTHESES NOT CLOSED (G)

During syntax check is detected that the comment parentheses are not closed.

Remark: With data input/output it is allowed to give comment between parentheses stretching over more blocks, then the error will be detected at the end of the program.

D143: DOUBLE ADDRESSES IN BLOCK (G)

1) In a block in the Part program or macro there are more than four P-words pr-grammed.

2) In a block the same address is used twice or more.

D150: MC770-MC776 WRONG, DIO USES DEFAULT (G)

When during data input/output one or more machine constants from the group MC770-MC777 is wrong, the default setting will be used.
Reading will be started after the error is cleared.

Solution: Change machine constant code MC770-MC777

D151: PROGRAM DOESN'T MATCH %ID (G)

Reading a program or a macro via the B-protocol, the first block of the program or macro must have the same number as the identifier.

For example:

| Correct: | %PM9000 | Not correct: | %PM9000 |
| | N9000 | | N9002 |

D152: PROGRAM/MACRO IS BEING EDITED (G)

While editing a program or macro it cannot be sent with DATA IO or DNC.

Solution: Leave edit mode to transfer the program.

D153: SELECTED PROGRAM NOT FOUND (H)

The requested program is not in the CNC memory.

D154: MEMORY LOCK (G)

The memory is locked with the external lock switch to protect the program against unauthorised editing.

Solution: switch off the memory lock

AUXILIARY AXIS ERRORS

Note that the auxiliary axes error codes indicated with n can be replaced by relevant auxilary axis identification.

Hn01:   LINEAR MEASURING SYSTEM PRE-ALARM (H)

Measuring system voltage is reduced by deposit of dirt at the measuring system. The measurement is still correct.

Solution: 1) Cleaning the measuring scale
                   2) Cleaning the measuring window of the transducer

Hn02:   MEASURING SYSTEM ALARM (C)

Measurement is not reliable because the measurement system voltage is too small.

Solution: 1) Cleaning the measuring scale
                   2) Replace the measuring scale (if the scale is damaged)
                   3) Replace transducer

Hn03:   MEAS. SYSTEM POWER SUPPLY FAILURE (C)

Measuring system power supply failure.

Solution: 1) Check power supply unit
                   2) Check the regarding drivecard

Hn04:   MAX. FOLLOWING DISTANCE TOO LARGE (C)

The following distance is more than 110% of the maximum following distance, defined in machine constants MC215 or MC217 for the first axis and the relevant machine constants for the other axes.

Possible cause: Defect in the servo system (amplifier, motor or measuring system)

Hn05:   SOFTWARE LIMIT SWITCH TRIPPED (E)

Axis passed the software limit switch.

Solution: 1) Clear error message
                   2) Jog the axes from the limit switch with the softkeys

Hn06:   MOTION FASTER THAN RAPID (E)

The calculated traverse speed is more than the maximum allowed traverse speed for this axis (defined in machine constant MC355 for the first axis).

Solution: Clear error message and coprrect the program

Hn07:   DRIFTING OUTSIDE NO-MOTION WINDOW (D)

The axis position is shifted from the "commanded position" over a distance larger than specified in the no-motion window because of external influences, such as drift in the servo system.

Solution: 1) Check the servo system
                   2) With very heavy machining, it can be necessary to expand the no-motion window

Hn11:   SERVO IN OPEN LOOP (B)

The auxiliary axis servo system went in open loop because of a measuring system failure.

Hn40:   DESTINATION IS NO INDEX (D)

The programmed destination is not an index position.

Hn42:   INDEX SEARCH OVERFLOW (D)

At the given feed the index is too close to another index.

Hn91:   AUXILIARY AXIS: OPENLOOP (D)

The following error codes (H15 - H31) are not related to a specific auxiliary axis but to auxiliary axes in general.

H15:   AXIS NOT RESPONDING (E)

The auxiliary axis is not responding, the possible cause is that the regarding auxiliary axis module is defective.

H17:   ARBITER CHANGE BEFORE INPOD (H)

The IPLC commands the auxiliary axis too early, the cause is usually an error in the IPLC program.

H30:   ILLEGAL JOGMODE (G)

The selected jog mode is not allowed for this auxiliary axis.

H31:   USER JOG/RPF INHIBIT (G)

The IPLC controls the auxiliary axis, and the user is not allowed to jog the axis or perform Reference Point Finding.

INTERFACE ERRORS:

I01:   INTERFACE NOT INITIALIZED (D)

Interface is not initialized. The interface is not responding to the control signals.

I03:   IPLC NOT INITIALIZED (D)

IPLC program not initialized. An attempt was made to do RPF or JOG the axes after power on and before the IPLC program has been initialized.

I04:   FAST I/O'S OUT OF RANGE (D)

I05:   EMERGENCY STOP FROM MACHINE TOOL (D)

Emergency stop signal was generated by the machine tool.

I11:   WARNING: TOOL LIFE EXCEEDED (H)

During machining the Tool Life of the active tool is exceeded. The tool is not replaced immediately, but is used till the next tool change. If a spare tool is allocated, it is used at a coming tool change for the worn out tool.

I12:   TOOL LIFE EXCEEDED (E)

Tool change for a tool with exceeded tool life.
Tool life of the spare tool is exceeded.

I13:   WARNING: 1ST POWER LEVEL EXCEEDED (H)

During machining the 1st power level of the active tool is exceeded.
The tool is not replaced immediately, but is used till the next tool change. If a spare tool is allocated, it is used at the coming tool change for the worn out tool.

I14:   TOOLCHANGE AND 1ST POWER LEVEL (E)

Tool change for a tool with 1st power level exceeded.
1st power level of spare tool is exceeded.

I15:   WARNING: 2ND POWER LEVEL EXCEEDED (E)

Warning: 2nd power level exceeded.

I16:   MEASURING PROBE NOT READY (E)

The measuring probe is not activated or the infra red path of a relevant measuring probe is obstructed.

EXTERNAL PROGRAM CALL ERRORS:

I20:   EXT. PROGR. CALL IS STILL WAITING (H)

External program call is not executed. The ext. progr. call buffer is full. There is already a call for the program offered.

I21:   START EXT. PROGRAM WITHOUT PROGR. NUMBER (F)

There is an external program call and there is no program number in the buffer.

I22:   CHANGE EXT. PROGRAM IN EXECUTION (F)

There is an external program call for a program what is being edited or under execution at this moment.

I23:   NO PROGRAM NR. IN EXT. PROGRAM CALL (F)

In the parameter for the external program call is no programnumber.

I24:   EXT. PROGRAM NR. IS NOT IN MEMORY (F)

The external called program is not in the cnc memory.

I34:   TOOLHOLDER OPEN (E)

Machining cannot start because the toolholder is open.

I45:   NO NEXT BLOCK PERMITTED FROM IPLC (H)

The start button is pushed while the "next block permitted" signal is not present.

IPLC ERRORS:

I200:   IPLC: NO PROGRAM PRESENT IN EPROM (D)

There is no program present in EPROM (the IPLC memory can be selected with machine constant MC1086, 0 = program in EPROM).

I201:   IPLC: NO PROGRAM PRESENT IN RAM (D)

There is no program present in RAM (the IPLC memory can be selected with machine constant MC1086, 1 = program in RAM).

I202:   IPLC: PROGRAM MODULE NOT PRESENT (D)

This error is caused by a failure in the IPLC program, the called program module is not present.

I203:   IPLC: RECURSIVE CALL OF PR. MODULE (D)

This error is caused by a failure in the IPLC program, the program module is calling itself.

I206:   IPLC: CYCLE TIME OVERFLOW (D)

The execution time of the IPLC program is longer then the cycle time (50 ms).

Remark: The execution time of the IPLC program must always be shorter then the cycle time.

MEMORY ERRORS:

M01:   MC MEMORY CHECKSUM ERROR (B)

Machine constant memory is not reliable. Faulty Ram location is not definable.

M02:   TOOLOFFSET MEMORY CHECKSUM ERROR (C)

Tool memory is not reliable. Faulty Ram location is not definable.

M03:   MC SHADOW MEMORY CHECKSUM ERROR (B)

The copy of the machine constant memory is not reliable. Faulty Ram location is not definable.

M20:   SYSTEM PROM CHECKSUM ERROR (B)

System PROM cannot be fully tested, because the necessary information is incorrect. Any of the PROM's can be defective.
The error can be located with the test in the Diagnose Menu (service only).

M50:   CHECKSUM IPLC PROM (11-12) (B)

The IPLC PROM's (placed at position 11 and 12) are not reliable.
This error can only occur if machine constant MC15 indicates that the IPLC is on, and machine constant MC1086 indicates that IPLC memory is in PROM's.

M60:   PM/MM MEMORY CHECKSUM ERROR (C)

Defective RAM in the CNC.

The relevant RAM can be:

         1)   on the CPU-module (placed at position 15-22)
         2)   on the memory extension module

Remark: In the Diagnose menu's are facilities to locate this error.

OPERATION ERRORS:

O01:   EDITING AN ACTIVE PM/MM (G)

Editing of a partprogram or macro which is under execution is not allowed.

O02:   NO BLOCK NUMBER READ (G)

The block number is not entered.

O03:   BLOCKNUMBER OR ADDRESS NOT FOUND (G)

The required address or blocknumber has not been found in the memory.

O04:   READ ERROR IN ACTIVE PROGRAM (G)

Error during data reading from CNC memory, the required file cannot be used.

Solution: Delete file.

O05:   TOO MANY CHARACTERS IN A BLOCK (G)

There are more then 128 characters in a block.

Solution: subdivide the block into more blocks

O06:   BLOCKNUMBER ALREADY EXISTS (G)

The entered blocknumber is already existing in the partprogram.

Remark: Multiple blocknumbers are allowed when machine constant MC796 is set to 1.

O07:   PREWARNING MEMORY OVERFLOW (H)

Warning: Memory overflow imminent.

Solution: 1) Delete another not used existing program so that   memory space will be available for
                      the program to be entered, after that clear the error message (O07),and program en-
                      tering can be continued
                 2) If many programs must reside in memory it can be possible to enlarge the memory ca-
                      pacity, for that consult the machine tool builder.

O08:   PROGRAM MEMORY OVERFLOW (G)

Program memory overflow (there is no space enough in the memory to store the entered program).

Solution: See error code O07

O10:   PROGRAM NUMBER ALREADY EXISTS (G)

There is already a program with this identification number in the CNC-memory.

O11:   PROGRAM-/BLOCKNUMBER NOT SEARCHED (G)

Before starting a program a program/block search must be done to activate a program for execution.

Remark: Program/block search needs only to be executed once, after this the program remains activated
for execution.

O12:   NO PROGRAM NUMBER ENTERED (G)

This error occurs when a block is being stored without specifying a program number first.

Solution: program correct program number (program identification number must be 9000 or more)

O14:   READ ERROR NEXT BLOCK (G)

During reading a block the memory manager detected an error.

Program connot be used any longer and must be deleted from memory.

O17:   SEARCH, PLAYBACK: M30 FOUND (G)

During Teach-in, Playback or Search mode a M30 is encountered.

Solution: Delete the M30 until the program is finished.

O18:   NOT ENOUGH RAM-MEMORY AVAILABLE (G)

There is not enough RAM-memory for the requested memory configuration.

Solution: Change the configuration with the regarding machine constants

O20: UNKNOWN G-FUNCTION (G)

Entered G-function is not existing for this software version.

O21: TEACH-IN: START WITH ACTIVE G11/P (G)

Point definition or G11 function is active at TEACH-IN.

O22: SAME P NUMBER IN TOOLMEMORY (G)

In the toolmemory more than one tool is allocated at the same position. Every tool has an unique position.

O23: MC DATA OUT OF RANGE (C)

Machine constant value is too high or too low.

O24: PROBE COLLISION OUTSIDE CYCLE/M3 (D)

Measuring probe collision during a movement outside measuring cycle or during returning from measuring cycle.

O27: TOOL IS SPARE TOOL (G)

Tool number is already allocated as a spare tool.
A spare tool has already been allocated to the tool.
Only free spare tools can be allocated to free tools.

O29: EXT. PROGR. CALL WITHOUT PROGRAM NR. (G)

External program call without allocated part program number.

O37: ILLEGAL DELIMITER AFTER ADDRESS (G)

An illegal character detected after syntax check

For example: X100% Y10

O38: ADDRESS WITHOUT DATA (G)

Address without data detected.

For example X Y100.

O40: EXPRESSION IN NON PARAMETER MODE (G)

Algebraic expressions are only allowed in parameter mode.

O44: NON-EXISTING ADDRESS (G)

Address is unknown for this software version.

O45: SIGN NOT ALLOWED IN THIS ADDRESS (G)

Minus sign (-) is not allowed in this address.

O49: PROGRAM LOCKED (G)

Attempt to edit a locked program.

Possible cause: 1) The requested program is locked by the lock attribute (softkey).
2) The programs are locked by an external switch.

O50: MEMORY IN USE, CAN NOT BE CHANGED (G)

Memory is currently in use by a function other than edit.

Solution: Wait a moment and try again.

O51: NUMBER OF PROGRAMS > MC85 (G)

The maximum number of programs is reached, entering new programs in ID-directory is not possible.
Machine constant MC85 contains the maximum number of programs.

Solution: 1) Delete a not used program from memory.
If necessary change machine constant MC85 (remark: memory will be cleared!)

O52: TEMPERATURE MEMORY OVERFLOW (G)

The temperature memory is overflow.

O53: NO TEMPERATURE DATA PROGRAMMED (G)

The temperature word is not found in the program.

O60: T-NUMBER ALREADY EXISTS (G)

The entered T-word is already existing in another block.

O61: T-NUMBER < 3 DECADEN (G)

The T-number contains less than 3 digits (T-number must be more or equal to 100).

O62: OVERSIZED TOOL NEXT TO NORMAL TOOL (G)

The oversized tool is stored in a magazin position next to an occupied position.

Solution: 1) Choose another tool position for the oversized tool
2) Remove the tool in the position next to the oversized tool

O63: NO EMPTY PLACE NEXT OVERSIZED TOOL (G)

A tool is stored in the magazin next to a oversized tool position.

Solution: 1) Choose another toolposition
2) Remove the oversized tool

O66: DON'T COPY BLOCK WITH PROGRAM-NR. (G)

The first block contains the program identification number.
It is not allowed to copy this block.

O67: DON'T DELETE BLOCK WITH PROGR. NR. (G)

The first block contains the program identification number.
It is not allowed to delete this block.

O71:   PROGRAMMING ERROR IN GRAPHIC WINDOW (F)

Incorrect programming of the graphic window N0 or N1 in the graphic parameter memory.

Solution: program always the addresses X, Y, Z, I, J, K

O72:   PROGRAMMING ERROR IN ROUGH CONTOUR (F)

Incorrect programming of the rough contour (N2) in the graphic parameter memory.

Solution: program always the addresses X, Y, Z, I, J, K

O120:   MISSING "OPERATOR" (G)

The required operator for this expression is missing.

O121:   MISSING "OPERAND" (G)

The required operand for this expression is missing.

O122:   MISSING LEFT PARENTHESIS (G)

The left parenthesis in this expression is missing.

O123:   MISSING RIGHT PARENTHESIS (G)

The right parenthesis in this expression is missing.

O124:   "SYNTAX" ERROR IN EXPRESSION (G)

The syntax of the expression is not correct.

O125:   E-PARAMETER IN CALCULATORMODE (G)

An E-parameter is used in the calculator mode.

O126:   TOO MANY PARENTHESES IN EXPRESSION (G)

There are too many parentheses in this expression.

O127:   E-PARAMETER DIVIDE BY ZERO (G)

Value is infinitive because an E-parameter is divided by zero.

O128:   CONVERSION DEGREE-RAD. NOT ALLOWED (G)

Conversion from degrees to radials or vice versa is not allowed.

O129:   INTERMEDIATE RESULTS OUT OF RANGE (G)

Intermediate result is too big or too small.

O130:   EXPONENT SYNTAX ERROR (G)

This exponent is not allowed.

O131:   E-PARAMETER WITHOUT VALUE (G)

There is no value assigned to the used E-parameter.

O132:   CALCULATED VALUE TOO BIG (G)

The result of the calculation is too big.

O133:   CALCULATED VALUE TOO SMALL (G)

The result of the calculation is too small.

O134:   VALUE > MAXIMUM (G)

Programmed value for an address is more than the specified
maximum value e.g. maximum is 100 and in the program is:
X101, X=101, X=100+1.

Solution: program the correct value

O135:   VALUE < MINIMUM (G)

Programmed value for an address is smaller then the specified minimum value.

O136:   "SYNTAX" ERROR (G)

In the program a syntax error is detected.

Possible cause: 1) In an address with an equal sign (=) not a digit, but an arbitrary character is pro-
                    grammed.
                2) In an address with an equal sign (=) more than three characters (maximum is 3) are
                    programmed.
                    Solution: remove the superfluous digits
                3) It is not allowed to use 4,5,6,7,8 or 10 for sensing temperature compensation memo-
                    ry.
                    Solution: program the correct sense values (1, 2, 3, 11, 12 or 13).
                4) In the toolspare there is no 'T' programmed after the equal sign (=).
                    Solution: change program, toolspare format   must be Txx = Tyy.

O137:   CHANGE NOT ALLOWED (G)

                1) A spare tool is allocated to T0: T0=TX
                    Solution: do not allocate to T0

                2) A spare tool allocated to a tool outside the tool changer or a spare tool outside the
                    tool changer allocated to a tool inside the tool changer.
                    For example: Number of tools=20 number of tool positions in tool magazin=10:
                    T9=T11 or T11=T8
                3) A not activated machine constant is entered.

O138:   G-FUNCTION NOT PERMITTED (G)

The programmed G-function in combination with the active modal G- functions is not permitted.

O139:   TOO MANY CHARACTERS IN A WORD (G)

An address has too many characters before or after the decimal point.

O140:   DECIMAL POINT NOT ALLOWED (G)

In the address a decimal point is not allowed.

O141:   ADDRESS NOT PERMITTED (G)

1) In a block a non-permitted address is programmed.

2) In tool spare memory Txx is programmed without allocation, e.g. Txx is programmed. Txx=Tyy must be programmed

3) Programmed spare tool is higher than the number of tools for example in machine constants is set: Number of tools=20 and T22 is programmed.

O142:   COMMENT PARENTHESES NOT CLOSED (G)

During syntax check is detected that the comment parentheses are not closed.

O143:   DOUBLE ADDRESSES IN BLOCK (G)

1) In a block in the Part program or macro there are more than four P-words prgrammed.

2) In a block the same address is used twice or more.

O154:   MEMORY LOCK (G)

The memory is locked with the external lock switch to protect the program against unauthorised editing.

Solution: switch off the memory lock

O171:   STATUS MEASURING PROBE INCORRECT

This error occurs with a measuring cycle (G145-function) in case of :

1) The probe is already in the wanted position

2) after intervention, followed by a reposition movement, the probe status is not equal to the position at the start of G145

Solution: move probe in the wanted position

PROGRAMMING ERRORS:

P01:   ILLEGAL WORD IN A BLOCK (F)

In a block a word is programmed that:

   1) Cannot be used with a programmed or modal active G-function.

   2) Cannot be used in combination with other programmed addresses.

P02:   NECESSARY WORD MISSING FROM BLOCK (F)

In the required block a word is missing that:

   1) must be programmed with a modal active G-function

   2) must be used in combination with other programmed addresses

P03:   UNKNOWN G-FUNCTION (F)

Programmed G-function is unknown for this software version.

P04:   NO FEEDRATE PROGRAMMED (F)

In the program is not yet a feedrate programmed.

P07:   PROGRAMMED DATA OUT OF RANGE (F)

The value of the programmed variable is out of range.

P11:   G22: NO RETURN POSSIBLE (F)

At the end of a macro it is not possible to return to the program that contained the G22 (macro call).

Possible cause: The program containing the G22 was erased from the CNC memory.

P12:   G14/G29 NOT ALLOWED IN TEACH-IN (F)

It is not allowed to use a G14 or G29-function in teach-in mode.

P14:   G77/G79 NO FIXED CYCLE DEFINED (F)

There is no or incorrect fixed cycle defined to execute with G77 or G79.

P15:   G77/G79 NO SPINDLE PROGRAMMED (F)

   1) To execute G77/G79 the spindle must be turning.

   2) To calculate the feed, the speed must be programmed.

P16:   POINT NOT DEFINED (F)

   1) For the programmed point number no axis values are stored in the point memory.
   2) The programmed point number is outside the point memory. A MC setting determines how many points (with a maximum of 255) can be stored in this memory.

P17:   PROGRAMMED AXIS IS NOT SPECIFIED (F)

The programmed axis is not specified via machine constants.

P20:　　　　　　TOOL RADIUS > PROGRAMMED RADIUS (H)

Tool radius is larger than the radius of the programmed or calculated circle.

P21:　　　　　　E-PARAMETER DIVIDE BY ZERO (F)

During parameter calculations a divide by zero is detected.

P22:　　　　　　G14 BEGIN AND END IN REVERSE ORDER (F)

G14 begin block (N1=) and end block (N2=) are in the wrong order.
This would cause the program to be executed in the reverse way, which is not possible. N1= (the starting block) must preceed the N2= (the ending block) in the program.

P23:　　　　　　TOOLRADIUS IS ZERO (F)

The fixed cycle calculation is:

　　　　　　1)　Incorrect because the value is smaller or equal to zero.

　　　　　　2)　Impossible because no toolradius is programmed.

P24:　　　　　　G87/G89: OVERLAP < 1% OR > 100% (F)

The overlap is less then 1% or more then 100%.

P25:　　　　　　SPEED NOT PROGRAMMED (F)

Speed (D address) is not programmed, this means that M3, M4, M13 or M14 cannot be executed.

P27:　　　　　　T-NR. TOO LARGE/WRONG M-FUNCTION (F)

Cause:　　　　　1)　the programmed tool is not in the magazine
　　　　　　2)　the programmed tool is not in the toolmemory

P28:　　　　　　E-PARAMETER NOT DEFINED (F)

　　　　　　1)　For the programmed E-parameter number no value is stored in the E-parameter memory.
　　　　　　2)　The programmed E-parameter number is outside the E- parameter memory. A MC setting determines how many points (with a maximum of 255) can be stored in this memory.

P30:　　　　　　PROG. IS INCH, NC IS METRIC OR VV. (F)

Warning:　　　　during read in conversion inch/metric or metric/inch takes place. Dit kan niet!!!

P31:　　　　　　SPEED OUT OF RANGE (F)

Programmed speed is too low or too high for this gear.

P34:　　　　　　ERROR IN BLOCK N@@@@@ - N@@@@@ (G)

The memory manager detected an error between Nxxxx and Nxxxx.

The blocknumbers are displayed when selecting error text with the softkeys.

P35:   ENDPOINT NOT ON CIRCLE (F)

A circle is programmed using centre point and endpoint.
The distance from the centre to begin point differs from the distance between centre and endpoint more than allowed by the relative machine constant.

Cause: 1) The programmed endpoint is not on the circle.
                    2) The programmed centre is not the centre point of the circle.

P36:   MEASURING PROBE COLLISION (D)

Measuring probe collision during measuring cycle.

Cause: 1)   Measuring probe is not present.
                    2)   Measuring probe is triggered in rest condition.
                    3)   Measuring probe is triggered during positioning movement

P37:   MEASURING SURFACE NOT FOUND (F)

No measuring point found during measuring cycle, because the measuring probe was not triggered during measuring movement.

P38:   MEASURED DELTAS > TOLERANCE (F)

Measured values are outside programmed tolerance.
Tolerance is not programmed.

P39:   MEASURED DELTAS NOT AVAILABLE (F)

During a G49 a delta value is requested for an axis in which no measurement has been performed.
During G46, the tool correction is not possible because the result after processing the delta value is negative.

P40:   MEASURING PROBE NOT ACTIVE (MC840) (F)

Measuring probe is not active because machine constant MC840
is 0 or the measuring probe is not assigned to an axis or the measuring probe is assigned to an invalid axis.

P41:   G23 NO PROGRAM NUMBER (F)

The program number is incorrect or there is no program number programmed for G23.

P42:   G23 BLOCK NOT FOUND (E)

The return block for G23 is not found or is not existing.

P43:   G23 PROGRAM NOT FOUND (E)

The program called with G23 is not found

P44:   G23 CALLED PROGRAM IS BEING EDITED (E)

The program called by G23 is being edited.

P45:   G23 RET. TO PROGRAM THAT IS EDITED (E)

Return program called with G23 is currently being editing.

P47:   M3/M4 BY ACTIVE MEASURING PROBE (F)

It is not allowed to start the spindle with an active measuring probe.

P48:   G-FUNCTION NOT ALLOWED AFTER ROTATION (F)

The programmed combination is not allowed (for example not valid G-function because rotation is on).

P49:   WARNING: TOOLRADIUS > PROGR. RADIUS (F)

Warning: the toolradius is more than the programmed radius.

P50:   IN FMS REMOTE M66 NOT ALLOWED (F)

If CNC is in "REMOTE" only automatic tool changing is allowed.

GEOMETRY PROGRAMMING ERRORS

Note that the geometry errors are always accompanied by error code P34, indicating the blocknumber(s) in which the error was detected.

P51:   LINE WITH POINT ONLY NOT ALLOWED (F)

Programming a line with only a defined point is not allowed in this case.

P52:   LINE WITH ANGLE ONLY NOT ALLOWED (F)

Programming a line with only a defined angle is not allowed in this case.

P53:   LINE WITH POINT AND ANGLE ONLY NOT ALLOWED (F)

Programming a line with only a defined point and angle is not allowed in this case.

P54:   LINE WITH TANGENT POINT NOT ALLOWED (F)

Programming a line with a tangent point to an arc is not allowed in this case.

P55:   IINE WITH INTERSECTION POINT IS NOT ALLOWED (F)

Programming a line with an intersection point with an arc or line is not allowed in this case.

P56:   CHAMFER NOT ALLOWED (F)

Programming a chamfer is not allowed in this case.

P57:   CIRCLE WITH POINT ONLY NOT ALLOWED (F)

Programming a circle with only a defined point is not allowed in this case.

P58:   CIRCLE WITH CENTRE-POINT ONLY NOT ALLOWED (F)

Programming a circle with only the centre point defined is not allowed in this case.

P59:   CIRCLE WITH TANGENT POINT IS NOT ALLOWED (F)

Programming a circle with a tangent point to another circle or a line is not allowed in this case.

P60:   CIRCLE WITH INTERSECTION POINT IS NOT ALLOWED (F)

Programming a circle which has an intersection point with another circle or a line is not allowed in this case.

P61:   INTERSECTION POINT TWO LINES NOT COMPUTABLE (F)

Intersection point between two lines not computable (Lines are parallel).

P62:   INTERSECTION POINT CIRCLE-LINE NOT COMPUTABLE (F)

Intersection point between a line and circle not computable. Line does not run through the circle.

P63:   INTERSECTION POINT CIRCLE-CIRCLE (F)

Intersection point between two circles is not computable (I1= , J1= is missing).

P64:   NO TANGENT POINT CIRCLE LINE CIRCLE (F)

Tangent points between circle-line-circle not computable.

P65:   ROUNDING LINE-CIRCLE TOO SMALL (F)

The connection circle does not fit between line and circle, so there are no two tangent points (radius is too small).

P66:   CIRCLE-CIRCLE WITHOUT ROUNDING (F)

Rounding between two circles not (correctly) defined.

P67:   ROUNDING TOO LARGE CIRCLE-CIRCLE (F)

The connection circle does not fit between the two circles, so there are no two tangent points (radius is too small or too big).

P68:   NO TANGENT POINT LINE-CIRCLE (F)

Tangent point between line and circle not computable.

P69:   ROUNDING TOO LARGE LINE-CIRCLE (F)

The connection circle does not fit between the circle and the line, so there are no two tangent points (radius too small or too big).

P70:   ROUNDING NOT ALLOWED (F)

Programming a rounding is not allowed in this case.

P71:   INIT. ROUNDING NOT POSSIBLE (F)

Default connection circle is not possible.
A possible cause is that the direction of rotation is wrong.

P72:   PROGRAMMED DATA NOT ALLOWED (F)

G63 or M30 is not allowed in this block.
The movement in the previous block has no endpoint.

P73:   START-POINT = CENTRE-POINT (F)

The circle centre point is not allowed as line starting point.

P74:   K1= , J1= OR R1= NOT ALLOWED (F)

Programmed value for R1=, J1= or R1= is not allowed.

P75:   GEOMETRY NOT ALLOWED (F)

During teach-in it is not allowed to program geometric movements.

P76:   COORDINATE MISSING (F)

Necessary coordinate is not programmed.

P77:   G-FUNCTION NOT PERMITTED (F)

The programmed G-function in combination with the active modal G- function is not permitted.

P79:   NO EMPTY PLACE NEXT TO OVERSIZED TOOL (F)

In the tool magazine a tool is situated next to an oversized tool.

Solution: Choose another position

P110:   SPARE TOOL IN PART PROGRAM (F)

In the part program a spare tool is programmed with M6 or M66.

P111:   PROGRAM LOCKED (F)

Part program is not available for changing, program is locked.

Solution: 1) Reset the program memory lock by the external switch
             2) Unlock the partprogram with the relevant softkey

P113:   WRONG PLANE FOR TOOLCHANGE M6 (F)

The axes are in the wrong plane to make a toolchange.

Solution: program a G17, G18 or a G19 function

P114:   TOOL IN SPINDLE NOT FROM MAGAZINE (F)

The tool in the spindle is not from the magazine.

Solution: program M66 to remove the tool manually, after that program a M6 function

SPINDLE ERRORS:

S01:   LINEAR MEASURING SYSTEM PRE-ALARM (H)

Measuring system voltage is reduced by deposit of dirt at the measuring system. The measurement is still correct.

Solution: 1) Cleaning the measuring scale
                   2) Cleaning the measuring window of the transducer

S02:   MEASURING SYSTEM ALARM (C)

Measurement is not reliable because the measurement system voltage is too small.

Solution: 1) Cleaning the measuring scale
                   2) Replace the measuring scale (if the scale is damaged)
                   3) Replace transducer

S03:   MEAS. SYSTEM POWER SUPPLY FAILURE (C)

Measuring system power supply failure.

Solution: 1) Check power supply unit
                   2) Check the regarding drivecard

S04:   MAX. FOLLOWING ERROR TOO LARGE (C)

The following distance is more than 110% of the maximum following distance, defined in machine constants MC515 or MC517 for the first gear range or the relevant machine constants for the other gear ranges.

Possible cause: Defect in the servo system (amplifier, motor or measuring system)

S05:   SOFTWARE LIMIT SWITCH TRIPPED (E)

Spindle is running against the software limit switch.

Solution: 1) clear error message
                   2) jog the spindle from the end switch with the softkeys

GROUP S NUMBER 6: MOTION FASTER THAN RAPID (E)

The calculated rotation speed is more than the maximum allowed speed for the spindle (relevant to the active range, the regarding machine constant is MC571 to 574).

Solution: clear error message and change the program

S7:   DRIFTING OUTSIDE NO-MOTION WINDOW (D)

The spindle position is shifted from the "commanded position" over a distance larger than specified in the no-motion window because of external influences, such as drift in the servo system.

Solution: 1) check the servo system

SYSTEM ERRORS:

If one of the error codes listed below is displayed, a qualified service engineer should be called.

#01:   UNEXPECTED CONDITION (SYSTEM) (A)

#02:   CALCULATION ERROR (C)

#03:   TEMPERATURE IN NC > 65 DEGREES (H)

The CNC contains a temperature sensative switch, which is triggered in the case that the temperature in the CNC becomes higher than 65 degrees centigrade.

#05:   EMERGENCY STOP FROM MACHINE TOOL (D)

The emergency stop button has been depressed or the machine tool interface has generated an emergency stop signal.

#95:   SYSTEM ERROR: (CALCULATION ERROR) (C)

#96:   SYSTEM ERROR: (DIVIDE ERROR) (A)

#97:   SYSTEM ERROR: (OVERFLOW) (A)

#98:   SYSTEM ERROR: (INTERRUPT) (A)

#99:   SYSTEM ERROR: (DEBUG) (C)

The error codes #01-#03 and #95-#99 only occur when something is wrong with the CNC system software. There are no external causes to trigger these faults. If one of these error codes appears please contact your local Philips representative or the Machine Tool Manufacturer.

EXTERNAL ERRORS:

External error codes are derived from six binary inputs which are in conjunction with the optional "automatic package".
The error code number (xx) is the decimal equivalent of the input six-bit binary number (that is; 01 to 63, not 00).
The meaning of these numbers is given in the appropriate documentation of the machine tool manufacturer.

11. FEATURES AND THEIR INTERFACING

11.1 Preparing the CPU386 for IPLC

Figure 11.1.-1 shows the lay-out of the CPU386 in IPLC configuration with the positions of the PROM's and RAM's the shaded area shows the position for IPLC program PROM's.

NOTE: Only the IPLC PROM's delivered with the CPU board can be used (type 27512 or 27010)
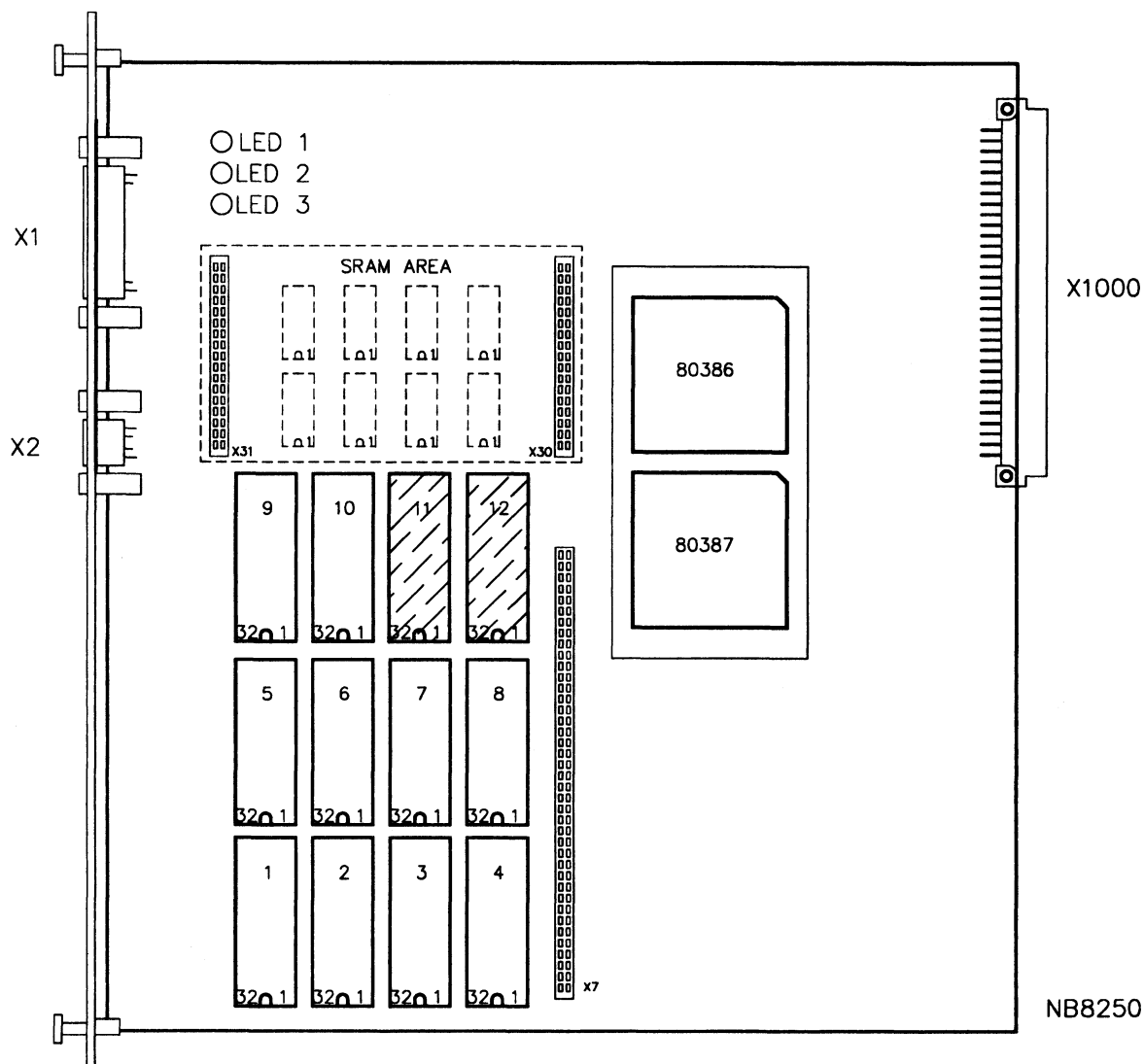
Fig. 11.1.-1   CPU in IPLC configuration; shaded area shows positions for the IPLC program PROM's

Developing an IPLC program can be done with a MIPS software package loaded on a Personal Computer, for downloading and debugging also a CNC-system is needed.
For detailed information about the MIPS software package and the IPLC language see the IPLC Machine Interface Programming Manual.

For detailed information about Machine Constant settings and serial data communication see the Interfacing manual M700.

11.2. REMOTE CONTROL PANEL

11.2.1. General information

Connection cable on Remote Control Panel
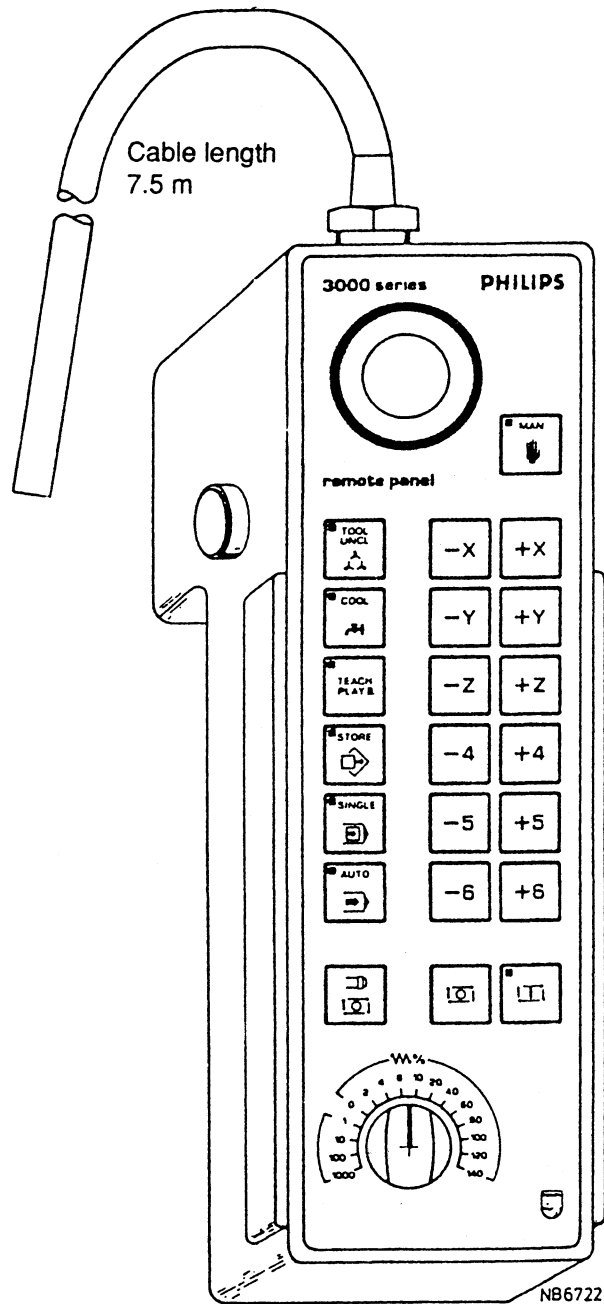
Fig. 11.2.1.-1. Remote Control Panel

The Remote control panel enables the machine tool to be remotely manually controlled.
The control keys on this panel have the same functions as the control keys on the control unit of the CNC 3460.

11.2.2. Connecting the Remote Control Panel
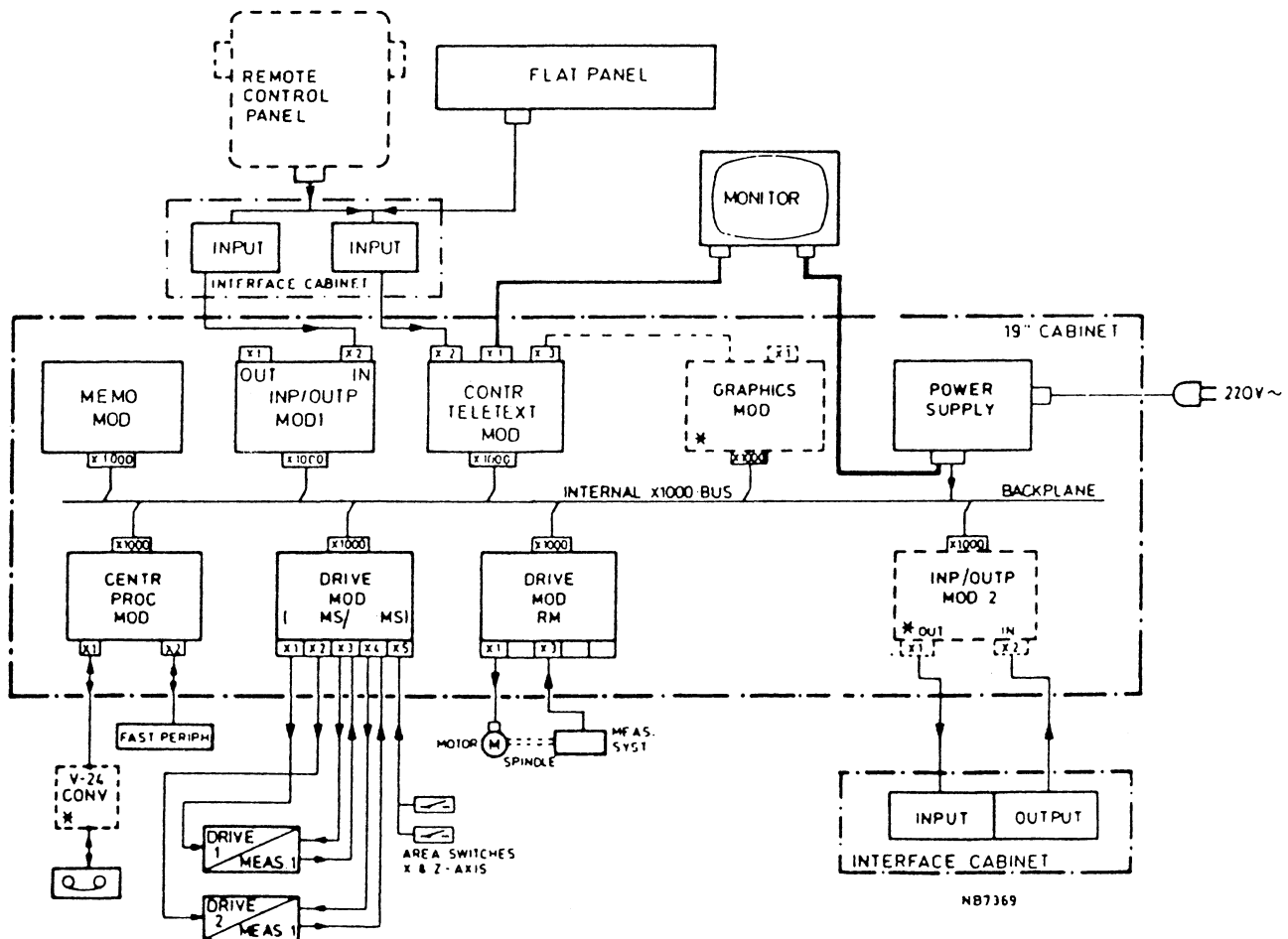
Fig. 11.2.2.-1. Interconnection of Remote Control Panel, Interface cabinet and Control unit

The Remote Control Panel must have a connection with the CONTROL TELETEXT MODULE of the control unit and a connection with four lines of the first INPUT/OUTPUT MODULE for the EXTERNAL FEED OVERRIDE.

The I/O module is connected to the interface cabinet via a 37-core screened cable, and the Control Teletext Module is connected to the interface cabinet via a 30-core screened cable.

The Remote Control Panel is directly connected to the interface cabinet via its 30-core + $3\times0.75mm^2$ screened cable.

In figure 11.2.2.-2 the connection table is given between the control teletext module, the I/O module and the Remote control panel.

control teletext		external jog		I/O module one	
X2	**module**	**X1**		**X1**	
pin	**mnemonic**	**pin**	**colour**	**pin**	
1	SCAN0	3	green		
2	SCAN2	5	grey		
3	SCAN4	7	blue		
4	SCAN6	9	black		
5	RETA0				
6	RETA2				
7	RETA4				
8	RETA6				
9	OUT00	17	yellow/brown		
10	OUT02	20	white/pink		
11	OUT04	22	white/blue		
12	OUT06	24	white/red		
13	RETB0				
14	RETB2	11	grey/pink		
15	RETB4	13	white/green		
16	HORIZ				
17	+5VF1	15	brown/green		
16	white/yellow				
18	PPULS				
19	0VF2	25	brown/red		
26	white/black				
20	SCAN1	4	yellow		
21	SCAN3	6	pink		
22	SCAN5	8	red		
23	SCAN7	10	violet		
24	RETA1				
25	RETA3				
26	RETA5				
27	RETA7				
28	OUT01	18	white/grey		
29	OUT03	19	grey/brown		
30	OUT05	21	pink/brown		
31	OUT07	23	brown/blue		
32	RETB1				
33	RETB3	12	red/blue		
34	VIDEO				
35	VERTI				
36	0VF1				
37	0VF2				
		1	white	35-37	OUTPUT SIDE +24VE
		2	brown	35-37	INPUT SIDE 0VE
		27	brown/black	5	FEED OVERRIDE B1
		28	grey/green	6	FEED OVERRIDE B2
		29	yellow/grey	7	FEED OVERRIDE B3
		30	pink/green	8	FEED OVERRIDE B4
		14	key		
	3x0.75mm^2	black ⎤ ⎦ black	Emergency switch		
		green	not used		

Fig. 11.2.2.-2. Table for connecting the Remote Control Panel

11.2.3. Feed override switches

There are four signals generated by the Remote Control Panel that are used to proportionally override the selected feedrate.

It is also possible to mount external feed-override switches on the machine tool operator's panel.
In this case the feed override switches of the Remote Control Panel may not be used.
It is possible to connect external feed override switches when there is no Remote Control Panel. In this case the value in MC4 must be 5.

These four signals are connected to the I/O Module as shown in figure 11.2.2.-2 and represents a bit code. The truth table of this bit code is given in figure 11.2.2.-3.

input	8	7	6	5	Feed override position (%)
	0	0	0	0	140
	0	0	0	1	120
	0	0	1	0	100
	0	0	1	1	80
	0	1	0	0	60
	0	1	0	1	40
	0	1	1	0	20
	0	1	1	1	10
	1	0	0	0	8
	1	0	0	1	4
	1	0	1	0	2
	1	0	1	1	0

input	8	7	6	5	Feed override position (incr.)
	1	1	0	0	1
	1	1	0	1	10
	1	1	1	0	100
	1	1	1	1	1000

Fig. 11.2.3.-1. Truth table of the signals FEED OVERRIDE B1 up to B4. Pin 5 to 8 on I/O Module.

11.2.4. Machine constant concerning the Remote Control Panel

There is only one machine constant present for the Remote Control Panel option, and that is to select the mode in which the Remote Control Panel will be used. The machine constant mentioned is machine constant MC4 and the values that can be assigned are:

0   for no remote control panel
1   for remote control panel operative, incl. safety switch
2   for remote control panel operative, except safety switch
5   for connection of external feed override switch. See input card 1, inputs 5-8. The pushbuttons on the remote control panel are inoperative now.

11.3. MEASURING CYCLES

11.3.1. Connecting the Measuring probe

By connecting a measuring-probe to the control-system, automatic workpiece measurements and/or tool measurements can be performed, provided the software contains the option 'measuring cycles'.

The measuring-probe may either be fixed (at a certain position with respect to the machine reference point R) or moving.

Three types of measuring-probes can be used by the CNC. An inductive probe or an infra-red measuring probe (shown in the figure below) or a hard wired probe.

NB7078
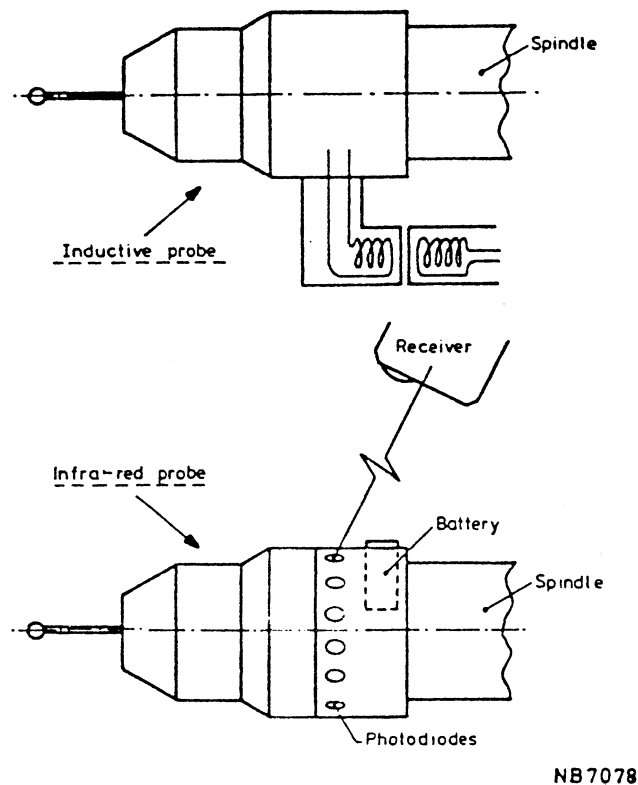
Fig. 11.3.1.-1. Inductive and infra-red measuring probes

The probe is connected to pin 1 on socket X5 (AREA-switch input) of the measuring-system which has been selected by MC840; usually the AREA-switch of the electronic handwheel is taken.

11.3.2. Inputs and outputs concerning measuring cycles

If a measuring cycle has been programmed the signal ACTIVATING MEASURING PROBE is set on the IPLC window. Via the IPLC this signal must be transferred to an output. The signal is used to indicate that a measurement takes place. In case an infra-red measuring probe it is used also to switch the probe on. The measuring probe signals its active state to an input. This input must be set on the window by the IPLC program as MEASURING PROBE READY. This signal has also the function to inform the control that the infra-red path is obstructed if an infra-red probe is used.

There is also a signal AIR BLOWING MEASURING PROBE present at the IPLC window. This signal can be used to switch on the air supply for cleaning the surface to be measured. The air supply can last 384 seconds at most.

11.3.3. Machine constants concerning measuring cycles

See for the description of the machine constants concerning measuring cycles section 14 of this manual.

The drive-card to which the Measuring probe is connected is defined by means of MC 840. By means of MC 841 the type of measuring probe is defined, and MC 847 is used to define the width of the fixed measuring probe.

NB7016
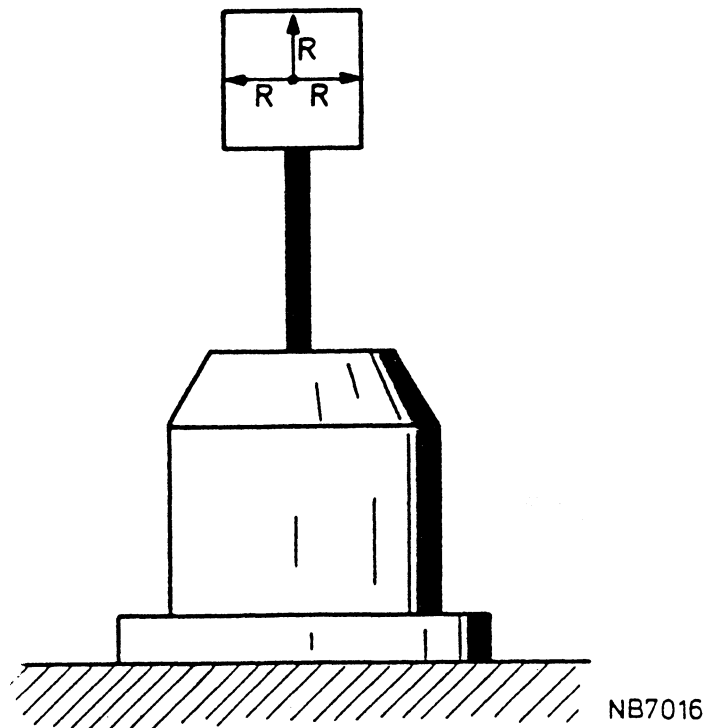
Figure 11.3.3.-1. Width of measuring probe.

For every axis the position of the fixed measuring probe can be defined with the help of a machine constant. The position is defined with respect to the machine reference point R (MC 240, MC 241; MC 290, MC291; MC340, MC341). See figure 11.3.3.-1.

For calibration of the measuring probe it is possible to have a calibration ring on the machine tool which radius is defined by MC 848 and which centre position is defined with respect to the machine reference point R by means of a machine constant for every axis (MC 242, MC 292, MC 342, MC 392, MC 442 and MC 492).

The actual measuring cycle is defined by means of 5 machine constants. The probe will be moved to the point of pre-measuring distance (MC 844) with rapid speed (MC 849). Before the pre-measuring distance is reached the workpiece surface can be cleaned by means of an air supply for a time defined by MC842. During the pre-measuring distance, the measuring distance and the post-measuring distance (MC 845) the measuring probe will move at the measuring feed rate (MC 843).

The retraction of the measuring probe is done with rapid. Before the pre-measuring distance the workpiece surface can be cleaned by means of an air supply for a time defined by MC 842.

If this time is expired the pre-measuring distance can start.

STARTPOINT OF
MEASURING FEED

DEFINED
MEASURING POINT

PRE-MEASURING
DISTANCE

WORK PIECE

N B7074
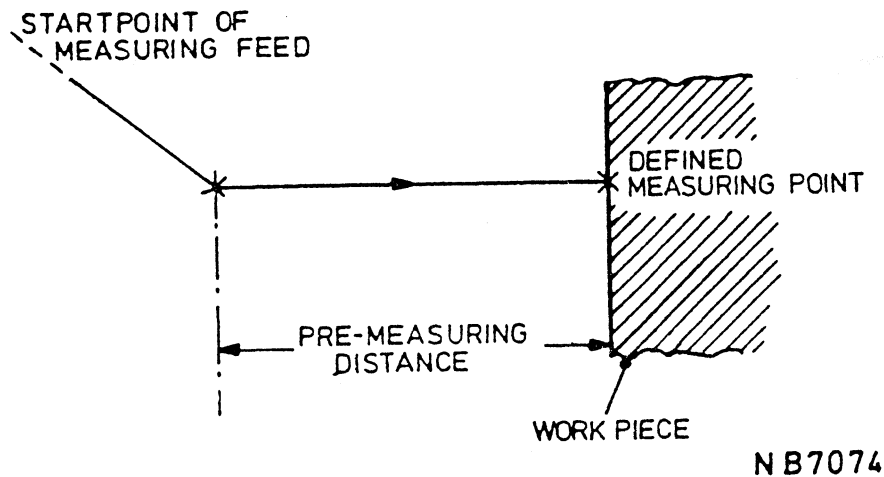
Fig. 11.3.3.-2. Pre-measuring distance.

V

START POINT
OF MEASURING
FEED

programmed
measuring
point

MC 845
Post-measuring
distance

Retraction with rapid

End point of
measuring feed

MC 844
Pre-measuring
distance

N B7081
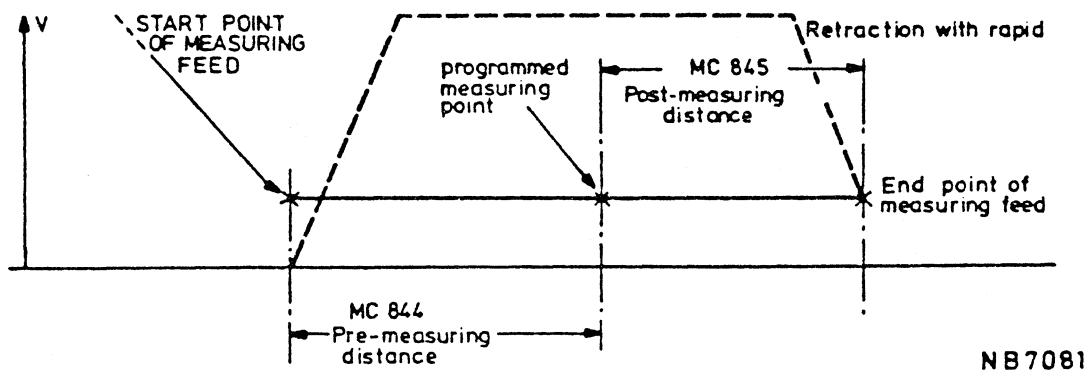
Fig. 11.3.3.-3. Post-measuring distance.

The collision detection can be changed to collision detection with delay for rapid movements during measuring cycles.

The delay is used to prevent probe triggering as a result of machine tool vibrations during rapid movements.

The selection of the movements is done by MC 850, and the delay is done by MC 851 in steps of 15 msec.

With measuring cycle G50 the angular displacement of a rotary table can be measured, see figure 11.3.3.-4.
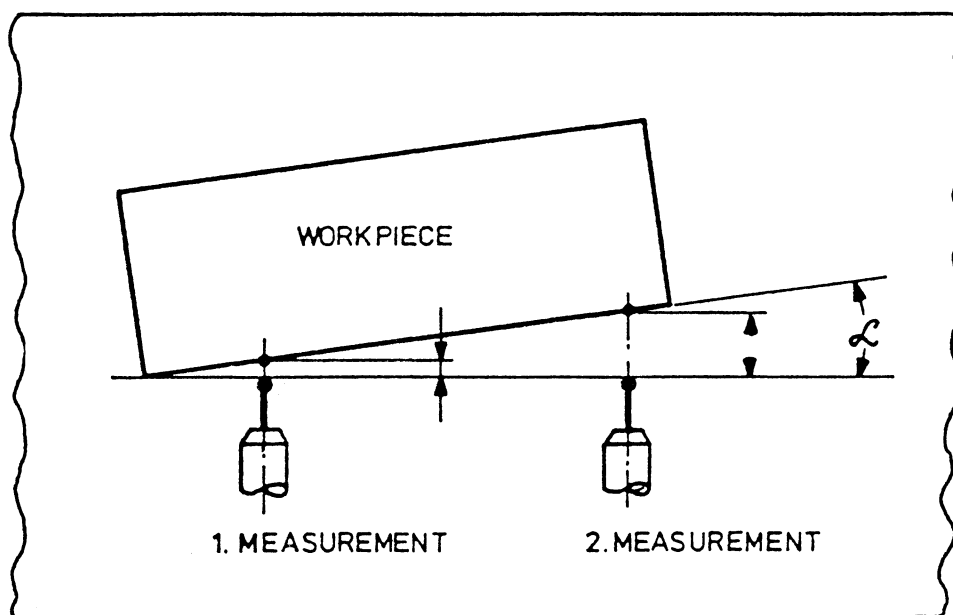
Fig.11.3.3.-4. Resolution of the rotary axis.

The resolution of the rotary axis in degrees per countpulse is defined in MC846.

11.3.4. Renishaw probe MP7

An infra-red measuring probe is normally switched on by an infra- red flash, and is switched off by a delay timer inside the probe in order to save battery power.

The Renishaw measuring probe MP7 is switched on and off by spinning the probe which activates a centrifugal switch inside the probe. The Renishaw probe is selected by entering a value 3 in machine constant MC 841. The spinning speed is defined by machine constant MC 852 and the spinning time by machine constant MC 853. The Renishaw probe must spin for at least one revolution at a minimum speed of 500 revolutions per minute. The minimum time between each spin on/off is defined by machine constant

MC 854 and must be at least 15 seconds for the Renishaw probe. This switch on/off delay time is set to be sure that the centrifugal switch is in rest.

11.4. AUTOMATION FUNCTIONS

The CNC contains several automation functions which are related to tool handling, external program call and external activation of conditional jumps in a program.

11.4.1. Tool handling

The tool change sequence is programmed in the IPLC. The size of the tool memory in the CNC can be defined with MC27 and the number of places in the tool magazine is entered in MC28. Via the IPLC it is possible to program a manual tool change sequence and/or an automatic tool change (with or without random access tool memory).

The tool memory can be extended by a tool life monitoring system. The remaining life time of the tools concerned is calculated in steps of 1 minute. When tool life is exceeded an error code is displayed or a spare tool is used at the next tool change for the relevant tool. The selection is done with MC29.

The tool memory can also be extended by a cutting force monitor. The cutting force of the tool is measured by an external measuring device. The CNC supplies the measuring device with two threshold levels for the active tool. In case the first level is exceeded, the measuring device returns a signal to the input module (to be processed by the IPLC to 'FIRST THRESHOLD EXCEEDED'). The CNC generates a warning on the display.
As soon as the second level is exceeded a second signal is generated by the measuring device ('SECOND THRESHOLD EXCEEDED'). In this case an alarm is generated or the spare tool is used at the next tool change for the relevant tool.

During tool change it is possible to measure the tool length with an external tool breakage measuring device. This function is activated by MC32.
The CNC supplies the measuring device with a tolerance value defined in MC33. The measuring device measures the tool when it is placed in the spindle and again when it is replaced in the tool magazine. In case the two measurements deviate beyond the tolerance level an error is generated.

The tool data output via the IPLC can be done in BCD or in four decade. The selection is made with MC35.
MC37 is used to select the format of the tool word in the tool memory.

11.4.2. External program call

An external device (for instance a computer) can call a program from the program memory in the CNC and activate it.
The program selection is done via an input module and is processed by the IPLC.
The input allows a 3-decade number.

This number is used by the CNC in two ways; the selection is done with MC42:

1.- The input refers to the last 3 decades of a program number. So input of 342 will call program number N9342. However when the memory contains a program with for instance identification number N94342, this program can be called too.
The CNC will scan the program memory from top to bottom and activate the first encountered program. Assign 1 to MC42 to enable this function.

2.- An external program call memory is activated in the CNC. In this memory the program number is assigned to a parameter, for instance E126=N9342. In this case <u>not</u> the program number is transferred to the CNC but the parameter number (126 in this example). In this way there is no confusion about simular program numbers, but there is no link between the program number and the parameter number. The number of parameters is defined in MC43 with a maximum of 256 (E0- E255). Assign 2 to MC42 to activate this function.

The external program call is active as long as the external device activates an input. The program call numbers are stored in two buffers. Inputs -which must be processed by the IPLC to CHANGE FIRST and SECOND EXTERNAL PROGRAM CALL- define which buffer must be loaded. When the buffers are empty the CNC returns a signal.

11.4.3. Conditional jump

Programmed conditional jumps are always executed when MC44 is set to 0.
The execution of conditional jumps depends on the status of an input signal (determined by the IPLC).
Assign 1 to MC44 to enable this function.

11.5 AUXILIARY AXIS MODULE

11.5.1 INTRODUCTION

An auxiliary axis module is a microprocessor controlled card capable of driving a linear positioning axis. It contains, besides 8085-microprocessor, 16K of PROM-memory with system software and 4K RAM (standard). The RAM-memory can be extended to 16K. The module is also equipped with hardware for the measuring system, the digital inputs and outputs and for communication with the main- processor on the Central Processor Module. Auxiliary axis modules are mainly used to control automatic material-supply and transport systems which support the lathe. The axis can be operated manually via manual-menu option "AUXILIARY AXIS"; see operator's manual. It is possible to have 2 auxiliary axis modules in the 15-slots and the 20 slots rack.

11.5.2 SPECIFICATION AND CONFIGURATION

The module has been designed to control a linear positioning axis. It operates separately from the main axes X, Y and Z.
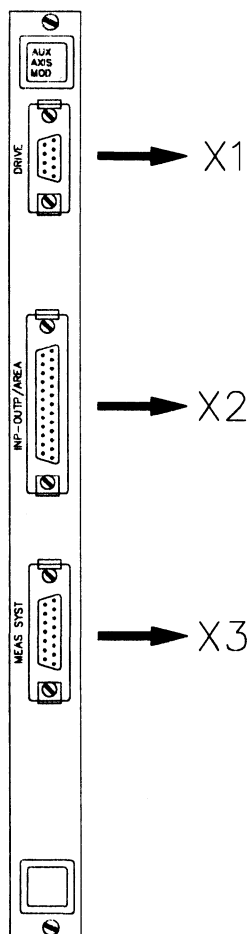
For the purpose of being able to operate separately, the board is equipped with a socket (X2) for AREA-switch connections.
Besides this, it has a socket (X1) to drive the servo-amplifier and a socket (X3) to which a linear or a rotary measuring system can be connected; this, to be able to operate in closed-loop position control. The positions of the respective sockets are shown in the drawing at the left (figure 11.5.2.-1).

As it is possible to use 2 auxiliary axes, each module must be addressed differently.
The address of a module (axis number) is defined by setting the 4 DIL-switches on each board.
The locations of these switches (S1) on the module are shown in figure 11.5.2.-2.

NB7294

Fig. 11.5.2.-1. Positions of sockets on the front of aux. axis module
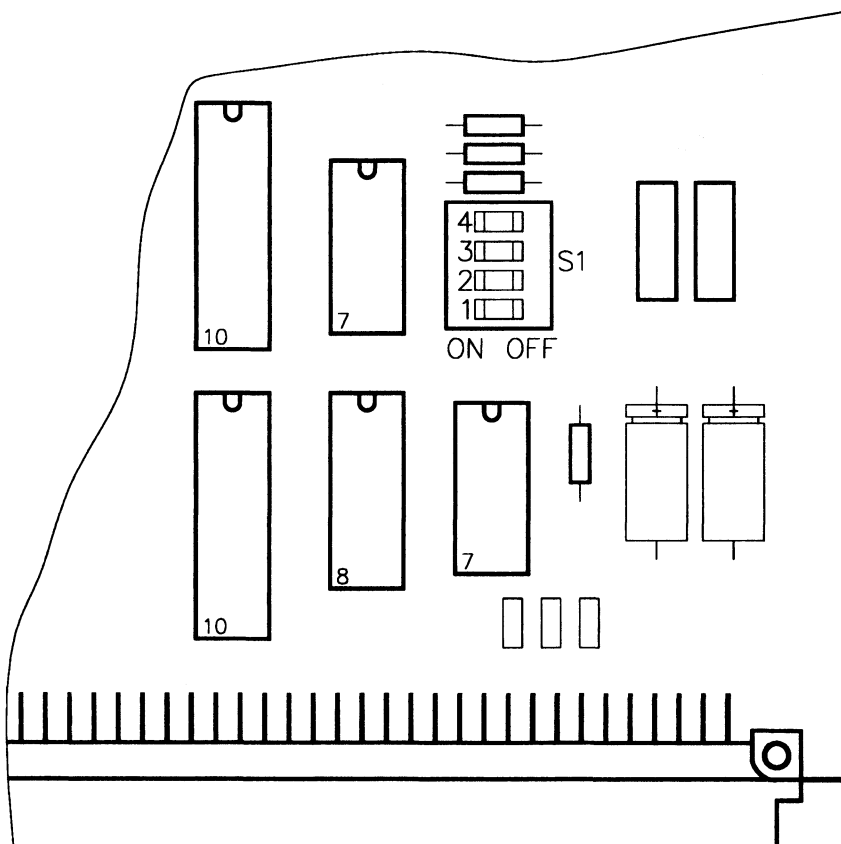
NB7757

Fig. 11.5.2.-2. The address-selector switches on the module

The positions of the 2 switches to address the axes H1 and H2 are given below.

AUX. AXIS NUMBER	SCREEN INDICATION	SWITCH POSITION			
		S1-4	S1-3	S1-2	S1-1
1	H1	OFF	OFF	OFF	OFF
2	H2	OFF	OFF	OFF	ON

The module can be equipped with 4 EPROM-chips and 2 static RAM-chips for system software.
In most cases the software has been put in 3 chips which must be placed in sockets 27A, 27B and 27C as indicated in figure 11.5.2.-3.
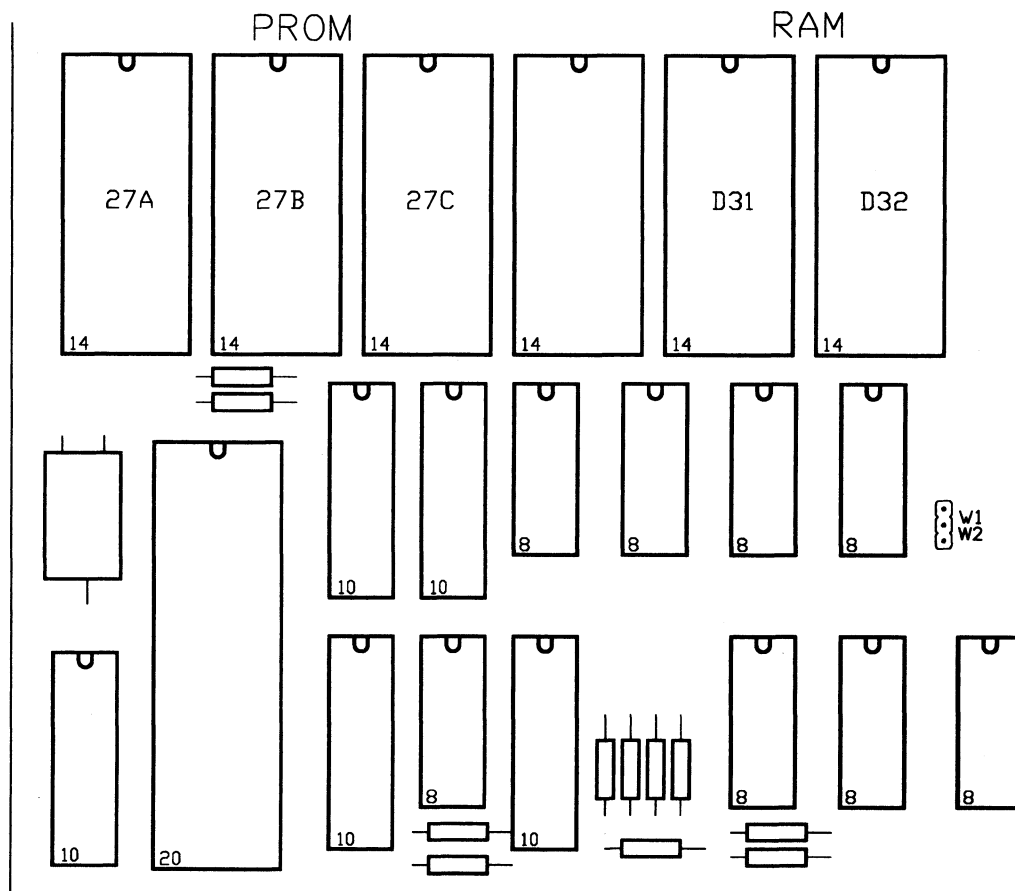
Fig. 11.5.2.-3. The PROM and RAM sockets on the module

For RAM-memory, 2K*8-chips (HM6116) and 8K*8-chips (HM6264) can be used, depending on the required memory size.
The chips must be placed on positions D31 and D32. See figure 11.5.2.-3.

IMPORTANT: (see figure 11.5.2.-3)
- When using 2K*8-chips, the jumper must be put in position W2.
- When using 8K*8-chips, the jumper must be put in position W1.

The auxiliary axis module is equipped with 2 sockets for Fusible Link Proms (FLP) to define the on-board addressing and memory-mapping.

In figure 11.5.2.-4 the positions of the sockets are shown; they are marked "FLP I" and "FLP II".

The identification numbers for the required FLP's are:
- FLP I : 02171 D61
- FLP II: 02161 D43

The above mentioned FLP's are fitted standard from factory.

NB7756
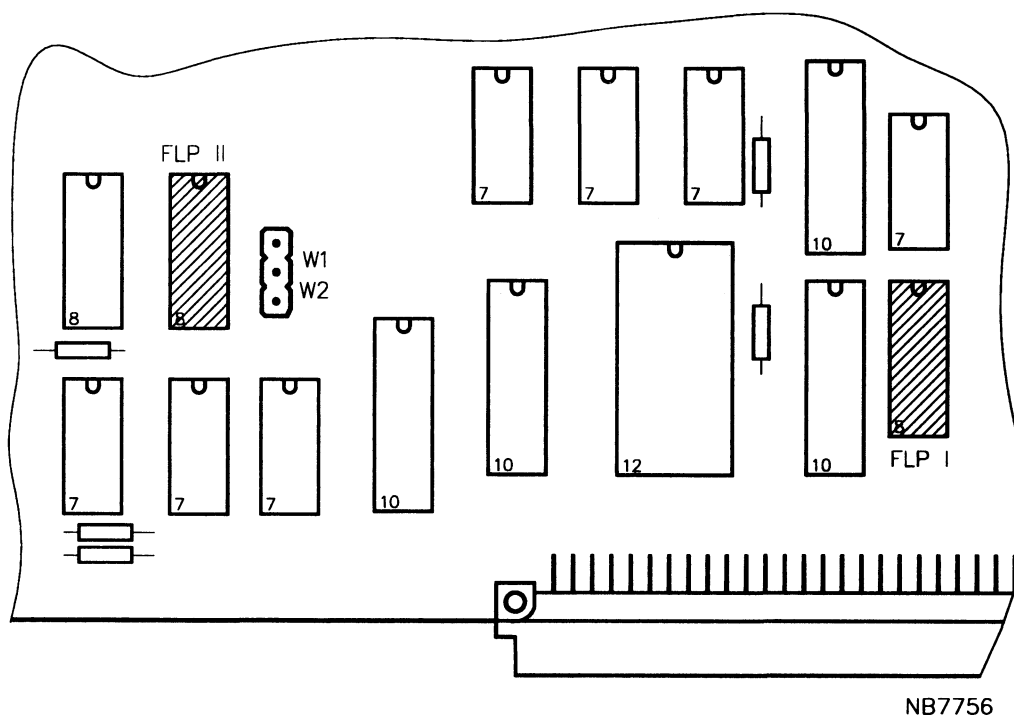
Fig. 11.5.2.-4. The fusible link proms on the module

11.5.3. THE MEASURING SYSTEM

Per auxiliary axis module a linear or a rotary measuring system can be connected.
As linear measuring system, the PHILIPS PE 2580/20 can be used.
As rotary measuring system, all 5V and 12V transducers can be used which have S00/S90 output-signals in accordance with the specification given in section 6.1.

The adjustments to connect a certain measuring-system type (RM 5V, RM 12V or LM) are made via interconnection of pins on the plug of the relevant measuring-system's cable. Wiring instructions on connecting the cable for the measuring-system are given in section 3.3.

Linear measuring system

When applying a linear measuring system, the module requires the Philips PE2580/20 transducer to be employed. The screened cable to connect the transducer can be ordered from Philips with ordering number DT 0712 187 09001. Figure 11.5.3.-1 shows the connection of the transducer PE2580/20 with the measuring system input of the auxiliary axis module. Note that in figure 11.5.3.-1 the supply wires (0V and 12V) have been drawn thicker than the other wires to indicate that for these supply-voltage lines the thick wires in the specified cable are to be used.
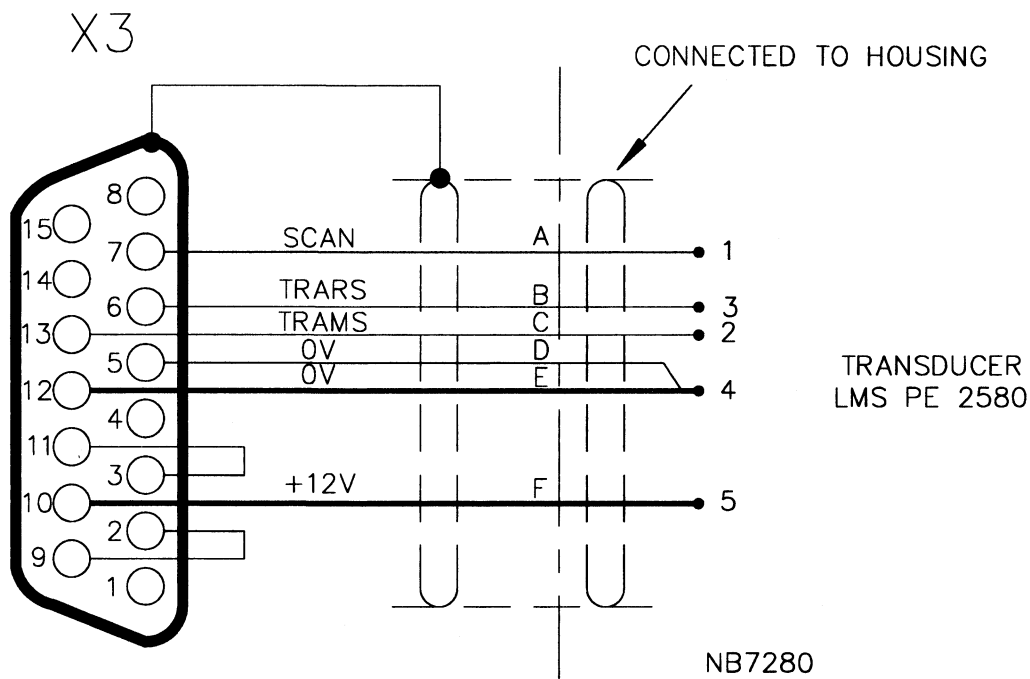
Fig. 11.5.3.-1. Connection diagram for the PHILIPS PE2580/20 linear transducer

Rotary measuring system

Since the auxiliary axis module can deliver up to 300mA to the rotary transducer, the two supply-wires in the cable must be thicker than the signal wires.

Therefore a special cable is needed to connect both 5V and 12V transducers.

This screened cable has $2 \times 3 \times 0.14$ mm^2 wires plus 2×0.75 mm^2 wires for supplying the transducer; it can be ordered from Philips with ordering number 0722 491 00001.

Figure 11.5.3.-2 and figure 11.5.3.-3 show the connection diagrams for respectively 5V and 12V transducers.
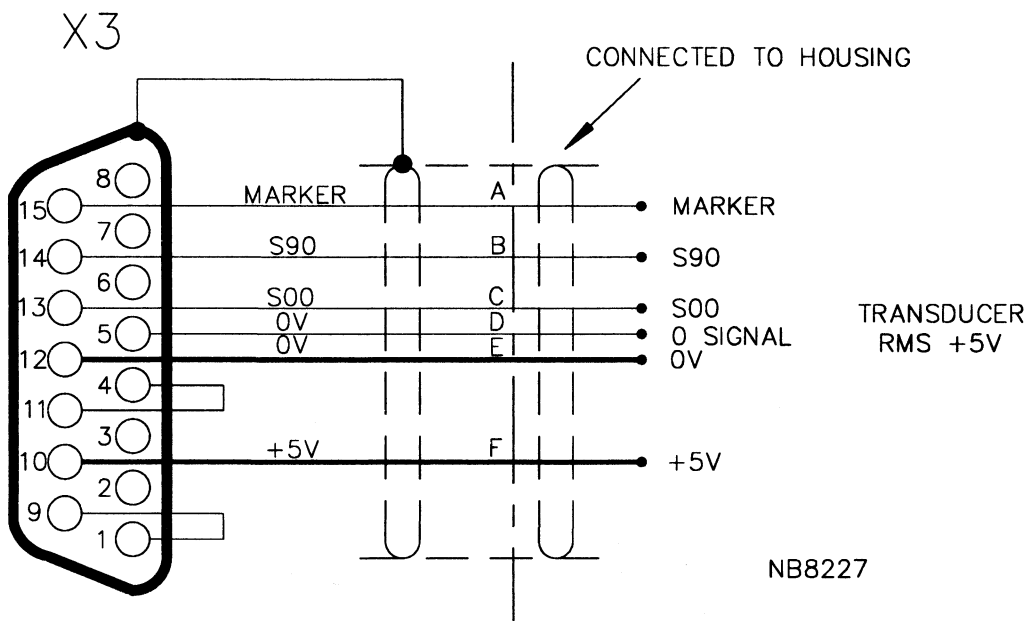
Fig. 11.5.3.-2. Connection diagram for 5V rotary transducer
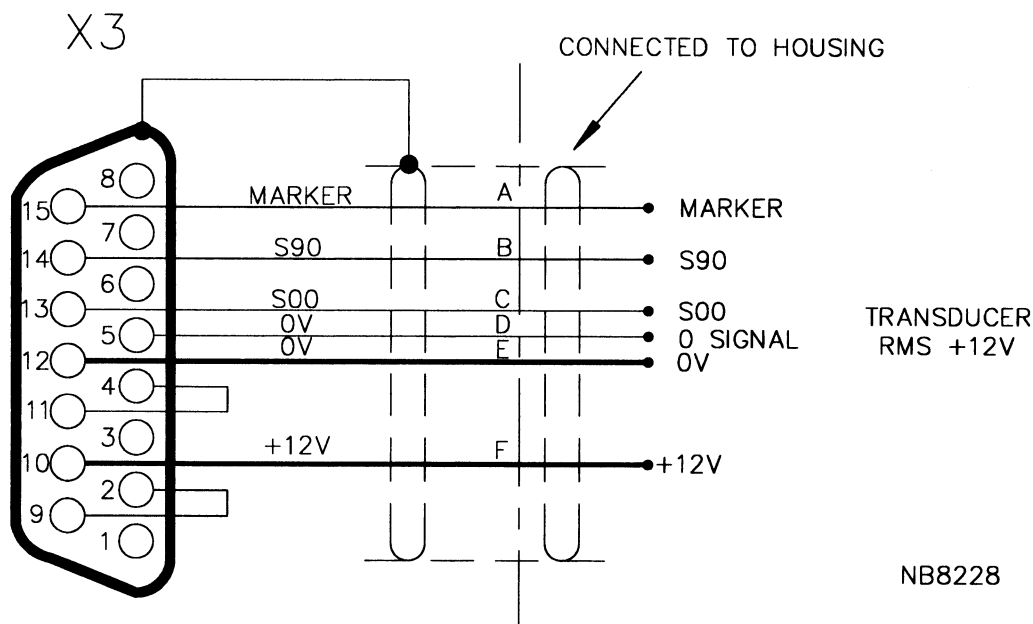
Fig. 11.5.3.-3. Connection diagram for 12V rotary transducer

Analog servo drive connection (X1)

The auxiliary axis module provides an analog output signal to drive the servo amplifier. Together with the measuring-system input X3, close-loop position control can be established.
The servo drive system to which the drive output is connected, must have DIFFERENTIAL inputs.
The characteristics of the drive output are:

- Output voltage: +10V ... 0 /V ... -10V
- Output current: 5 mA max.

NOTE: The analog output may never be short-circuited to earth, otherwise the output will be damaged.

Figure 11.5.3.-4 shows the connection diagram for the analog-drive cable.
Wiring instructions on connecting this cable can be found in section 3.3.

The cable, 4 x 0.14 mm^2 with screen, can be ordered from Philips with ordering number 0712 187 03005.
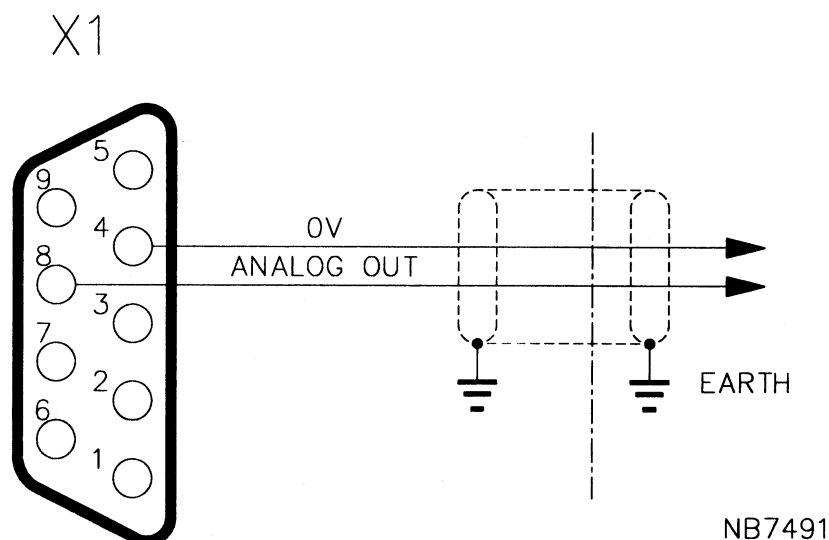
Figure 11.5.3.-4. Connection diagram for analog-drive cable

11.5.4.   INPUTS, OUTPUTS AND AREA SWITCH

The input and output signals must be programmed in a IPLC program and are therefor not described in this manual. Information about the I/O configuration must be obtained from the IPLC program.
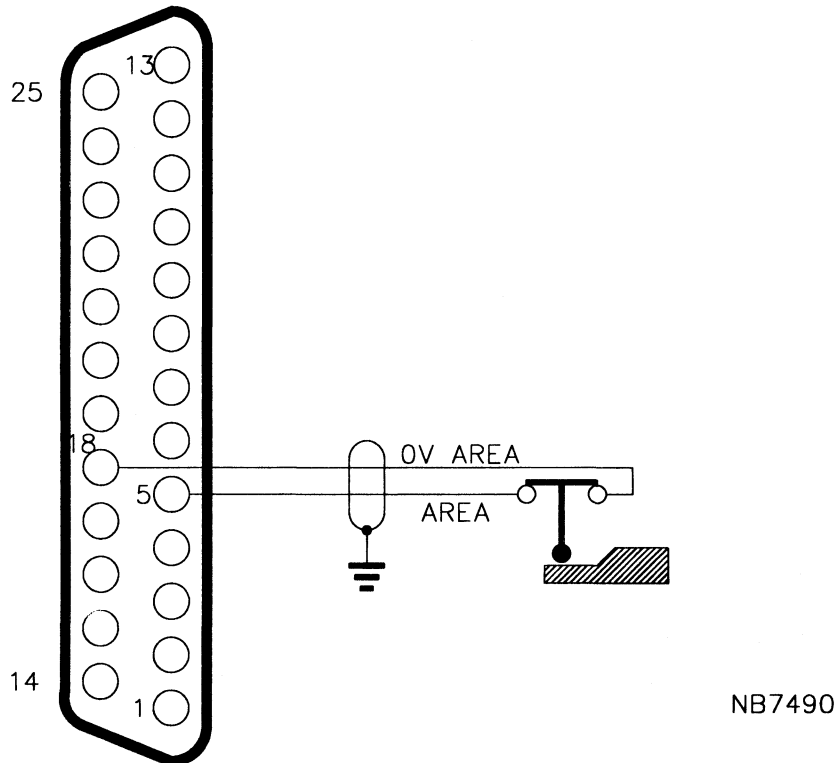
Fig. 11.5.4.-1. Wiring diagram for socket X2

Figure 11.5.4.-1 shows the wiring diagram for socket X2 to connect area-switches. Wiring instruction on connecting the cables to socket X2 can be found in section 3.3.

11.5.5.   MACHINE CONSTANTS

For each auxiliary axis, a block of 100 numbers for machine constants has been reserved to set the parameters of the related axis.
As it is possible to have 2 auxiliary axis modules in the rack, 2 blocks of machine constants are available. Which block has been assigned to an axis, is shown below.
The number of auxiliary axes can be set by machine constant MC91:

    Assign   0   for no aux. axes
              1   for 1 aux. axes
              2   for 2 aux. axes

    MACHINE CONSTANTS   AUXILIARY AXIS

    From 1800 to 1899   H1
    From 1900 to 1999   H2

The descriptions of auxiliary axis machine constants are given in section 14.

11.5.6.   SWITCHING ON WITH AUXILIARY AXIS

After switching on for the first time with auxiliary axes it is important to set all machine constants related to these axes (1800 to 2199) with provisional values. For a certain number of these constants no common value can be given since they strongly depend on the application and use of the axes.

Depending on the number of which are going to be employed, MCxx23 of each axis MC-block must get the value "1" to activate the axis and "0" to de-activate it.

Besides this constant, MCxx33 determines whether the activated axis is to be used in DEMO mode or in operational mode.

Principally, installation of auxiliary axes is not different from installing the main-axes X and Z. Therefore, see also the sections 12 and 13 when installing the auxiliary axes. Items like determination of MC-values for axis-gain characteristic, velocities, limit-switch settings and RPF-offset positions is done in a similar way as to the main axes.
Optimalisation of constants can in most cases only be done via the experimental way. To know the function of a constant and the effect of different values on the operation of the equipment, carefully read the description of the constant(s) in section 14 especially when changing machine-constant values.

Safety precautions

Since in most applications auxiliary-axis modules are used to control material transport and supply systems, they are mostly situated in the surrounding area of the lathe. Transport lines, driven by auxiliary-axis modules can reach velocities of up to 4 m/s. Therefore, be extremely careful when operating this equipment at installation and when testing programs.

WARNING: Be always sure that during any of these circumstances NO objects or persons are present in the working area of these systems.

Hints for safety at installation and at testing or debugging IPLC program modules.

- Test the emergency button during movement of the auxiliary axes to be sure of functioning in case it may be operated in an emergency situation.

- When testing programs never be out of reach of the emergency stop button.

- Always have visual contact with the axes to see their response: if this is not possible, have a second person at hand who can see the axes and can report their response.

- Rules to be obeyed when testing and debugging programs:

    NEVER: - attempt to test too much at a time
             - write programs without an overall planning
             - test and debug programs without an overall plan

All these faulty approaches can result in dangerous situations as well as being inefficient and time-consuming.

12. PUTTING THE AXES INTO OPERATION

12.1. DETERMINATION OF PRELIMINARY VALUES FOR MC'S

* **Preliminary values for MC 203, MC 253, MC 303, MC 353 MC 403, MC 453**

The above machine constants allow the resolution of any rotary transducers to be defined. MC 203 applies to axis 1, MC 253 to axis 2, etc.

Proceed as follows:
Determine the required number of count pulses per S00/S90 cycle. For 4 pulses per S00/S90 cycle assign the value 2 to the MC concerned.
For 2 pulses per S00/S90 cycle assign the value 1 to the MC concerned.
For 1 pulse per S00/S90 cycle assign the value 0 to the MC concerned.

Any values assigned to these machine constants need not be changed when linear transducers are being used in the corresponding axes or when the axes are not present.

Refer also to section 6, paragraph "rotary measuring system"

* **Preliminary values for MC 205, MC 255, MC 305, MC 355, MC 405, MC 455**

These machine constants allow the maximum velocities (rapid traverse rates) in the machine tool axes to be defined. MC 205 applies to axis 1, MC 255 to axis 2, etc.
For Metric systems, the values to be assigned to these machine constants are determined by the formula:

MC-value = max. slide velocity (m/min) x 10,000

Example:
assuming that the maximum velocity in an axis is 100 Inch/min, the MC-value is 100 x 100 = 10,000

* **Preliminary values for MC 206, MC 256, MC 306, MC 356, MC 406, MC 456**

The values assigned to these machine constants should be equal to those assigned to MC 205, 255, 305, 355, 405, 455.

* **Preliminary values for   MC 215, MC 216, MC 217**
                               MC 265, MC 266, MC 267
                               MC 315, MC 316, MC 317
                               MC 365, MC 366, MC 367
                               MC 415, MC 416, MC 417
                               MC 465, MC 466, MC 467

The above machine constants allow the preliminary values for the gain of the servo systems to be defined. MC 215-217 apply to axis 1, MC 265-267 to axis 2, etc.

When one or more axes are not employed, the corresponding machine constants must be assigned the value 32 000.

Figure 12.-1 illustrates how the preliminary values can be determined. The Kv-resultants shown represent a number of gain factors (Kv) used in servo systems.

Proceed as follows:

Draw a horizontal line from the point on the vertical axis indicating the maximum velocity of the slide.
When the Kv-factor of the servo system employed is known, use the closest Kv-resultant. In case of doubt, always use the lower one. When the Kv-facor is unknown, use the resultant representing Kv 0.5. Draw a vertical line from the intersection point of the horizontal line just drawn and the Kv-resultant selected.
The point where this vertical line intersects with the horizontal axis gives the value (x1000) to be assigned to machine constants 215, 265, 315, 365, 415 or 465, depending on the axis concerned.
Figure 12.1.-1 shows that a slide velocity of 4m/min and a Kv- factor of 0.5 produces a MC-value of 9000.

Remarks: The MC-value found must also be assigned to the associated 2 machine constants of the relevant axis. For instance, the value found for MC 215 of axis 1 must be assigned to MC 216 and MC 217 as well.
The minimum value that can be assigned to these machine constants is 128, the maximum value 28500.
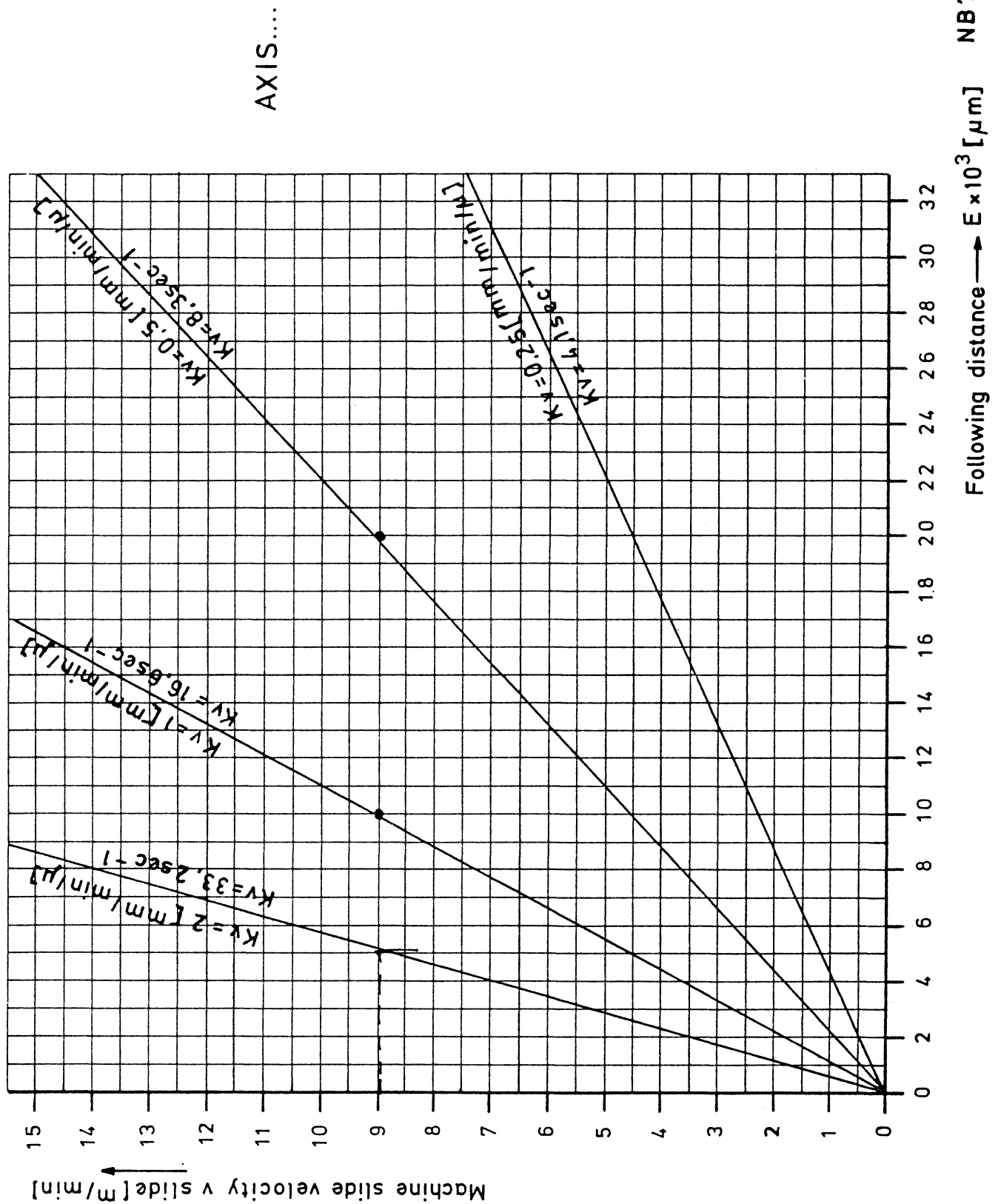
Fig. 12.1.-1. Graphical determination of the servo drive gain from machine slide velocity and following error
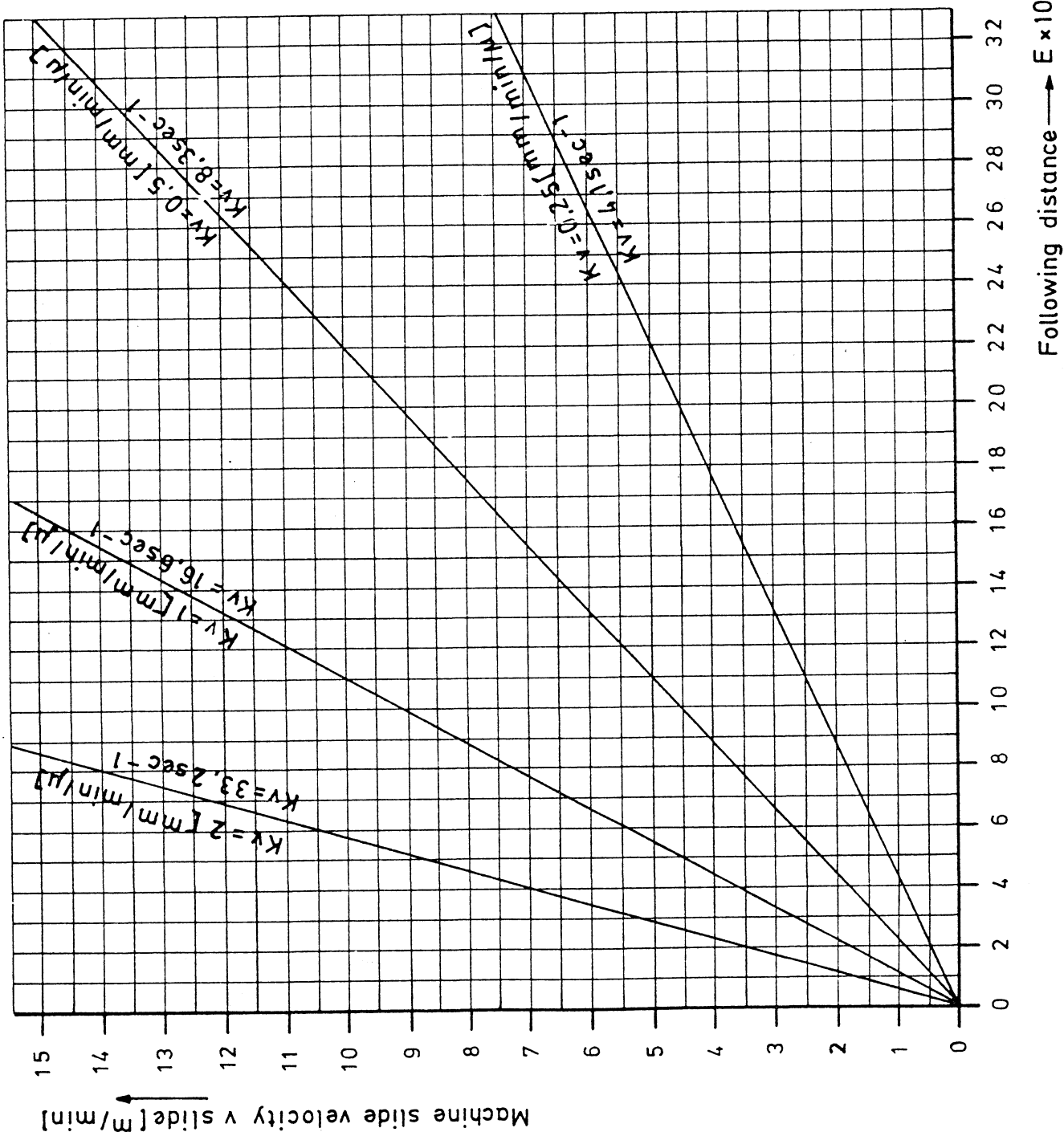
AXIS....

Fig. 12.1.-2. 1st plotting sheet for determination of servo drive gain
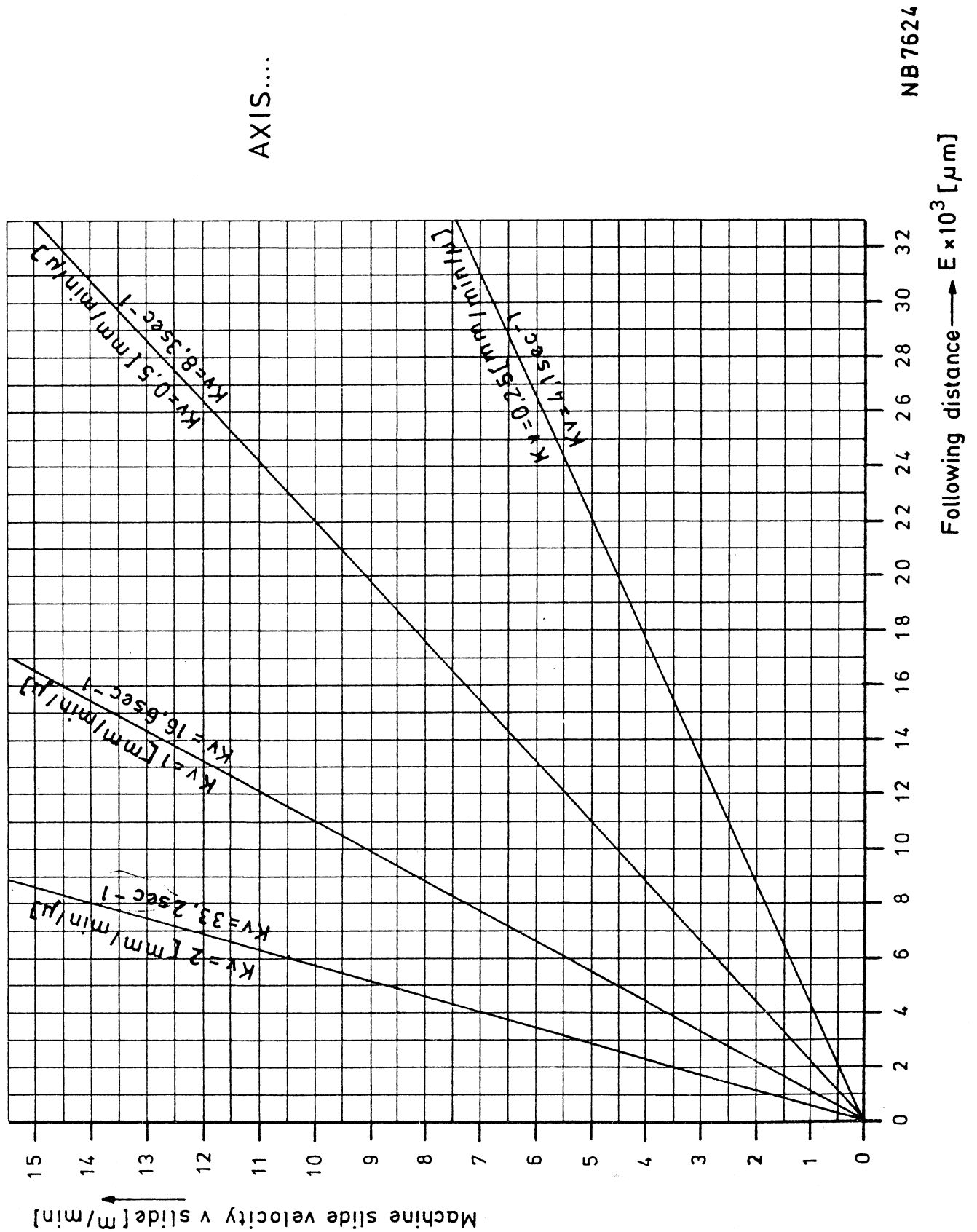
Fig. 12.1.-3. 2nd plotting sheet for determination of servo drive gain

AXIS.....

NB 7624

Following distance ⟶ E × 10³ [μm]

Machine slide velocity v slide [m/min]
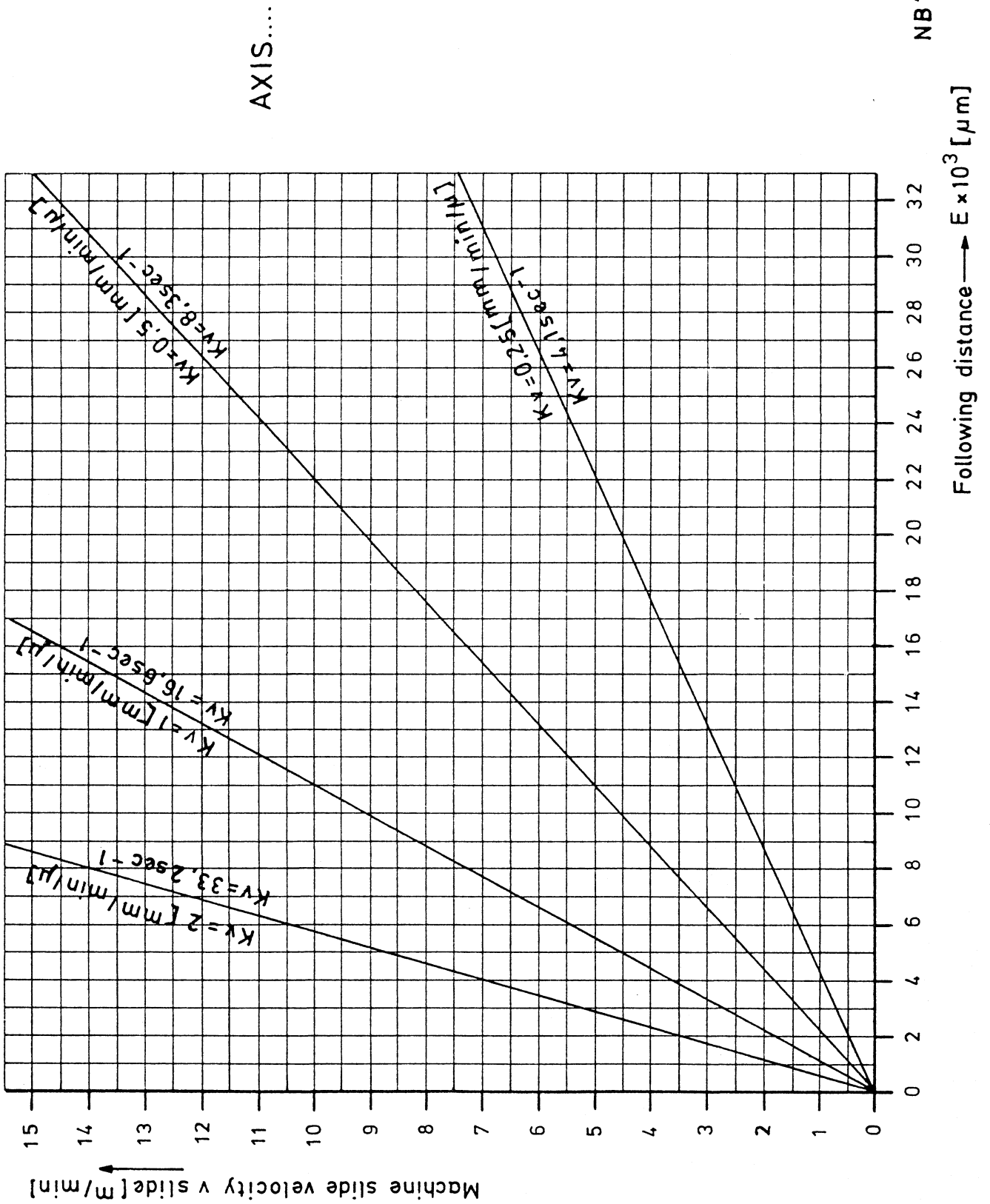
Fig. 12.1.-4. 3rd plotting sheet for determination of servo drive gain

*   **Preliminary values for MC 230, MC 280, MC 330, MC 380, MC 430, MC 480**

These machine constants allow the direction of approach of the Area switched to be defined.
Assign the value +1 for an approach in the positive direction of the axis, and the value -1 for an approach in the negative direction.

*   **Preliminary values for MC 231, MC 281, MC 331, MC 381, MC 431, MC 481**

The values assigned to these machine constants should be half the values assigned to MC 205, 255, 305, 355, 405, 455.

*   **Preliminary values for MC 232, MC 282, MC 332, MC 382, MC 432, MC 482**

The values assigned to these machine constants should be 1% of the values assigned to MC 231, 281, 331, 381, 431, 481.

*   **Preliminary values for MC 235/236, MC 285/286, MC 335/336 MC 385/386, MC 435/436, MC 485/486**

These machine constants allow the positions of the software limit switches to be defined. MC 235/236 apply to axis 1, MC 285/286 to axis 2, etc.
Proceed as follows: (see fig. 12.1.-5)

Measure the distance d1 between reference point and position of the negative emergency switch and the distance d2 between reference point and position of the positive emergency switch.
Allow for a safety clearance, depending on the accuracy of the measurements.

Assign the value found for d3 (d1 minus safety clearance) to
MC 235 (or MC 285, 335, 385, 435, 485, depending on the axis) and the value found for d4 (d2 minus safety clearance) to MC 236 (or MC 286, 336, 386, 436, 486, depending on the axis).
The values are to be entered as increments with sign.

Example:

(One increment is 0.001 mm)

d1 = 500 mm
d2 = 100 mm
d3 = 500 mm - 20 mm (safety clearance) = -480,000 increments
d4 = 100 mm - 20 mm (safety clearance) = + 80,000 increments

Note: The overall working range of the machine tool slide may not exceed 20 m. This means that |d3| + |d4| may never be in excess of 20,000,000 increments.

Emergency Limit Swith

Emergency Limit Swith

Reference Point

NB6869

Safety Area

Safety Area
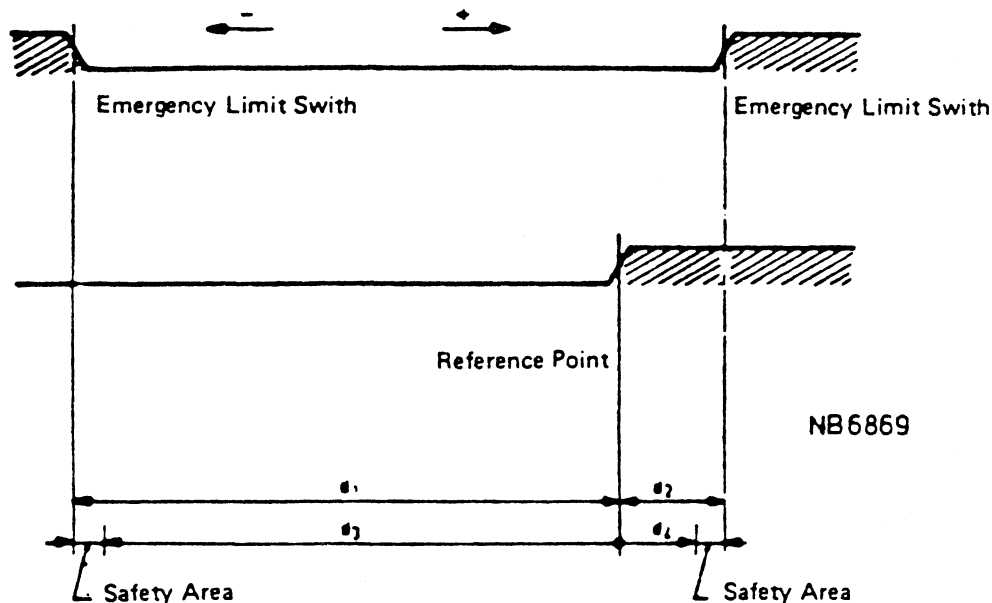
Fig. 12.1.-5. Determination of the software limit switch positions

12.2. INITIAL OPERATION OF THE MACHINE TOOL SLIDES

Make sure that the slides are at the centre of their traversing ranges. Enable the slide motions one at a time.
Mind the slides might move without being controlled!
If this should occur, the movement will be inhibited within a brief delay as a result of an error detection by the CNC (axis error 04 or 07). In this case, the counting direction for the axis in question must be reversed. This is done by reversing the sign assigned to the relevant machine constant, viz. MC 202, 252, 302, 352, 402, 452, depending on the axis concerned.

When all the slides keep their positions after being enabled, jogging operations may be carried out.

Allow a feed override of 20% using the appropriate key on the control panel of the CNC (see Operation Manual).

Now press the +X button

A movement at low feed rate along the X-axis will occur.
After releasing the +X button the movement should stop.

Should the feed rate increase, even after releasing the +X button the counting direction is incorrect and must be reversed, as described previously.
Allow a feed override of 100% and observe the response of the slide. A slow response or no response at all when the +X button is again pressed signifies that the values assigned to the machine constants for the gain of the servo system of the X-axis are too high (see chapter 12.1.) and need to be changed.

For the other axis proceed in the same way.
After completiuon of the adjustment procedure, the slides are to be moved towards the reference points of the respective axes.
Example for the X-axis:

- press the button ENTER

On the screen X-RP will be displayed.

- press the start button

A movement along the X-axis towards the Area switch will be effected. If the direction of motion is not correct, the value assigned to machine constant MC 230 must be reversed.
The procedure is repeated for the other axes.

When the reference points of all axes have been found, the software limit switches become effective. The CNC is now ready for operation.

Next, the final values for all machine constants employed must be determined. Detailed information is given in section 14.

13.   AXES ADJUSMENTS.

13.1.   Optimizing the machine constants for the servo drives.

In section 12 provisional values have been determined for the machine constants which define the gain of the servo system.
This section gives a practical method to obtain the correct MC values.
As example the machine constants for the first axis are used, the same procedure can be followed for the other axes.
The provisional values are supposed to be entered as indicated in section 12!

- Enter in MC206 the value of MC205
- Enter 32000 in MC216 and MC217
- Enter in MC81 the value 1 or 3
- If value 1 is assigned to MC81 the reference point finding procedure must be followed
- If value 2 is assigned to MC81 the software end switches are not operative! The CNC assumes that the position of the endswitches are related to the position at which the axis was reset to zero
- Select a feed override of 100% In jogging mode the axis will move at rapid traverse rate.
- Jog the axis.
  Under DIST TO GO the 'following distance' is displayed on the screen.

There are two possibilities now:

a - The following distance continues to increase.
b - The following distance slowly increases, then remains stable after some time.

Case a:
If the following distance keeps building up, or if error code x04 appears, the gain is too high.
Enter into MC215 a value that is 10% higher than the actual value. Recheck the following distance, if necessary, increase the value in MC215 again by 10%, etc. Repeat this until the following distance is stable during the rapid traverse.

Case b:
If the machine tool behaves slowly, for instance if the following distance builds up slowly and remains stable after a while the gain can be too low.
Enter into MC215 a value that is 10% lower than the actual value.

Recheck the machine tool behaviour.
Repeat lowering the value in MC215 until during jogging error x04 appears or until the axis starts overshooting. (Overshoot can be detected from the following distance display; as soon as the axis comes to a standstill the sign changes, which means that the axis is hunting.)
Now enter a value in MC215 being 10% higher than the present value. This is the correct value for MC215.

- Find the value of MC215 on the horizontal of the graph shown in figure 13.1.1. Draw a vertical line through this point. Find the value of MC205 on the vertical of the graph (MC205 is in 100/incr/min, graph in m/min!).
  Draw a horizontal line through this point.
  Draw a line from the origin (point0) through the intersection point of the horizontal and vertical lines.
  This line represents the Kv of the servo system.

Figure 13.1.-2 shows an example. The value for MC215 was 13000. The maximum slide velocity is 6 m/min. Line [a] represents the Kv of the servo system.

The established gain may satisfy the servo system under all circumstances. However most slides need a higher gain factor at the machining velocities. This can be checked by lowering the feed override until the slide moves with the machining velocity.

> There are two possibilities now:
> 1 - The slide behaves satisfactorily.
> 2 - The slide responds too slowly.

Possibility 1:

- Observe the slide behaviour. If it is satisfactory (the Kv is usually high), the second gain factor and hence the nod point will not be required.
  MC216 and MC217 become the same value as MC215.

Possibility 2:

- If the slide responce is not satisfactory, for instance the slide approaches the position too slowly, a second gain factor must be introduced.

Procedure:

- Enter the previously found value of MC215 in MC217.
- Enter the maximum machining velocity into MC206.
- Enter a value into MC215 a value being 10% lower than the present value.
- Set the feed override to 140% and jog the axis.
  Check the following distance and lower the MC215- value until overshoot occurs.
- Enter into MC215 a value being 10% higher than the last entered value.
  This is the final value for MC215.
- Find in the graph of figure 13.1.-1 the corresponding value (MC215) on the horizontal and draw a vertical line through this point.
- Draw a line from the origin (point 0) through the intersection point of the new vertical line and the existing horizontal line.
  This newly found line represents the new gain.
- Draw a horizontal line representing the maximum machining velocity + 10%.
- Draw a vertical line through the intersection point of the highest Kv and the last drawn horizontal line.
  The vertical line indicates the nod point to be entered in MC216 (value of the horizontal * 1000).

The graph shown in figure 13.1.-3 gives an example.
The values found for MC215, 216 and 217 are 6000, 1800 and 13000 respectively.
Line [a] represents the first gain factor, line [b] the second gain factor.
The gain factor is changed at point [N].

AXIS.....

Following distance ⟶ E ×10³ [μm]

Kv=0.5[mm/min] ⁻¹
Kv=8,3sec⁻¹

Kv=0.25[mm/min] ⁻¹
Kv=4,1sec⁻¹

Kv=1[mm/min] ⁻¹
Kv=16,6sec⁻¹

Kv=2 [mm/min] ⁻¹
Kv=33,2sec⁻¹

Machine slide velocity v slide [m/min] ⟵
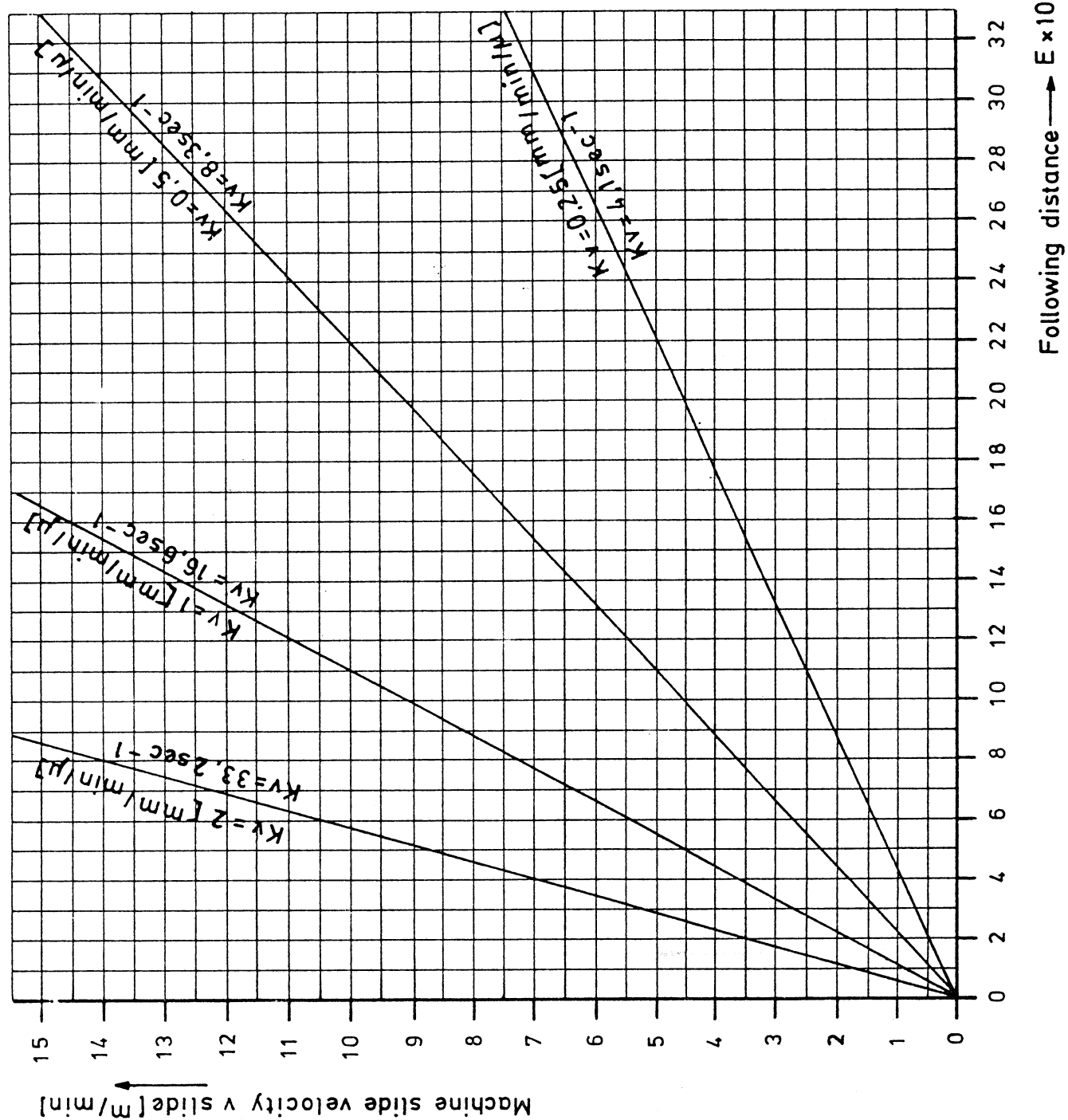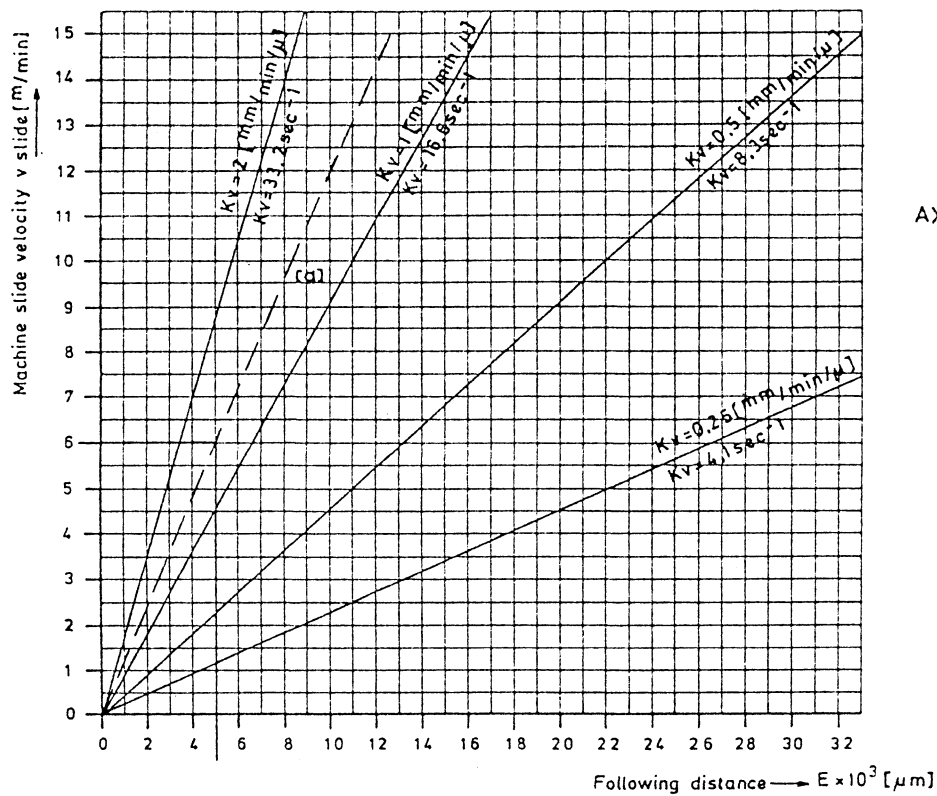
Fig. 13.1.-1. Kv-graph.

Fig. 13.1.-2. Example of Kv-graph with one gain factor.
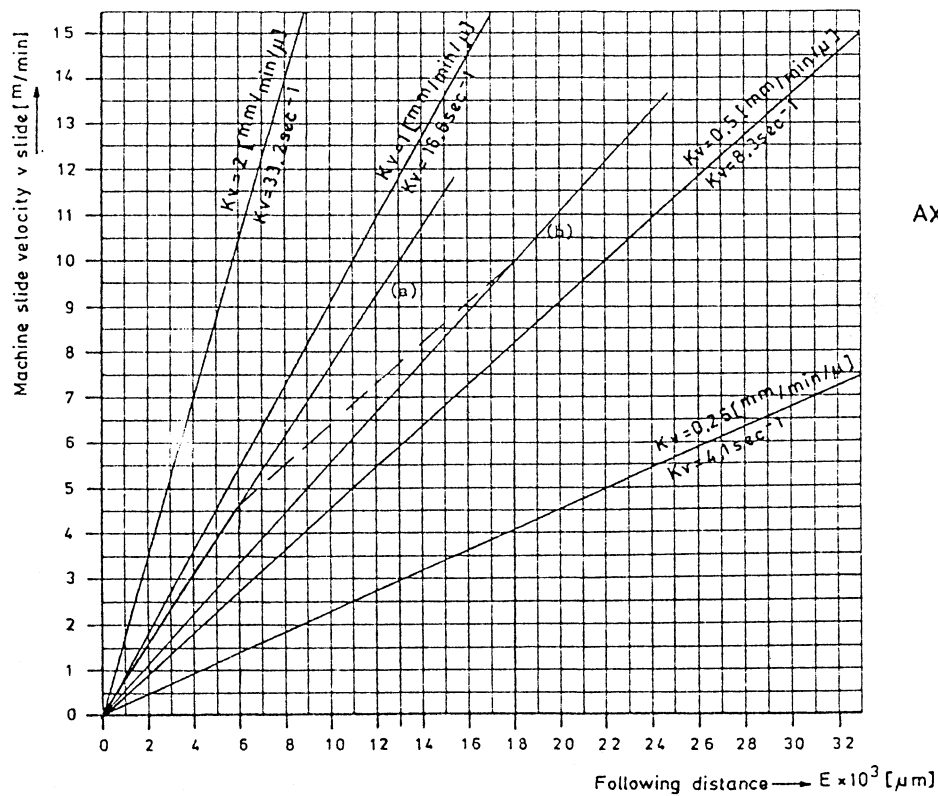
Fig. 13.1.-3. Example of Kv-graph with two gain factors.

13.2 Adjusting acceleration/deceleration.

If the two gain factors do not satisfy the requirements of the machine tool, a more sophisticated servo drive control can be activated.

Using the acc/dec servo control the servo drive output signal will not change abruptly but according to an e- factor curve.

MC218 activates the acc/dec feature.

Note that selecting the acc/dec function also in feed movements may have a negative effect on circular movements (oval shape). To avoid this care must be taken to obtain equal Kv's for the participating axes.

The curves of first and second order are given in figure 13.2.-1.

NB8192
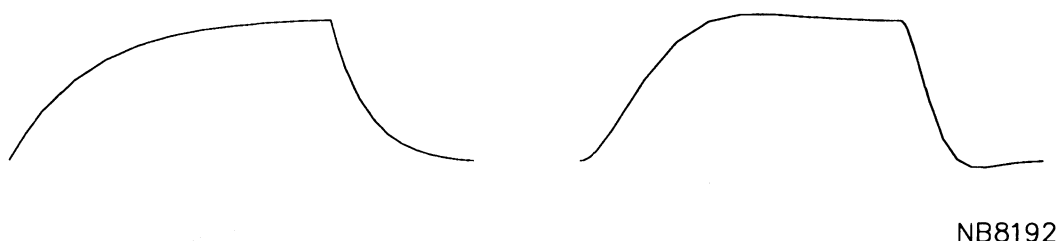
Figure 13.2.-1. First and second order acc/dec.

Note that second order acc/dec improves the slow down behaviour of the slide, however during starting and stopping the axis will be slow. in case this gives problems a threshold can be defined with MC220. Below this threshold the acc/dec function is switched off and the normal servo gain is used, see figure 13.2.- 2.
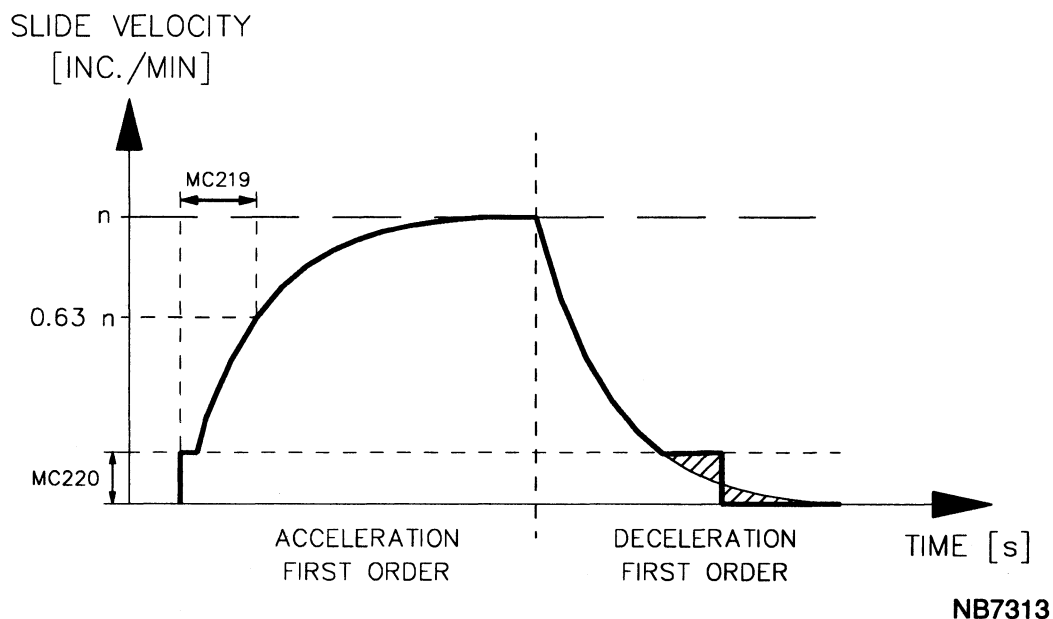
Figure 13.2.-2. Acc/dec threshold.

Establishing the acc/dec values in an empirical way.

- Usually only one gain factor is used. In this case the gain factor for the machining movements is used. Enter the value of MC215 also in MC216 and MC217.
- Enter in MC206 the rapid traverse rate.
- Jog the axis at 140% feedrate.
This will produce a trailing error related error message.
- Assign 200 to MC219.
- Jog the axis again.
If the error code still appears, increase the value.
If no error appears, decrease the value in order to find the optimum value for MC219.

13.3 Axes-compensation.

The axes-compensation can compensate linear as wel as cyclic deflections of the tool-path against the measuring-system-input.

The compensation is defined by a linear and/or cyclic compensation-curve for every axes. The curve gives the difference between the actual slide position of the axes and the measured position found by the measuring-system.
The axes-compensation for an axis is activated after reference- point-search, and will be active until the machine is initialized again.

A compensation-curve consists of at most 27 line-segments. These line-segments are defined by 28 points on the slide and the compensation-values for these points. The points are: the base- points (maximum 25), the reference-point and the start- and end- point of the curve. The start- and end-point are for linear compensation the software endswitches negative and positive. The start- and end-points are for cyclic compensation the negative and positive pitch-limit which has to be defined by machine constants.

When a slide reaches a compensation-point, the compensation-value is added to, or substracted from, the measured position at that point, depending on the direction of the movement.

Remark:
- The addition of the compensation-values must not exceed 200 increments.
- Maximum compensation is 1 increment per movement of 100 increments.
- Only the base-points between the start- and end-point of the curve are taken into account.
- The compensation-value of a base-point defined on the reference-point is not taken into account.
- When two base-point are defined on the same position, only the compensation-value of the first one defined is taken into account.
- The compensation-value of the start- or end-point is the same as the compensation-value of the base-point next to it.

13.3.1 Machine constants concerning axes-compensation

The function axes-compensation is activated by machine constant MC88. This machine constant also defines which type of compensation must be activated.
The following value can be assigned:

        0 for no axes-compensation
        1 for only linear axes-compensation
        2 for only cyclic axes-compensation
        3 for both linear and cyclic axes-compensation

If the value of machine constant 88 is not equal to zero, the following machine constants will be activated after disabling "EDIT MC". Of course this depends on the number of axes that are active and the number of base-points that are defined.

Value MC 88	Machine constants activation
1	226, 276, 326, 376, 426, 476, 1200-1499.
2	227, 243, 244, 277, 293, 294, 327, 343, 344, 377, 393, 394, 427, 443, 444, 477, 493, 494, 1500-1799.
3	226, 227, 243, 244, 276, 277, 293, 294, 326, 327, 343, 344, 376, 377, 393, 394, 426, 427, 443, 444, 476, 477, 493, 494, 1200-1799.

MC226, 276, 326, 376, 426 and 476 define the number of base- points needed to define the linear compensation-curve.

MC227, 277, 327, 377, 427 and 477 define the number of base- points needed to define the cyclic compensation-curve.

MC243, 293, 343, 393, 443 and 493 define the negative pitch-limit of the cyclic compensation-curve.

MC244, 294, 344, 394, 444 and 494 define the positive pitch-limit of the cyclic compensation-curve.

The even numbers between 1200 and 1499 define the positions of the base-points with respect to the reference point for the linear axes-compensation.

The odd numbers between 1200 and 1499 define the compensation- value of the base-points for the linear axes-compensation.

The even numbers between 1500 and 1799 define the positions of the base-points with respect to the reference point for the cyclic axes-compensation.

The odd numbers between 1500 and 1799 define the compensation- value of the base-points for the cyclic axes-compensation.

13.3.2. The compensation-curve

The compensation-curve can be found by substracting the actual position from the measured position over the total span of the axis. The measured position is the position that is found by the CNC, so the position found by the measuring-system. The curve of this measured position will be an ideal curve.
The actual position is the position found by a so called reference-measuring-system. This reference-measuring-system is mostly a laser-measuring-system.
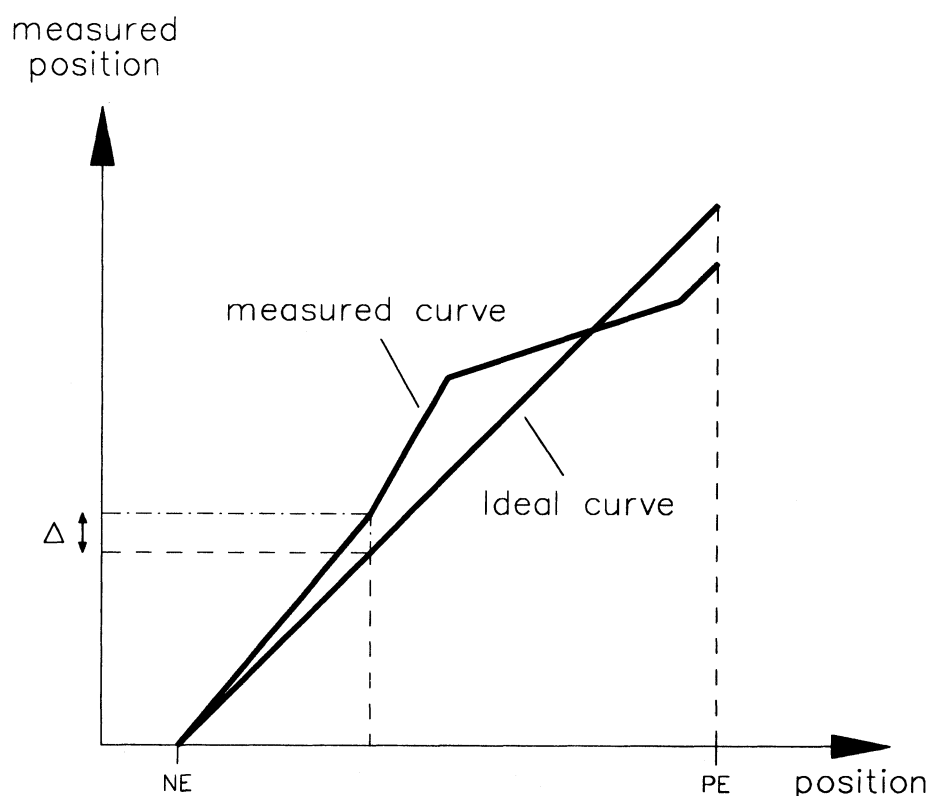
NB8182

Fig. 13.3.2.-1. The commanded and measured position over the axis

By substracting the actual position from the ideal position we get the compensation-curve.
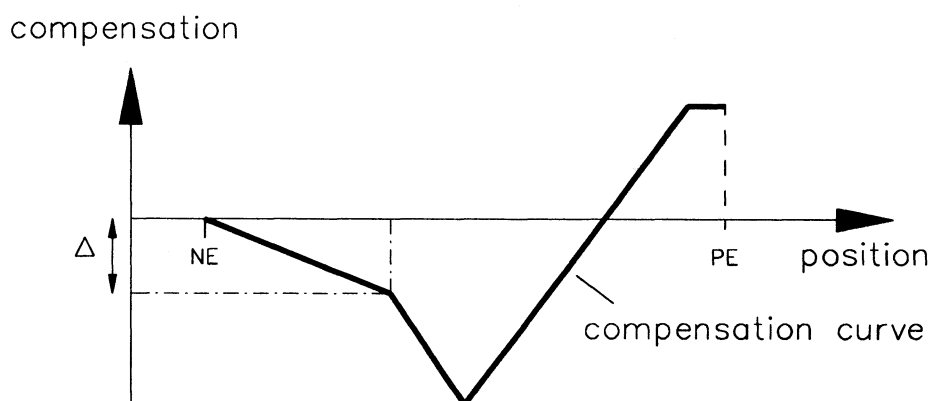
NB8183

Fig. 13.3.2.-2. The compensation-curve

This compensation-curve must be estimated by line-segments. The intersection-points of these line-segments are the base-points to be defined.

Most of the time the compensation-curve will be only linear or only cyclic. But when a linear axis is fitted with a rotary measuring-system, it might be possible to subdivide the compensation-curve into a linear compensation-curve and a cyclic compensation-curve.
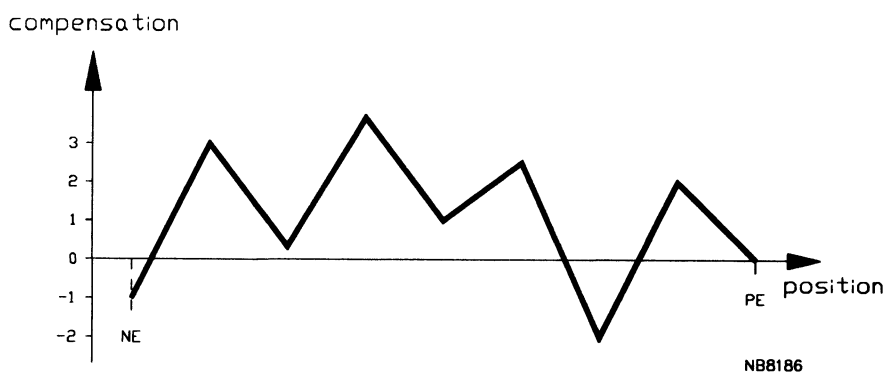
Fig. 13.3.2.-3 Compensation-curve with linear and cyclic component

After subdividing this compensation-curve the linear and cyclic compensation-curve will look like this:
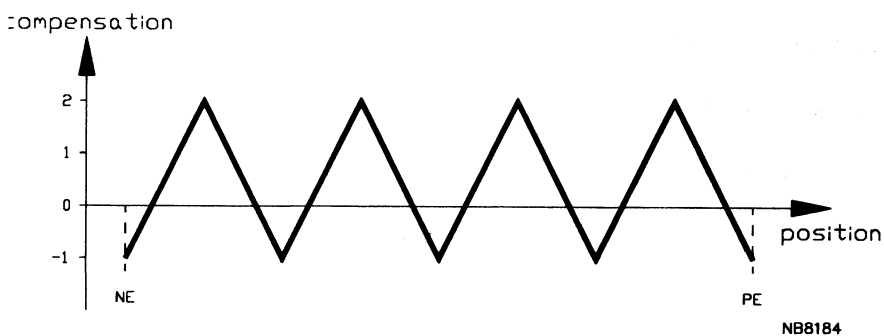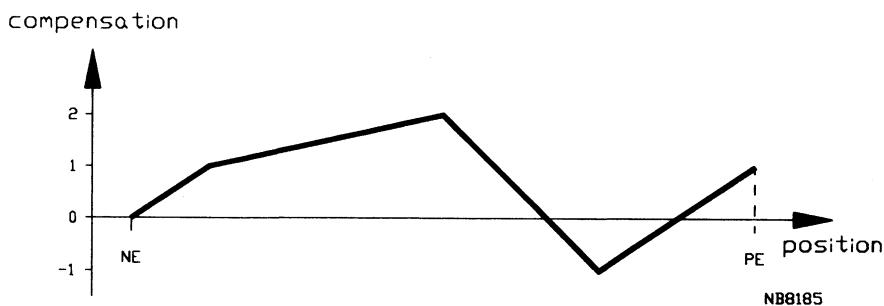
Fig. 13.3.2.-4. Linear and cyclic compensation-curve subdivided

13.3.3. Entering the linear axes-compensation

1 - Take the compensation-curve and divide it into line-segments.

compensation

NB8187
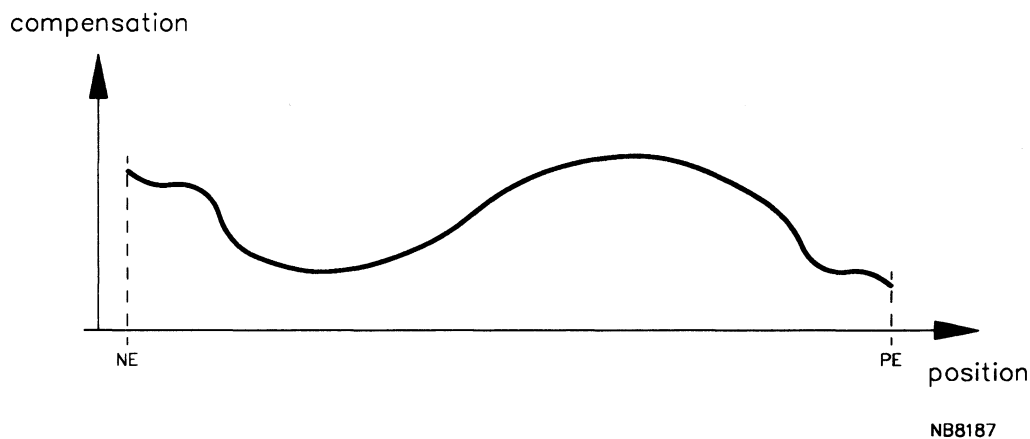
Fig. 13.3.3.-1. Compensation-curve for a linear axis

compensation

NB8188
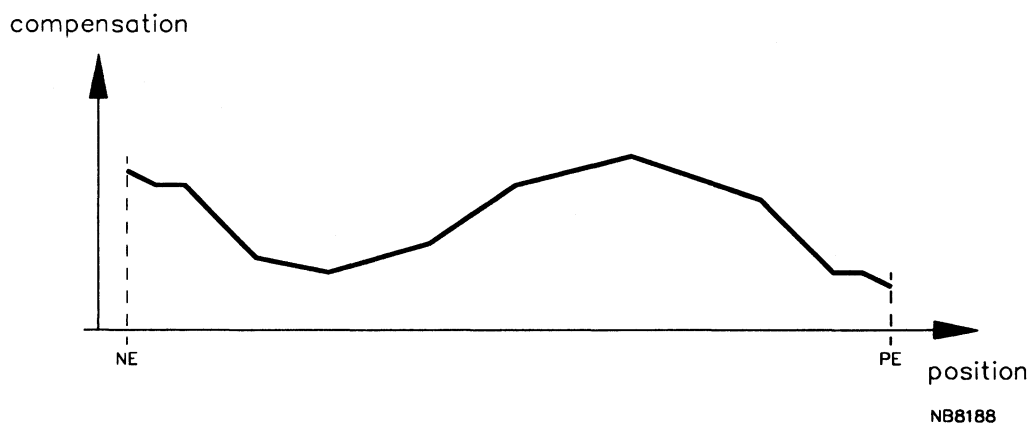
Fig. 13.3.3.-2. Compensation-curve divided into line-segments

2 - Define the base-points which are the intersection-points of the line-segments.

3 - Also define the software limit switches, the reference-point and the machine zero-point

4 - The positions of the base-points are noted as B1 to B8.

NSE = negative software endswitch

PSE = positive software endswitch

NB8189
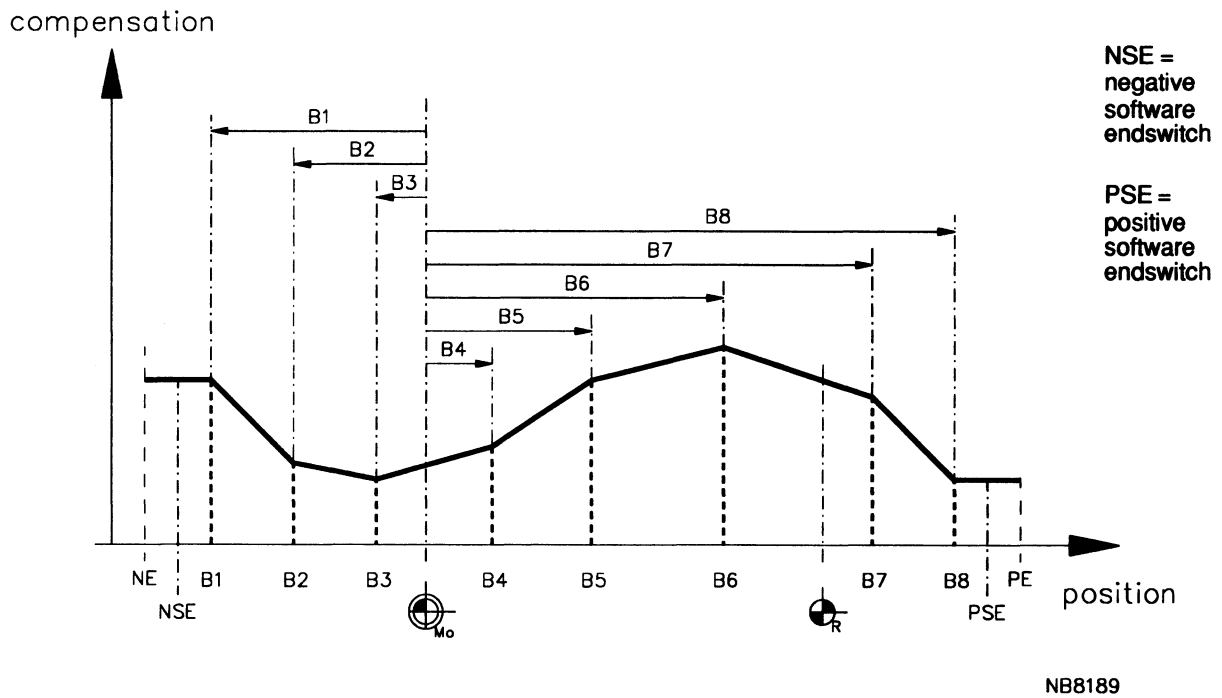
Fig. 13.3.3.-3. Base-points defined in compensation-curve

5 - Draw a horizontal line through the intersection-point of the reference-position and the compensation-curve. This line is the zero compensation-line.

6 - Define the compensation-values of the base-points with respect to this zero compensation-line and note them as V1 to V8.

NSE = negative software endswitch

PSE = positive software endswitch

NB8190
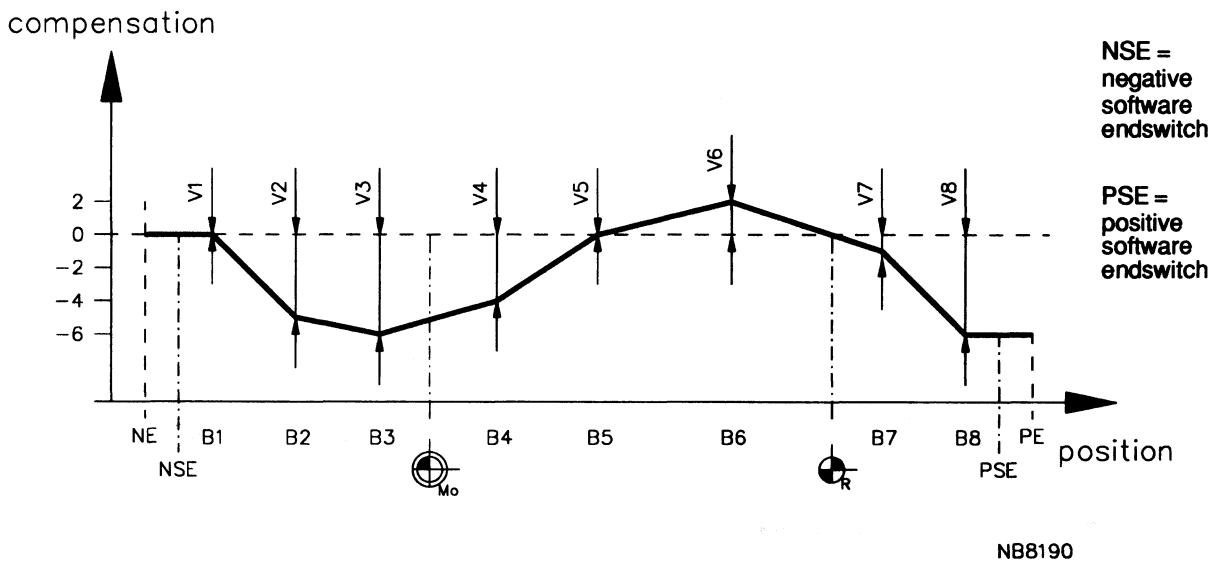
Fig. 13.3.3.-4. Compensation-values defined in compensation-curve

7 - Note the compensation-value per base-point.

P1 = (B1, V1)   P2 = (B2, V2)
P3 = (B3, V3)   P4 = (B4, V4)
P5 = (B5, V5)   P6 = (B6, V6)
P7 = (B7, V7)   P8 = (B8, V8)

8 - Put all the values in machine-constants.

For instance for the first axis:

MC 88 = 1   (only linear compensation)

MC 226 = 8   (8 base-points)

MC 1200 = B1(position of base-point for P1)
MC 1201 = V1(compensation-value P1)
MC 1202 = B2
MC 1203 = V2
MC 1204 = B3
MC 1205 = V3
MC 1206 = B4
MC 1207 = V4
MC 1208 = B5
MC 1209 = V5
MC 1210 = B6
MC 1211 = V6
MC 1212 = B7
MC 1213 = V7
MC 1214 = B8
MC 1215 = V8

13.3.4. Entering the cyclic axes-compensation

The way of finding the base-points and the compensation-values is the same as for linear axes-compensation with exeption of the following points:

1 - First the cycle-time or period of the cyclic compensation must be found.

2 - The position of the reference-point must be within this period. The startpoint of the period is called the negative limit-pitch, and the endpoint is called the positive limit- pitch. These two values are entered in two machine-constants for every axis.

AXIS	NEGATIVE LIMIT-PITCH	POSITIVE LIMIT-PITCH
1	243	244
2	293	294
3	343	344
4	393	394
5	443	444
6	493	494
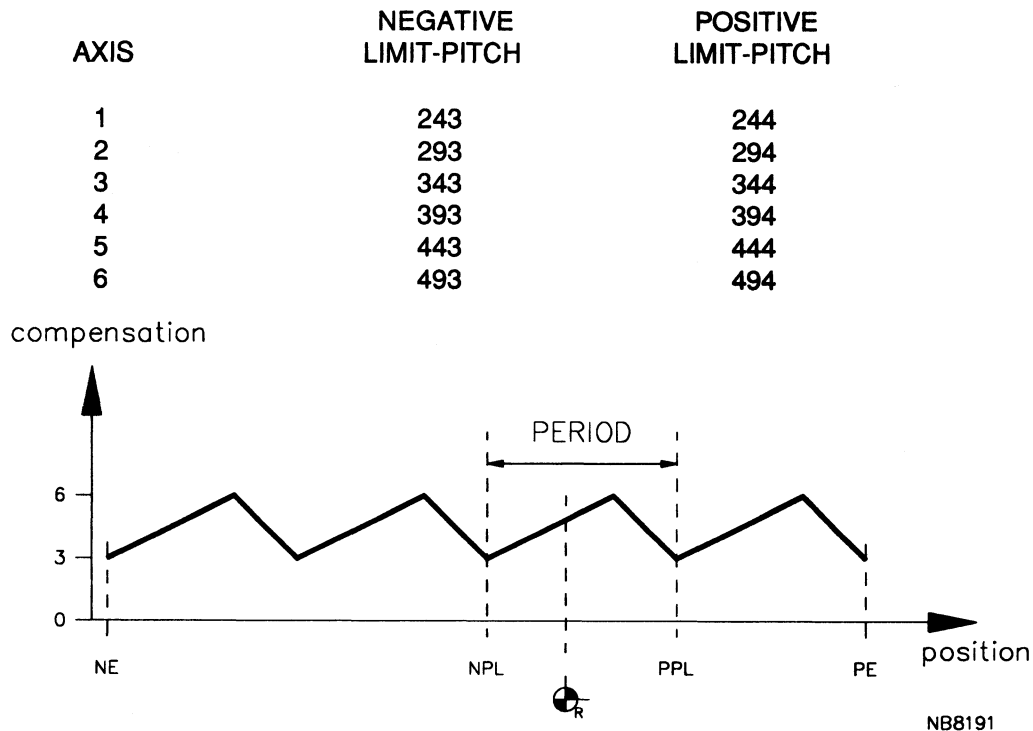
Fig. 13.3.4.-1. Finding the cycle-time in the compensation-curve

3 - After the values of the negative and positive limit-pitch are found, the base-points and compensation-values can be found the same way as stated in the previous chapter, called: Entering the linear axes-compensation.

13.3.5. Error codes

A number of axes-errors are reserved for axes-compensation- errors. These errors are all class-H errors. The axes- compensation for the axis that causes the error is switched off.

Note that in the axis error-codes below 'n' will be the axis designation, that is X, Y, Z, A, B or C.

n10 The addition of all the linear compensation-values is more than 200.

n11 At one or more positions the linear compensation is more than 1 increment per 100 increments movement.

n12 The addition of all the cyclic compensation-values is more than 200.

n13 At one ore more positions the cyclic compensation is more than 1 increment per 100 increments movement.

13.4. Dynamic Following Distance Guarding

The CNC checks the following distance for reaching a value entered in the following distance 1 or 2 (MC 215, MC 217) + 10%.
Error code X04 is then displayed and the axes movements are stopped. Under certain circumstances (e.g. during accelleration or during slow movements) this error-check may take too much time to prevent a hazardous situation for the machine tool.

The Dynamic Following Distance Guarding checks the machine tool behaviour more accurately. This is done by checking whether the following distance is inside a narrow field around the axis-characteristics.

Before activating the Dynamic Following Distance Guarding, the servo-system of the axis must be correctly adjusted. Now the Machine Slide Characteristic has to be found. After this the machine constants for the gain of the servo-system may be changed to find the threshold-factors. Machine slide behaviour changements however will be detected by the Dynamic Following Distance Guarding, and will result in an error.

Adjusting the dynamic following distance guarding is done in three steps:

    1.-Measuring of the dynamic following error characteristic (MC210).
       Start values of MC210=0, MC211=0, MC212=0, MC213=0. See section 13.4.1.

    2.-Setting Thresholds for G0 and G1 (MC211 and MC212).
       Start values of MC211=25, MC212=25, MC213=0. See section 13.4.2.

    3.-Setting Threshold decrease factor (MC213).
       Start value of MC213=2. See section 13.4.2.

13.4.1. Setting the Machine Slide Characteristic

The Dynamic following distance guarding is set by means of four machine constants: MC 210, MC 211, MC 212 and MC 213.

Before establishing the real machine slide characteristics make sure that:

- The machine is provided with its nominal workpiece-load
- The position loop gain (MC 215, 216 and 217) is set correctly
- The Acc/Dec function (if applicable) is set correctly (MC 218, 219 and 220)
- The Threshold decrease factor (MC 213) is set to "0"

After setting machine constants 215, 216 and 217 the machine slide characteristic may look like this:

NB7376
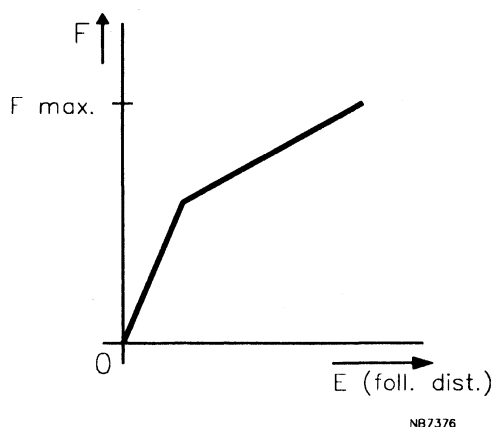
Fig. 13.4.1.-1. Machine slide characteristics according to MC-setting

However the "real" machine slide characteristic may look like this:

NB7377
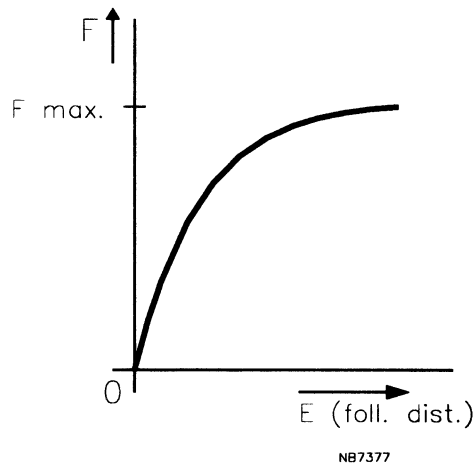
Fig. 13.4.1.-2. "Real" machine slide characteristic

To find the "real" machine slide characteristic proceed as follows:

- Select following distance display; set MC 81 to "1"
- Move the axis with a contineous feed
- Note the following distance display (displayed under >DISTANCE TO GO<) when a stable situation is obtained

Remark:If Acc/dec function is activated via MC218, it will take longer to obtain the stable situation.

To calculate the value for the Machine Slide Characteristics two formulas are possible:
If the feed was higher than the feed at which the second gain factor becomes active (Following distance greater than the value in MC 216) the following formula is used:

$$\text{Value MC 210} = \frac{F}{4000} \times \frac{MC215}{MC216 + \dfrac{MC215 - MC216}{MC217 - MC216} \times (E - MC216)}$$

If the displayed following distance is lower than the value in MC 216 use the following formula:

$$\text{Value MC 210} = \frac{F}{4000} \times \frac{MC215}{E}$$

Where   "F" is the Feed in increments per minute
        "E" is the displayed following distance in increments

After the value of the Machine Slide Characteristic is found, the values of the position loop gain may be changed. The value is directly related to the Kv factor of the servo loop.
Only when the drive amplifiers are readjusted, the value must be obtained again.

REMARK: - Do not use a feed lower than 100 mm/min to establish the value of MC 210.
          - Do not use a feed around the "nod-point". In this case it will not be clear which formula must be used.

13.4.2. Setting the following threshold factors.

Although the Machine Slide Characteristic approaches the exact servo characteristic very close, it will never be the same.
The servo loop characteristic may be as shown below:
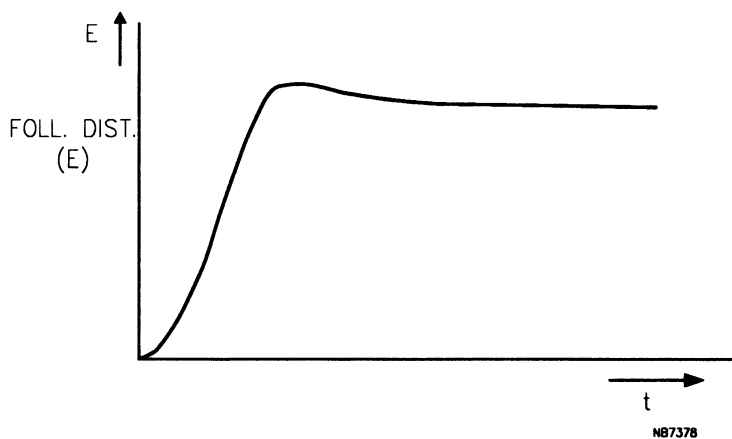
Fig. 13.4.2.-1. Actual servo loop characteristic

Around the Machine Slide Characteristic a narrow "band" is defined which encloses the actual servo loop characteristic.
Within this "band" no error will occur.
The upper and lower threshold of the band are symetrically spaced around the Machine Slide Characteristic.
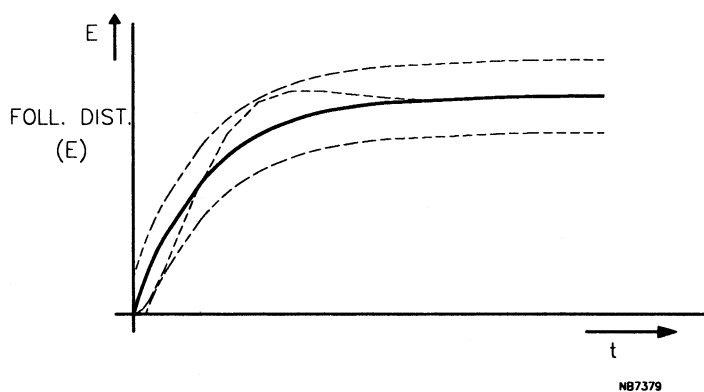
Fig. 13.4.2.-2. Upper and lower threshold

The CNC considers a band for positioning movements (G0, jogging and RPF) and for feed movements (G1, G2 and G3).
The threshold value is stored as a percentage of the following distance at rapid (for positioning movements) or at maximum feed i.e. MC740 (for feed movements).

$$\text{Threshold} = \frac{MC211}{100} \times E \ (G0) \quad \text{for positioning movements}$$

$$\text{Threshold} = \frac{MC212}{100} \times E \ (G1) \quad \text{for feed movements}$$

E is the displayed following distance in increments at rapid or feed.

Since it is difficult to measure the servo loop characteristic without the help of special tools such as a storage scope, it is recommended to find the threshold value by trial and error method.

Make sure the machine is provided with ist nominal workpiece- load. Start with a value of 25%. Move the axis at rapid for MC211 or maximum feed for MC212. If no error code appears, lower the value in steps of 2%, till an error code (X09) is displayed. Increase the value with 2%. This is the correct threshold value. If necessary increase the value with 2% till no error code is displayed. Do the latter also when at starting value already an error code is generated.

The differences between the Machine Slide Charracteristic and the servo loop charracteristic is biggest during starting and stopping the movement. This period is tree times the time- constant of the servo system. After this time a stable situation occurs, allowing to decrease the threshold value.
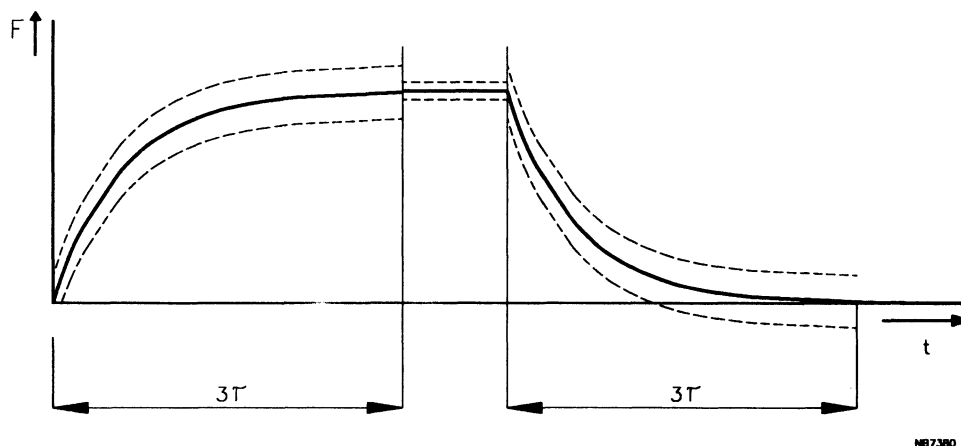
Fig. 13.4.2.-3. Decrease of threshold

The threshold decrease factor is stored in machine constant 213, in which 2, 4 or 8 can be stored. Entering another value results in no decrease factor at all.
Again by trial and error method the correct value can be found.

Remember that the use of the Acc/Dec function will result in delayed switching on the threshold decrease or not switching it on at all.

14. DESCRIPTION OF MACHINE CONSTANTS (MC's)

MC 0   NUMBER OF IO-CARDS (0=1,1=2,2=3,3=4,4=5,5=6)

Assign   0 when the CNC system is equipped with 1 I/O module
              1 when the CNC system is equipped with 2 I/O modules
              up to six I/O modules are possible

MC 1   NUMBER OF MEASURING INPUTS (0-8)

Including measuring-system inputs for spindle and handwheel.
Assign   0 for no measuring-system input
              1 for one measuring-system input
              2 for two measuring-system inputs
                  up to eight measuring-systems inputs are possible

MC 2   RAM MEMORY SIZE (1-2304 KBYTE)

The capacity of the RAM-memory is specified in the data-sheets supplied with the CNC system.
Enter the value in Kbytes.

MC 4   REMOTE CONTROL PANEL (0=OFF,1-5=ON)

Assign   0 for no Remote control panel
              1 for Remote control panel including safety switch
              2 for Remote control panel except safety switch
              3 not valid for this version
              4 not valid for this version
              5 for connection of an external feed override switch

MC 5   LANGUAGE (0=GER,1=ENG,2=FRE,3=DUT,4=ITA)

The CNC software comprises a language PROM, containing the text for support information in 2 languages.
The languages are specified in the data sheets.

Assign   0 for German support information
              1 for English support information
              2 for French support information
              3 for Dutch support information
              4 for Italian support information

MC 9   GRAPHICS MODULE (0=NO,1=2PL.,2=8PL.)

Assign   0 for no graphics module
              1 for a 2-planes graphics module
              2 for a 8-planes graphics modules

MC 10 NUMBER OF AXES (3-6)

Except spindle and **handwheel**. The minimum number of axes is three.
>   Assign   3 for three axes
>   4 for four axes
>   up to 6 axes possible

MC 11 PLANE INIT.(0=G17,1=G18,2=G19)

The plane that becomes active after power on.
>   Assign   0 for the G17 related plane
>   1 for the G18 related plane
>   2 for the G19 related plane

MC 13 COORD.SYSTEM INIT.(0=G180,2=G182)

The coordinate system that becomes active after power on.
>   Assign   0 for G180 related coordinate system
>   2 for G182 related coordinate system

MC 14 MEAS.SYST.RESOLUTION (70=INCH,71=METR)

The resolution (in metric units or in inches) of the measuring-system employed.
>   Assign   70 for a machine tool with Inch-type measuring-system
>   71 for a machine tool with metric-type measuring-system
>   All axes are required to have the same resolution.

MC 27 NUMBER OF TOOLS (0-255)

The size of the tool memory is defined. The maximum number of tools that can be stored when MC35 is set to 0 and 99 when MC35 is set to 1.
Part of the CNC memory will be reserved for the storage of the tool compensations.

MC 28 NR.OF TOOLPOSITIONS IN MAGAZINE (0-255)

The number of tool positions in the tool magazine is defined.

MC 29 TOOL LIFE MONITORING (0=OFF,1-2=ON)

A tool life can be assigned to every tool employed.
During machining operations, i.e. feed rate active (G1, G2, G3), spindle rotating (M3, M4), TXX <> T0, the actual and remaining tool life will be adapted in steps of 1 minute. When the remaining tool life becomes lower than 0, warning I11 is generated.
When at the tool change the tool life of the tool concerned is exceeded, either error code I12 is displayed or the spare tool is employed.
Error code I12 will also appear when the tool life of the spare tool is exceeded.
>   Assign   0 for function not active
>   1 for function active, display of error code
>   2 for function active, substitution by spare tool

For further details refer to the operator's manual.

MC 30   TOOL UNCLAMP (0=OFF, 1-2=ON)

Assign: 0   for unclamp not active
          1   for toggle button function
               Pushing the tool unclamp button inverses the state of the window marker signal.
          2   for constant button function
               When the button is depressed the window marker signal is high, releasing the button results in a low state of the window marker.

MC 31   CUTTING FORCE MONITOR (0=OFF,1-4=ON)

A cutting force monitor enables the cutting force of a tool to be checked.
Input signal "1st threshold exceeded" produces warning I13. At the next tool change with the same tool either error code I14 is displayed and the program stops or the spare tool employed is used. Input signal "2nd threshold exceeded" immediately produceserror code I15, causing feed and spindle speed to be inhibited.

Assign   0   for function not active
         1   for signal "1st threshold exceeded" producing an error message
         2   for signal "1st threshold exceeded" causing the spare tool to be used
         3   not valid for this version
         4   not valid for this version

MC 32   TOOL BREAKAGE MONITORING (0=OFF,1-2=ON)

By entering 1 the tool breakage monitoring feature is enabled.

MC 33   TOOL BREAKAGE TOLERANCE (0-999UNITS)

The permissible tolerance must be entered in measuring units of the monitoring device.

MC 34   TOOL EXCHANGE POSITION (0-255)

Not valid for this version, please enter 0.

MC 35   TOOL DATA OUTPUT (0-1)

Assign   0   for output of the programmed toolnumber (0-255) or the tool-position data (MC28 is not 0) in BCD code.
         1   for use of four decade tool programming. The lowest two decades are used for the toolnumber in the memory. Hence only 99 tools can be used. The maximum setting for MC27 is 99.

MC 37   TOOL MEMORY SELECTION (0=3,1-3=ID-NR)

Assign   0 for 3-decade tool number (FMS off)
         1 for 9-decade tool number (FMS1 on)
         2 for 9-decade tool number (FMS2 on)
         3 for 9-decade tool number (FMS+DNC)

MC 42 EXTERNAL PROGRAM CALL (0=OFF,1-2=ON)

Assign   0 for no external program call
               1 for external program call with fixed assignment
               2 for external program call with free assignment

For further details refer to the operating manual.

MC 43 NR.OF EXTERNAL PROGRAM CALLS (0-255)

Defines the amount of external program call numbers for the variable call memory. Part of the CNC memory will be reserved for the storage of external program calls.

MC 44 CONDITIONAL JUMP (0=ALWAYS,1=DEP.ON I.)

Assign   0 for jump commands always executed
               1 for execution of the jump commands depending on the status of the input marker WOX_ENB_PRG_JMP.

MC 51 ANALOG SPINDLE (0=OFF,1=ON)

Assign   0 for no analog spindle
1 for analog spindle
By switching of the machine constant enable signal and switching it on again, the relevant machine constants will be available now.

MC 78 M13/M14=SPINDLE,COOLING (0=OFF,1=ON)

Assign   0 M13/M14 is transferred to the IPLC without spindle action
               1 M13/M14 is converted to M3-M8/M4-M8 and transferred to the IPLC. The spindle action is executed by the CNC.

MC 79 SELECTED NEUTRAL GEAR (0=OFF)

Assign   0: M40 is tranfered to the IPLC without spindle action
               1: M40 is used as a neutral gear. If automatic range selection is active, programming S0 results in transferring M40 to the IPLC. If automatic range selection is not active programming M40 results in S0 and transferring M40 to the IPLC.

MC 80 SELECTION DEMO MODE (0=OFF,1=ON,2=IPLC)

Assign   0 for normal operation
               1 to activate DEMO-mode, IPLC not running
               2 to activate DEMO-mode, IPLC window active, no I/O's

Demo-mode means, that the NC system is operating without I/O-signals and without measuring-system signals. The program
execution is simulated in the NC-system. The servo outputs allow
an X/Y-recorder to be operated.

MC 81 DISPLAY MODE (0=NORMAL,1=FOL.DIST.,2-5)

Assign   0   for display of the distance to go (normal display)
             1   for display of the following distance
             2   for display of the distance to go; without reference point search after power on
             3   for display of the following distance; without reference point search after power on
             4   for display of the distance to go; without reference point search after power on; error codes are automatically reset after one second; UNCONTROLLED DISPLACEMENTS MAY OCCUR!
             5   for display of the following distance; without reference point search after power on; error codes are automatically reset after one second; UNCONTROLLED DISPLACEMENTS MAY OCCUR!

Note: Assigning 2, 3, 4, or 5 is only allowed for commissioning of the system and NEVER for normal operation, since THE SOFTWARE LIMIT SWITCHES ARE INOPERATIVE!

MC 82 NUMBER OF POINT DEFINITIONS (0-254)

Defines the amount of point definitions that can be stored.
Part of the CNC memory will be reserved for the storage of point definitions.

MC 83 NUMBER OF E-PARAMETERS (0-255)

Defines the amount of parameters that can be stored.
Part of the CNC memory will be reserved for the storage of parameters.

MC 84 PROGRAM DIRECTORY (0=WITH TEXT,1=INFO)

Assign   0   for listing with text
             1   for listing with information on memory requirements and program status

MC 85 MAX.NR.PART- AND SUBPROGRAMS (16-1000)

Defines the number of part- and subprograms that can be stored in the CNC memory.

Remark: After changing this machine constant and switching off the machine constant enable signal the memory of the CNC is re-initialised, which means that all the memories are cleared.

MC 87 TEMP.COMP.MEMORY LOCK (0=OFF,1=ON)

By entering 1 the temperature compensation memory is locked to protect it against unauthorized editing.

MC 88   MACH.COMP.(0=OFF,1=LIN,2=CYC,3=LIN+CYC)

The CNC allows spindle pitch error compensation.

Assign   0 for switching off error compensation
         1 for linear compensation only
         2 for cyclic compensation only
         3 for linear and cyclic compensation

After disabling machine constant editing and re-enabling machine constant edit-
ing the relative machine constants are activated.

MC 91   NUMBER AUXILIARY AXES (0-6)

The number of auxiliary axes are defined.

Assign   0 for no aux. axes
         1 for 1 aux. axis
         2 for 2 aux. axes
         up to six aux. axes are possible

Note that only the first two aux. axes can be defined as index-axes.

MC 93   BTR MACRO SIZE (4-2304 KBYTES)

Behind Tape Reader. When this function is activated (a softkey in Manual Block
Search) large programs are read in and executed at the same time. At start the
first part of the program is read in and stored in a macro. This macro is execut-
ed and at the same time another macro is created in which the next part of the
program is received.
If the first macro is completely executed the second macro is started, the first
macro is deleted, and the third part of the program is received in the third mac-
ro.
This is going on until the complete program is executed. The size of the macro is
defined by means of the number of bytes to receive.
The minimum size of a macro is 4 kbytes, and the maximum size of a macro is
2304 kbytes.

MC 100   AX1 LOOP (0=NO,1=CLOSED,2=OPEN,3=DEMO)

Assign   0 for axis not used
         1 for axis in closed loop
         2 for axis in open loop
         3 for axis in demo mode

Demo mode:
The testrun menu in AUTO and SINGLE mode is extended with testrun 3.
Under this testrun the axes which are NOT in DEMO mode are moved to the
programmed positions with the inputs and outputs switched off.

MC 102 AX1 ORIENTATION (1-3 LIN+,4-6 ROT+)

This code defines the axis orientation in a right-handed orthogonal system.
By means of this orientation e.g. the rotation direction of G02/G03 is estab-
lished for linear axes or the angle is calculated for the rotary axis in measuring
cycles.

The following configurations are allowed:
Maximum of 6 axes.
The minimum configuration can consist of 2 linear axes.
If more than 2 axes are used the **first three** axes must be linear.

E.g:
First 4 axes linear + 2 rotary axes is a valid combination
First 5 axes linear + 1 rotary axis is a valid combination
First 2 axes linear + 1 rotary axis is an **invalid** combination
First 3 axes linear + 1 rotary axis + 2 linear axes is a valid combination

The following values are possible:

1 = 1st linear axis in positive direction
2 = 2nd linear axis in positive direction
3 = 3rd linear axis in positive direction
4 = 1st rotary axis in positive direction
5 = 2nd rotary axis in positive direction
6 = 3rd rotary axis in positive direction

Note: A tracking axis or an independent axis can be assigned orientation code 1.

NB7069

Example of axis orientation.

MC 103 AX1 ASCII CODE (65=A-90=Z)

An axis is identified by the ASCII-value of an ASCII character.

Permissable characters are:

Character	ASCII-value	Character	ASCII-value
A	65	X	88
B	66	Y	89
C	67	Z	90
U	85		
V	86		
W	87		

MC 105 - MC108

These machine constants are equal to machine constants 100 - 103, but apply to the second axis.

MC 110 - MC113

These machine constants are equal to machine constants 100 - 103, but apply to the third axis.

MC 115 - MC118

These machine constants are equal to machine constants 100 - 103, but apply to the fourth axis.

MC 120 - MC123

These machine constants are equal to machine constants 100 - 103, but apply to the fifth axis.

MC 125 - MC128

These machine constants are equal to machine constants 100 - 103, but apply to the sixth axis.

MC 150   AX1 DISPLAY (0=NO,1-6=LINENR.)

Since there are only 6 lines available for the axis position display a selection must be made of the main axes and auxiliary axes to be displayed during "AUX-AX" display.

Assign   0 for no display of the first axis
             1 for the first axis to be displayed on the first line
             2 for the first axis to be displayed on the second line
               etc.

MC 151   AX1 JOG (1=HOR,2=VER,3=ROT)

When the first axis is displayed during "AUX-AX" display the axis can be jogged with softkeys.
The jog softkeys can be used in conjunction with the following text:

Assign   1 for horizontal axis;display: LEFT RIGHT
             2 for vertical axis;display: DOWN UP
             3 for rotary axis;display: BACKWFORW
             4 for rotary axis;display: CCWCW
             5 not valid for this version

MC 152 AX1 INCREMENT LEFT (1=POS,2=NEG)

When the first axis is displayed during "AUX-AX" display the axis can be jogged with softkeys.
Assign   1 for a positive axis movement when depressing the left softkey
Assign   2 for a negative axis movement when depressing the left softkey

MC 155- MC   157

These machine constants are equal to machine constants 150 - 152, but apply to the second axis.

MC 160- MC   162

These machine constants are equal to machine constants 150 - 152, but apply to the third axis.

MC 165- MC   167

These machine constants are equal to machine constants 150 - 152, but apply to the fourth axis.

MC 170- MC   172

These machine constants are equal to machine constants 150 - 152, but apply to the fifth axis.

MC 175- MC   177

These machine constants are equal to machine constants 150 - 152, but apply to the sixth axis.

MC 200 MEASURING SYSTEM INPUT SELECTION (0-8)

This parameter-block can be assigned to a measuring-system input and its servo-drive output. Be sure not to use the same input for an other axis, since this will result in unpredictable behaviour of the machine-tool.

Assign   0 for no measuring-system input
              1 for drive-card 1 socket X3
              2 " " 1 "X4
              3 " " 2 "X3
              4 " " 2 "X4
              5 " " 3 "X3
              6 " " 3 "X4
              7 " " 4 "X3
              8 " " 4 "X4

MC 202 COUNT DIRECTION MEASURING SYSTEM (-1,1)

This machine constant defines the count direction of the measuring system. Assigning 1 will give a positive count direction while moving in the orientation direction. Assigning -1 gives a negative count direction in the same direction of movement.

MC 203 MULTIPL.FACTOR S00/S90 (0=*1,1=*2,2=*4)

Assign   0 for 1 count-pulse decoded per S00/S90 cycle
            1 for 2 count-pulses decoded per S00/S90 cycle
            2 for 4 count-pulses decoded per S00/S90 cycle

When the axis is fitted with a linear measuring-system or when the axis does not exist, this machine-constant has no function.

MC 204 INVERT DRIVE-VOLTAGE (-1,1=NORMAL)

In case the drive output voltage must be inverted, change the sign of the value 1. (Note that a positive sign is obtained by entering 1 without sign).

MC 205 RAPID TRAVERSE RATE (1-320K*100INC/MIN)

The maximum traversing rate is stored in steps of 100 increments per minute.

Example: Required rapid traverse rate is 6 m/min. The value to be stored is 60,000.

MC 206 JOG FEED RATE (0-320000*100INC/MIN)

The feedrate during jogging at 100% feed-override is stored in steps of 100 increments per minute.

Example: Required jog feed rate is 3 m/min. The value to be stored is 30,000.

MC 210 DYN.FOLLOWING ERR. CONTROL (0=OFF-32000)

To obtain a dynamic following error guarding, the real machine slide servo characteristic must be defined. This is done with one single value.
The value is independent from the Kv of the servo loop.
How to find the value for this machine-constant see section on Dynamic Following Distance Guarding.

MC 211 DYN.FOLLOWING THRESHOLD G0 (0-100%)

The G0 following threshold is a percentage of the following distance at rapid.
The actual servo loop characteristic must be within the lower and upper threshold, otherwise an error is generated. The lower and upper threshold are symmetrically spaced around the machine slide characteristic.

MC 212 DYN.FOLLOWING THRESHOLD G1 (0-100%)

The G1 following threshold is a percentage of the following distance at maximum feed movements.
The actual servo loop characteristic must be within the lower and upper threshold, otherwise an error is generated. The lower and upper threshold are symmetrically spaced around the machine slide characteristic.

MC 213 THRESHOLD DECREASE FACTOR (0-8)

The difference between the machine slide characteristic and the slide servo loop characteristic is biggest during starting and stopping the movement. After three times the time constant of the servo system a stable situation occurs, allowing to decrease the threshold value. The threshold can be decreased by a factor 2, 4, or 8.

MC 215 FOLLOWING DISTANCE 1 (128-28500INC.)

Defines the position loop gain 1 of the servo drive. Gain 1 is effective until a iven nod point is attained. From this point on the lower gain 2 is activated.
Not the gain factor is stored, but the following distance at which the servo output voltage is maximum.
The following distance is stored in increments. The smaller the number of increments, the higher the gain.

MC 216 NOD POINT (0-28500INC.)

The point at which the gain factor changes, is stored in increments.

MC 217 FOLLOWING DISTANCE 2 (128-28500INC.)

Gain 2 becomes active after the nod point has been attained.
Not the gain factor is stored, but the following distance at which the servo output voltage is maximum is stored in increments. The smaller the number of increments, the higher thegain.

MC 218ACC/DEC MODE (0=NO,1-2=G0,3-4=G0+G1)

The standard gain control does not satisfy the requirements of all type of machine tools. Some machine tools need a more sophisticated servo drive control.
The CNC offers a so-called acceleration/deceleration (acc/dec) function for the servo drive outputs to meet the requirements of those machine tools.
Activating the acc/dec function the servo output is updated according to an e-factor curve instead of steps.
Assign   0 for no acc/dec
          1 for acc/dec first order only in rapid movements
          2 for acc/dec second order only in rapid movements
          3 for acc/dec first order in rapid and feed movements
          4 for acc/dec second order in rapid and feed movements

MC 219 ACC/DEC TIME CONSTANT (0-3840MS)

The acc/dec time constant is the time (in msec.) after which the servo drive output signal must reach 63% of the maximum output voltage following the e-curve.

MC 220 ACC/DEC THRESHOLD(0-1048575*100INC/MIN)

When the acc/dec function is used and the axis responds too slowly when starting and moving into position, a threshold value can be stored. Below this value the acc/dec is inoperative.
The value must be entered in steps of 40.

MC 221 INPOD DELAY TIME (0-255*15MS)

The "inpod" delay time makes sure that the machine slide is "in position" before the next command is executed.
The delay time is active only after positioning movements, and is started when the "in position"-window is reached.
The value is stored in steps of 15 msec.

Example: The delay time required is 100 msec. By assigning 7, a true delay time of 105 msec (7x15 msec) is achieved.

MC 222 IN POSITION WINDOW (0-32000INC.)

To define the distance before the command position, where the "inpod"-delay time must start. The distance is stored in increments.
Theoretically, the servo output voltage is 0 V at the end of the positioning movement. In praxis the servo output will show a small positive or negative offset value. Consequently, thefollowing distance display will show an offset value as well.
The "in position"-window must be slightly larger than the displayed following distance.

Example: If the displayed following distance is 4 increments, 10 has to be stored.

MC 223 STANDSTILL MONITORING (0-320000INC.)

The servo loops are permanently checked for errors by checking the following distance. When the following distance increases during standstill, the CNC checks whether the standstill monitoring value is exceeded. If so, an error message is produced.
When setting the number of increments, the drift of the servo system as well as the displacement in the axis as a result of milling operations are to be taken into account.

MC 224 BACKLASH COMPENSATION (0-32000INC.)

The backlash compensation will be activated at the first movement in the opposite direction. The backlash is stored in increments.
Backlash compensation is only employed with rotary measuring-systems. For linear measuring-systems 0 has to be assigned to this machine constant in order to prevent erratic starts of positioning movements.

MC 225 OVERSHOOT COMPENSATION (0-32000INC.)

The overshoot compensation will be activated at the first movement in the opposite direction. The overshoot is stored in increments.
Overshoot compensation is only employed with rotary measuring-systems. For linear measuring-systems 0 has to be assigned to this machine constant in order to prevent erratic endpoints of positioning movements.

MC 226 NR.OF LIN.COMPENSATION BASEPOINTS(0-25)

Defines the number of basepoints necessary to define the linear compensation curve. See also Machine Constants 1200 - 1249.

MC 227 NR.OF CYC.COMPENSATION BASEPOINTS(0-25)

Defines the number of basepoints necessary to define the cyclic compensation curve. See also Machine Constants 1500 - 1549.

MC 230 RPF DIRECTION APPROACH (-1=NEG,1=POS)

Assign   1 for approaching the "area switch" in positive direction
         -1 for approaching the "area switch" in negative direction

MC 231 RPF FEED RATE (0-320000*100INC/MIN)

Defines the velocity with which the "area switch" is to be approached.
The feedrate is stored in steps of 100 increments per minute.

Example: The required RPF feedrate is 2 m/min. The value to be stored is 20,000.

MC 232 RPF CREEP FEED (0-320000*100INC/MIN)

After the "area switch" is actuated the machine slide starts moving in the reversed direction for reference point search. The velocity is stored in steps of 100 increments per minute.

The usual velocity for reference point search is 160 mm/min. The maximum error related to the approaching velocity is:

velocity	value entered	maximum error
80 mm/min	800	0.5 um
160 mm/min	1600	1 um
330 mm/min	3300	2 um
500 mm/min	5000	3 um

MC 233 RPF OFFSET (INC.BETWEEN MZP AND RP)

The distance between the Machine Zero Point and the Reference Point is stored in increments.

MC 234 RPF AREA SWITCH (0=OFF,1=ON)

Assign   0 for RPF finding using marker signal only (E.g. for turntables)
           1 for RPF finding using area-switch and marker signal

MC 235 SOFTW.LIMIT SWITCH POSITIVE (INC.TO RP)

The position of the software limit switch in the positive or negative quadrant is defined. The sign indicates the quadrant, the value is the absolute position in increments with respect to the Machine Reference Point.

MC 236 SOFTW.LIMIT SWITCH NEGATIVE (INC.TO RP)

The position of the software limit switch in the opposite quadrant of MC 235 is defined. See MC 235.

MC 237 HOME POSITION 1 (INC.TO MZP)

The position of home position 1 is stored in increments. The sign indicates the quadrant, the value is the absolute position with respect to the Machine Zero Point.

MC 238 HOME POSITION 2 (INC.TO MZP)

The position of home position 2 is stored in increments. The sign indicates the quadrant, the value is the absolute position with respect to the Machine Zero Point.

MC 239 HOME POSITION 3 (INC.TO MZP)

The position of home position 1 is stored in increments. The sign indicates the quadrant, the value is the absolute position with respect to the Machine Zero Point.

MC 240 POS.FIXED MEASURING PROBE (INC.TO MZP)

The position of the fixed measuring probe is stored in increments. The sign indicates the quadrant, the value is the absolute value with respect to the Machine Zero Point.

MC 242 POSITION CALIBRATION RING (INC.TO MZP)

The position of the calibration ring centre is stored in increments. The sign indicates the quadrant, the value is the absolute value with respect to the Machine Zero Point.

MC 243 BEGIN PITCH CYCLIC COMP.(-999999999-0)

The position of the start point of the pitch of the cyclic compensation with respect to the Machine Reference Point is entered in increments.
The start point is always in the negative quadrant.

MC 244 END PITCH CYCLIC COMP.(0 - 999999999)

The position of the end point of the pitch of the cyclic compensation with respect to the Machine Reference Point is entered in increments. The start point is always in the positive quadrant.

MC 245 HOME POSITION 4 (INC.TO MZP)

The position of home position 4 is stored in increments. The sign indicates the quadrant, the value is the absolute position with respect to the Machine Zero Point.

MC 246 HOME POSITION 5 (INC.TO MZP)

The position of home position 5 is stored in increments. The sign indicates the quadrant, the value is the absolute position with respect to the Machine Zero Point.

MC 250 - MC296

These machine constants are equal to machine constants 200 - 246, but apply to the second axis.

MC 300 - MC346

These machine constants are equal to machine constants 200 - 246, but apply to the third axis.

MC 350 - MC396

These machine constants are equal to machine constants 200 - 246, but apply to the fourth axis.

MC 400 - MC446

These machine constants are equal to machine constants 200 - 246, but apply to the fifth axis.

MC 450 - MC496

These machine constants are equal to machine constants 200 - 246, but apply to the sixth axis.

MC 500 SPINDLE MEAS.SYSTEM INPUT (0=NO,1-8)

The analog output voltage for the spindle drive is available at connector X1 of the relevant measuring-system.

Assign   0 for no analog spindle output
1 for drive-card 1 socket X1 pins 4 and 8
2 for drive-card 1 socket X1 pins 2 and 6
3 for drive-card 2 socket X1 pins 4 and 8
4 for drive-card 2 socket X1 pins 2 and 6
5 for drive-card 3 socket X1 pins 4 and 8
6 for drive-card 3 socket X1 pins 2 and 6
7 for drive-card 4 socket X1 pins 4 and 8
8 for drive-card 4 socket X1 pins 2 and 6

MC 501 RESOLUTION SPINDLE TRANSDUCER (0-65000)

The number of S00/S90 pulses per revolution of the spindle,generated   by the transducer, is stored.
The frequency at the input of the measuring-system may not exceed 100 kHz. A transducer generating 65000 S00/S90 pulses perrotation of the spindle will therefor give a maximum spindle speed of 90 rev/min approximately.
When no spindle transducer is used, the value 0 has to be assigned. The displayed value under "ACTUAL S" is now the calculated value.

MC 502 DIRECTION SPINDLE MEAS.SYSTEM (-1,1)

The sign defines the count-direction of the spindle transducer.
A positive sign results in a positive count-direction at M3, and a negative sign results in a positive count-direction at M4.

MC 503 SPINDLE MULTIP.S00/S90 (0=*1,1=*2,2=*4)

Assign   0 for 1 count-pulse decoded per S00/S90 cycle
1 for 2 count-pulses decoded per S00/S90 cycle
2 for 4 count-pulses decoded per S00/S90 cycle

MC 505 SPINDLE JOG SPEED (0 - 9999REV/MIN)

This machine constant applies only to NC-systems fitted with the "Analog spindle" option.
The number of spindle revolutions per minute when operating the Jog button is in the range 0 to +9999.

MC 515 GEAR 1:FOLL.DISTANCE 1 (128-320000 INC)

Defines the position loop gain 1 for the first (lowest) gear range of the spindle drive. Gain 1 is effective until a given nod point is attained. From this point on the lower gain 2 is activated.
Not the gain factor is stored, but the following distance at which the servo output voltage is maximum.
The following distance is stored in increments. The smaller the number of increments, the higher the gain.

MC 516 GEAR 1: NOD POINT (0-320000 INC.)

The point at which the gain factor for the first gear-range changes is stored in increments.

MC 517 GEAR 1:FOLL.DISTANCE 2 (128-320000 INC)

Gain 2 for the first gear-range becomes active after the nod point has been attained.
Not the gain factor is stored, but the following distance at which the servo output voltage is maximum is stored in increments. The smaller the number of increments, the higher thegain.

MC 518 GEAR 1:ACC/DEC MODE (0=OFF,1-4)

In case the standard gain control does not satisfy the requirements of the spindle the acceleration/deceleration (acc/dec) function can be switched on.
Activating the acc/dec function the servo output for the spindle is updated according to an e-factor curve instead of steps.
Assign   0 for no acc/dec
          1 for acc/dec first order in rapid movements
          2 for acc/dec second order in rapid movements
          3 for acc/dec first order in rapid and feed movements
          4 for acc/dec second order in rapid and feed movements

MC 519 ACC/DEC TIME CONSTANT (15-1340MS)

The acc/dec time constant is the time (in msec.) after which the spindle servo drive output signal must reach 63% of the maximum output voltage following the e-curve.

MC 520 ACC/DEC THRESHOLD(0-1048575*100INC/MIN)

When the acc/dec function is used and the spindle responds too slowly when starting and moving into position, a the threshold value can be stored. Below this value the acc/dec is not active.

MC 521 INPOD DELAY TIME (0-255*15MS)

The "inpod" delay time makes sure that the spindle is "in position" before the next command is executed.
The delay time is effected only after positioning movements, and is started when the in position"-window is reached.
The value is stored in steps of 15 msec.

Example: The delay time required is 100 msec. By assigning 7, a true delay time of 105 msec (7x15 msec) is achieved.

MC 522 IN POSITION WINDOW (0-32000INC.)

To define the distance before the command position, where the "inpod"-delay time must start. The distance is stored in increments.
Theoretically, the servo output voltage is 0 V at the end of the positioning movement. In praxis the servo output will show a small positive or negative offset value. Consequently, thefollowing distance display will show an offset value as well.
The "in position"-window must be slightly larger than the displayed following distance.

Example: If the displayed following distance is 4 increments, 10 has to be stored.

MC 523 STANDSTILL MONITORING (0-320000INC.)

The servo loops are permanently checked for errors by checking the following distance. When the following distance increases during standstill, the NC system checks whether the standstill monitoring is exceeded. If so, an error message is produced.

MC 525 - MC533

These machine constants are equal to machine constants 515 to 523, but apply to gear range 2.

MC 535 - MC543

These machine constants are equal to machine constants 515 to 523, but apply to gear range 3.

MC 545 - MC553

These machine constants are equal to machine constants 515 to 523, but apply to gear range 4.

MC 560 M19:MODE (0/1=MECH.,2/3=NC)

The function "oriented spindle stop" (M19) is used to bring and keep the spindle in an accurate angular position.

Assign   0/1M19 mechanically. M19 is written in the display and output via the IPLC window. The execution is effected by the machine interface.
2/3 M19 with transducer. The execution is effected by the CNC.

MC 565 M19:SEARCH SPEED (+/-1000REV/MIN)

The search speed of the spindle after spindle stop (M5) is stored in revolutions per minute.
By storing a number between 0 and 362, a clockwise spindle rotation (M3) is obtained. Storing a number between -362 and 0 will produce a counter-clockwise spindle rotation (M4).

MC 566 M19:MARKEROFFSET 1(+/-360000*.001DEGR.)

The offset of the M19 angular position to the point where the marker pulse appears is stored in steps of 1/1000 degree.

MC 568 M19:WITH D-ADDRESS (0=OFF,1=ON)

Assign   0 for inactive D-address during execution of M19
            1 for activating D-address programming during execution of M19. The actual offset value for the angular M19 position is calculated as: Offset value = value assigned to MC 566 + value programmed under D-address.

MC 570 AUTO SPINDLE RANGE SELECT.(0=OFF,1=ON)

Assign   0 for no automatic range selection, open loop
            1 for automatic range selection, open loop

MC 571 MAX.SPEED RANGE 1 (+/-9999 REV/MIN)

The maximum spindle speed for the first range is stored in revolutions per minute with sign. A positive sign causes Analog-Sto be positive with M3 and negative with M4, a negative sign to be negative with M3 and positive with M4.

MC 572 MAX.SPEED RANGE 2 (+/-9999 REV/MIN)

The maximum spindle speed for the second range is stored in revolutions per minute with sign. A positive sign causes Analog-S to be positive with M3 and negative with M4, a negative sign to be negative with M3 and positive with M4.

MC 573 MAX.SPEED RANGE 3 (+/-9999 REV/MIN)

The maximum spindle speed for the third range is stored in evolutions per minute with sign. A positive sign causes Analog-S to be positive with M3 and negative with M4, a negative sign to be negative with M3 and positive with M4.

MC 574 MAX.SPEED RANGE 4 (+/-9999 REV/MIN)

The maximum spindle speed for the fourth range is stored in revolutions per minute with sign. A positive sign causes Analog-S to be positive with M3 and negative with M4, a negative sign to be negative with M3 and positive with M4.

MC 575 MAX.SPEED RANGE G84 (+/-9999 REV/MIN)

The maximum spindle speed allowed during tapping cycle G84 is stored in revolutions per minute with sign. A positive sign causes Analog-S to be positive with M3 and negative with M4, a negative sign to be negative with M3 and positive with M4.

MC 580 MAX.ANALOG OUTPUT VOLT.M41 (+/-10000MV)

The maximum output voltage applied to the spindle drive is stored in mV without sign. When the maximum permissible voltage at the input of the spindle drive is 8 V, 8000 has to be stored.

MC 581 MIN.ANALOG OUTPUT VOLTAGE (+/-10000MV)

The minimum voltage required to obtain the appropriate motor response for spindle is stored in mV.

MC 582 GEAR CHANGE OUTPUT VOLTAGE (+/-10000MV)

The required voltage to obtain the necessary spindle motorrotation   for   gear changing is stored in mV.

MC 583 MAX.ANALOG OUTPUT VOLT.M42 (+/-10000MV)

The maximum output voltage applied to the spindle drive is stored in mV without sign. When the maximum permissible voltage at the input of the spindle drive is 8 V, 8000 has to be stored.

MC 584 MAX.ANALOG OUTPUT VOLT.M43 (+/-10000MV)

The maximum output voltage applied to the spindle drive is stored in mV without sign. When the maximum permissible voltage at the input of the spindle drive is 8 V, 8000 has to be stored.

MC 585 MAX.ANALOG OUTPUT VOLT.M44 (+/-10000MV)

The maximum output voltage applied to the spindle drive is stored in mV without sign. When the maximum permissible voltage at the input of the spindle drive is 8 V, 8000 has to be stored.

MC 586 DELAY SPINDLE ANALOG OUTP.(0-255*10MS)

At feed/speed hold the spindle is stopped after a delay time to ensure that the axes have come to a standstill before the spindle rotation is stopped.
The delay time is entered in steps of 10 msec.

MC 588 SPEED FOR NACT<NMIN (0-100 %)

The signal N<NMIN is present at the IPLC window as soon as the actual calculated spindle speed comes below the value defined.
The spindle speed is entered in rev./min.
Note that a spindle transducer is required to measure the actual spindle speed.

MC 589 SPEED PERCENTAGE FOR NACT = NCMC (0,1-100%)

The signal NACT=NCMND is present at the IPLC window as soon as the actual calculated spindle speed equals the commanded spindle speed +/- the range defined.The maximum speed of the actual gear range is defined by MC571-574. The range can be entered between 1 and 100% of the actual speed.
Assigning 0 means that the signal NACT=NCMD is always true.
Note that a spindle transducer is required to measure the actual spindle speed.

MC 590 SPINDLE CLOCK PHASE (0-255*15MS)

The time between two rotation pulses is stored in steps of 50msec. Storing a zero gives a non alternating gear change output voltage. See also MC 591.

MC 591 SPINDLE CLOCK IMPULSE (0-255*15MS)

For gear changing it can be necessary to have the spindle moving alternately with M3 and M4. To do so the CNC outputs the gear change voltage alternately positive and negative with an active duration defined by this machine constant. The duration is stored in steps of 50 msec.
Storing a zero gives a non alternating gear change output voltage.

MC 599 SPINDLE DELAY TIME BEFORE RPF (*100 MS)

MC 705 DECIMAL POINT COORDINATES (0-4)

The position of the decimal point in the axis addresses is given by the smallest number of the programmed increments.

Example:

Resolution	DisplayValue	assigned to MC
0.001 mm	0000.000	3
0.0001 inch	000.0000	4

MC 706 DECIMAL POINT FEED RATE (0-4)

Display	Dimension	value MC
000000		
	0.001 mm/min	0
00000.0		
	0.010 mm/min	1
0000.00		
	0.100 mm/min	2
000.000		
	1.000 mm/min	3
00.0000		

10.000 mm/min   4

MC 707 PROGR.START-UP MODE (70=INCH,71=METRIC)

Assign   70 for programming in inches
           71 for programming in metric units

MC 710 INTERMEDIATE CIRCLE (0-999999999 INC.)

Not used in this software version, please enter 0.

MC 711 INTERSECTION ANGLE G41,G42 (0-180DEGR.)

When a program contains a radius correction, the intersection angle determines whether an intersection is calculated or an intermediate block with a circular movement is inserted, to round off the angle.
The intermediate block is executed as soon as the angle is smaller than the number of degrees assigned, in steps of 1°

MC 712 CIRCULAR END POINT WINDOW XYZ (0-999999999 INC.)

The inaccuracy allowed for a programmed circle end point or of an incorrectly programmed circle centre point is stored in increments.

MC 714 SCALING MODE (0+2=FACTOR,1+3=%,2+3=3D)

Scaling can be programmed by a factor or by a percentage.
Besides, it can be established whether scaling is only executed in the machining plane (e.g. G17 in the X/Y-plane) or in all linear axes.

Value MC	Scaling in	programmed by a
0	machining plane	factor
1	machining plane	percentage
2	all linear axes	factor
3	all linear axes	percentage

The number of decimals for scaling depends on the scaling mode.
When scaling has been programmed by a factor, up to six decimals are possible. When scaling has been programmed by a percentage, up to four decimals are possible.

MC 715 DECIMAL POINT SCALING (0-6)

Assign   0 for no decimal position
           1 for one decimal position
              up to four when using percentage scaling
              up to six when using factor scaling

MC 720 OVERLAP POCKETMILLING (0-100%)

The overlap during pocket milling (G87 and G89) is defined as a percentage of the cutting tool diameter. In case no I-address is programmed the percentage (between 0 and 100) assigned to this mc is used.

MC 723 DECELERATION DISTANCE G84 (0-9999999999 INC.)

The spindle speed and the feed rate are reduced in a linear way over the deceleration distance during tapping cycles. At restart, the spindle speed and the feed rate are accelerated over the same distance. By this, tool breakage at "Cycle Interrupt" is avoided.
The value assigned can be overruled by a programmed I-address in the G84-cycle.

MC 724 DWELL DURING TAPPING G84 (0-10000*15MS)

After spindle stop during tapping cycles (G84) and drilling cycles (G81), a dwell time can be defined. The dwell is stored in periods of 15 msec.

MC 725 TAPPING G84 KV-FACTOR (0-999999999*10^-5)

During a tapping cycle a following distance will be build-up. To compensate this following distance an intermediate motion with the spindle at standstill can be made (compensation distance). This movement takes place at the start of the retract movement.

The compensation distance is proportional to the feed of the tool axis at the beginning of the tapping movement.

This proportionality factor is expressed with the following formula:

$$MC725 = \frac{\text{Compensation distance}_{(mm)}}{\text{Tap feed}_{(mm/min)}} * 10^5$$

The machine constant value can be found empirical, using a starting value of 200 (Kv=1, no acc/dec).

Note that this feature is activated only when a floating tapholder is used. Enter 0 to switch off the feature.

MC 726 G84 RETRAC RATE (0-MAX * 100 INC/MIN)

Defines the feed rate for the compensation movement over the distance defined by MC725. The maximum value is determined by the maximum feed rate allowed in the relevant axis.

MC 731 INPUTS OPENLOOP (0=OFF,1=ON)

Assign   0 : inputs open/closed servo loop are set inactive.
           The servo loop will always be closed.
        1 : inputs open/closed servo loop active
           A low voltage at an input concerned means servo loop closed, a high voltage means servo loop opened.

MC 740 MAX.FEED (1-320000*100INC/MIN)

The maximum allowed operating feed rate is stored in steps of 100 increments per minute.

MC 741 TESTRUN FEEDRATE (1-320000*100INC/MIN)

The feed rate during testrun is stored in steps of 100 increments per minute.

MC 744 FEED LIMITATION (1-20*15MS)

The feed is always calculated per sample in a way that the feed per sample is equal for the total movement.
Due to the fact that the minimum number of samples per movement is 2, there can be a problem with small movements.
To prevent stuttering of the slide due to variation of the feed output, the minimum number of samples per program block can be set higher. But, the higher the value, the slower the slide will move when small steps are programmed.
The maximum number of samples per program block is 20.

MC 745 FEEDOVERRIDE MAX.(100-150%)

The maximum percentage of feed override that can be set by the feed override buttons is stored in steps of 5%, between 100 and 150 percent.

MC 746 FEEDOVERRIDE MIN.(0-50%)

The minimum percentage of feed override that can be set by the feed override buttons is stored in steps of 5%, between 0 and 50 percent.

MC 747 SPEEDOVERRIDE MAX.(100-150%)

The maximum percentage of speed override that can be set by the speed override buttons is stored in steps of 5%, between 100 and 150 percent.

MC 748 SPEEDOVERRIDE MIN.(50-100%)

The minimum percentage of speed override that can be set by the speed override buttons is stored in steps of 5%, between 50 and 100 percent.

MC 750 FEED HANDWHEEL (0-320000*100INC/MIN)

The feed rate when rotating the handwheel is stored in steps of 100 increments. Moving the handwheel fast or slow doesn't influence the feed rate.

MC 751 MEAS.SYSTEM INPUT HANDWHEEL (0-8)

Defines the plug to which the handwheel is connected to.

```
Assign   0 for no measuring-system input
         1 for drive-card 1 socket X3
         2 "        "     1   "    X4
         3 "        "     2   "    X3
         4 "        "     2   "    X4
         5 "        "     3   "    X3
         6 "        "     3   "    X4
         7 "        "     4   "    X3
         8 "        "     4   "    X4
```

MC 752 HANDWH.FACTOR S00/S90 (0=*1,1=*2,2=*4)

```
Assign   0 for 1 count-pulse decoded per S00/S90 cycle
         1 for 2 count-pulses decoded per S00/S90 cycle
         2 for 4 count-pulses decoded per S00/S90 cycle
```

MC 770 DIO/DNC INTERFACE SELECTION (0-5)

The control unit includes a V24 and a V11 interface for data transfer. Data transfer can be effected in the following modes:

DATA I/Odata transfer with an external (passive)
peripheral (X-ON/X-OFF or RTS/CTS protocoll).
DNC LOCAL/REMOTE
data transfer with a computer (LSV-2 protocoll).

It is recommended to use the V24-connection for DATA I/O operation and the V11-connection for DNC operation

V24-connection	V11-connectionValue	MC
DATA I/O	-	0
-	DATA I/O	1
DATA I/O	DNC	2
DNC	DATA I/O	3
MINI PC	-	4
-	MINI PC	5

MC 771 DIO DATA CARRIER (0=ASCII,1=ISO,2=EIA)

The code to be used for coding the data carrier information is given by the external data peripheral.

Assign   0 for data operation with ASCII-code
          1 for data operation with ISO-code
          2 for data operation with EIA-code

Note that in case automatic code recognition is switched off (MC 772=0 or 2) the CNC checks the coding during reading in according to the selected code in MC771.

MC 772 DIO AUTO CODE RECOGNITION (0=OFF 1=ON)

Enables the CNC to recognize the code sent by the data peripheral. The code is displayed and the data loaded, provided it is ASCII, ISO or EIA-code.
When the automatic code recognition is disabled, only data code in accordance with the code defined by MC771 will be read.

Assign   0 for no auto code recognition
          1 for auto code recognition

MC 773 DIO FLOWC.(0,3=RTS;1=XON-7W;2=XON-3W)

Assign   0 for handshake control using RTS/CTS
          1 for handshake using XON/XOFF and active RTS/CTS
          2 for handshake using XON/XOFF without RTS/CTS
          3 for handshake with RTS/CTS

MC 774 DIO LEADER/TRAILER LENGTH (0-120)

A great deal of data peripheral generate one or more characters after sign CTS has gone low. To prevent these characters from being ignored by the NC-system, the number of characters (minimum 5) that are to be read after signal CTS has gone low, is stored.

MC 775 RS232C (X1) NUMBER OF STOPBITS (1 OR 2)

The number of stopbits required by the external data peripheral.
Assign   1 for 1 stopbit
           2 for 2 stopbits

MC 776 RS232C (X1) BAUDRATE (110-19200)

The baudrate specified for the transmitter at peripheral side.
Minimum baudrate is 110, maximum baudrate 19200 baud.

MC 782 LSV-2 CHARACTERSET (0=ASCII,1=ISO)

Assign   0 for data operation with ASCII-code
           1 for data operation with ISO-code

MC 783 LSV-2 FLOWCONTROL (0=RTS,1=3WIRE)

Assign   0 for RTS/CTS active
           1 for RTS/CTS not active

MC 785 RS422 (X2) NUMBER OF STOPBITS (1 OR 2)

The number of stopbits required by the external data peripheral.
Assign   1 for 1 stopbit
           2 for 2 stopbits

MC 786 RS422 (X2) BAUDRATE (110-19200)

The baudrate for receiving and transmitting for the V11 interface may be from 110 to 19200 baud.

MC 788 MINI PC: %-PROT. IN FILE (0 OR 1)

Assign   0 for no additional information
           1 add %-protocol to each file

MC 791 LSV-2 DATAFIELD SIZE (80-120)

The maximum length of the telegrams (maximum length of useful data) under the LSV/2 protocol is defined.

MC 792 LSV-2 TIMEOUT PERIOD (0-128S)

After the NC-system has sent ENQ and after the transfer of the LSV/2 tele-
gram DLE STX ....DLE ETX BCC, the NC-system waits for an answer dur-
ing the defined time out for DNC (from 0 to 128 sec.). When no answer is re-
ceived within this time, ENQ is sent again.
Note that assigning 0 means that there is no time out.

MC 793 LSV-2 NR.OF REPEATS (0=NO LIMIT,1-12)

The number of ENQ-repeats after the time out for DNC may be from 0 to 12.
Note that assigning 0 means that the CNC keeps on repeating ENQ until inter-
rupted by hand.

MC 794 START-UP MODE (0=DIO,1=DNC)

The data in/out mode that becomes active after switching on the NC-system.

Assign   0 for DATA I/O
           1 for DNC LOCAL
Note that this MC is not present when MC 770 is assigned with a value smaller
than 2.

MC 795 LSV-2 DELAY TIME (0-128MS)

When using V24 or V11 interface without the handshake signals, a delay time
can be introduced between receiving and transmitting, and vice versa. This is to
enable the peripheral to switch over. The delay time is entered in msec between
0 and 128.

Note that entering a delay time delays the communication speed.

MC 796 DATA-IN SYNTAX CHK (0=ON,1=NO NRS, 2=OFF)

Part programs and macros can be checked during input. This causes a consider-
able reduction of speed.
There are two checks available:
- Double line numbers check
- Syntax check

The program check can be disabled to increase the speed of data processing.
Assign   0 for both syntax check and double line number check
           1 for only syntax check
           2 no checks

MC 797 DNC MODE (0=LOCAL,1=REMOTE,2=LOCAL EXT)

The selection of DNC-remote operation must be enabled.
Assign   0 : DNC local
              1 : DNC local and DNC-remote
              2 : DNC local extended mode (only data I/O)

To activate the remote operation of DNC, the following operations must be carried out:

- press the "MANUAL" button
- press the "AUTO" button
- press the "DATA I/O" button
- press the "MENU" button
- select option 3 (DNC REMOTE) by pressing the "3" button

Now the DNC-remote operation is activated, and the CNC must be controlled by a peripheral computer.

MC 798 DATA INPUT:CLEAR TM (0=ON,1=OFF)

Assign   0 to clear the tool memory when selected during Data I/O
              1 to leave the present tool memory contents unaffected during Data I/O. Only the tool information for the tools transferred during Data I/O is changed in the tool memory.

MC 799 DATA I/O FORMAT (0=N<9000,1=N>9000)

Assign   0 to be compatible with V500
              1 to allow large block numbers

This machine constant determines the whether or not IPLC files or any other DNC-file contains only one or multiple programs or macros. The constants only function is to provide full compatibility with previous versions of the CNC (V500 or older).
Under normal circumstances MC799 can be set to 1, this ensures maximum flexibility. The MC should be (temporarily) set to 0 for the following reasons:

- A papertape or cassette has to be made, which must be read by a V500 CNC.
- A file in the computer contains multiple part programs or macros.
- The CNC is connected to a PC running CDS 1.3.

Note that setting MC799 to 0 will not allow a partprogram or macro to contain block numbers above 8999.

MC 800 TEMP.COMP. NR.OF SENSORS (0=OFF,1-3=ON)

Assign   0 to disable temperature compensation
              1 to enable temperature compensation, using 1 sensor
              2 to enable temperature compensation, using 2 sensors
              3 to enable temperature compensation, using 3 sensors

MC 801 MAX.TEMPERATURE SENSOR A (0-999*.1DEG)

Define the maximum measured temperature allowed by the first sensor.

MC 802 MIN.TEMPERATURE SENSOR A (0-999*.1DEG)

Define the minimum measured temperature allowed by the first sensor.

MC 803 MAX.TEMPERATURE SENSOR B (0-999*.1DEG)

Define the maximum measured temperature allowed by the second sensor.

MC 804 MIN.TEMPERATURE SENSOR B (0-999*.1DEG)

Define the minimum measured temperature allowed by the second sensor.

MC 805 MAX.TEMPERATURE SENSOR C (0-999*.1DEG)

Define the maximum measured temperature allowed by the third sensor.

MC 806 MIN.TEMPERATURE SENSOR C (0-999*.1DEG)

Define the minimum measured temperature allowed by the third sensor.

MC 813 MAXIMUM COMPENSATION VALUE (0-999)

Define the maximum compensation value allowed, in increments.

MC 814 NUMBER OF TEMPERATURE VALUES (0-99)

Define the maximum number of temperature compensation values for each S-memory.

MC 815 TEMP.COMPENSATION TIME (0-65000*50MS)

Define the time interval with which the CNC compensates the measuring system information to compensate thermal displacement. Each time interval the compensation value is one micron (or 0.0001") so that the total compensation value is spread in time.

MC 840 MEAS.SYSTEM FOR MEASURING PROBE (0-8)

The measuring probe is connected to the input "AREA" of the measuring system defined in this MC. The probe may for instance be connected to the measuring system assigned to the electronic handwheel.
Assign 0 if no measuring probe is used.

Drive card	measuring system	connect to pin of socket X5	MC value
one	first	1	1
one	second	2	2
two	first	1	3
two	second	2	4
three	first	1	5
three	second	2	6
four	first	1	7
four	second	2	8

MC 841 MEAS.PROBE TYPE (0=IND,1/3=INFR,4=CABLE)

Assign   0 for an inductive measuring probe, fixed or moving.
Probe is switched on automatically, signal 'measuring probe ready' indicates start of measuring cycle.
1 for an infra-red measuring probe.
Infra-red measuring probe is switched on by making the output 'measuring probe active' low for 1 second. Correct functioning is indicated by 'measuring probe ready'. The probe is switched off automatically after a given time in the probe electronics.
3 for an infra-red measuring probe, type Renishaw MP7. See MC's 852, 853 and 854.
4 for a hard wired measuring probe, probe must indicate 'measuring probe ready' within 5 seconds after signal 'measuring probe active' is on the IPLC window.

MC 842 DURATION OF AIR SUPPLY (0-3840*100MS)

To clean the workpiece surface before measuring, a blower can be started by the output "AIR BLOWING MEASURING PROBE". The time required for cleaning the surface is stored in steps of 100 msec.

MC 843 MEAS.FEED RATE (0-999999999*100INC/MIN)

The feed rate used for the measuring probe to contact the workpiece is stored in steps of 100 incr/min.
For linear measuring-systems a feed rate of 0.14 m/min is recommended, and for rotary measuring-systems a feed rate of 1.2 m/min is recommended. In this way, the NC-system will attain an accuracy of 1 um. The overall accuracy depends on the machine tool.

MC 844 PRE DISTANCE MEAS-POINT(0-999999999INC)

The pre-measuring distance is the distance between the starting point of the slow measuring movement and the programmed measuring point. The distance is stored in increments.

Note:The movement from the starting point to the pre- measuring distance is performed at rapid traverse rate, simultaneously in all axes. When all axes are "In position", the slow measuring movement in the programmed axis is performed.

MC 845 POST DISTANCE MEAS-POINT(0-999999999INC)

The post-measuring distance is the distance between the programmed measuring point and the point where the measuring movement is switched off. The distance is stored in increments.
When the measuring movement is completed and no surface has been contacted, the movement is stopped. A return to the starting point of the movement is performed. At the same time an error message is generated.

MC 846 RESOLUTION ROTARY AXIS (INC/0.001DEGR)

The resolution in degrees per count pulse of the rotary table measuring-system is stored in increments per 1/1000 degree.
The resolution is required when the G50-function is to be executed with rotary table correction.

MC 847 WIDTH FIXED MEAS.PROBE (0-999999999INC)

During the measuring cycles the fixed measuring probe is approached from various directions. The machine constants 240, 290, 340, 390, 440 and 490 define the position of the fixed probe as a point. However, the probe has a given width for all directions, which is stored in increments.

MC 848 RAD.CALIBRATION RING (0-999999999INC)

The calibration of a moving measuring probe requires a calibration ring to be used. The radius of this ring is stored in increments.

MC 850 MEAS.PROBE COLLISION DETECT (0=NO,1-2)

Normally the collision detection is active during all movements of the measuring cycle (except the measuring movement).
A collision error is generated and the movements are stopped when the measuring probe is triggered during the rapid movements.
When no measuring cycle is active and the measuring probe is triggered also a collision error with movement stop is generated.

It may occur that because of vibrations of the machine tool the measuring probe is triggered. In order to prevent unjust collision detection a trigger delay can be made active.

Assign   0 for collision detection always active.
             1 for collision detection active during non rapid movements. During rapid movements the collision detection is active with delay.
             2 for collision detection is active with delay during the retract movement. During all other movements the collision detection without delay is active.

MC 851 MEAS.PROBE COLLISION DELAY(0-255*15MS)

The collision detection delay is stored in steps of 15 msec.
The measuring probe must be triggered for at least the delay time, to generate a collision error.

MC 852 MEAS.PROBE MP7 ACT.SPEED(0-1000REV/MIN)

The spinning speed to activate (deactivate) the Renishaw measuring probe MP7 is stored in revs/min.

MC 853 MEAS.PROBE MP7 TURN TIME (0-255*10MS)

The spinning time needed to activate (deactivate) the MP7 probe is stored in steps of 10 msec.
Example: The MP7 probe must spin for at least one revolution at 500 revs/min. When 500 is set to MC852, one revolution takes 1/500 min, which is equal to 120 msec. So 12 has to be stored to this machine constant.

MC 854 MEASURING PROBE MP7 DELAY (0-255*1S)

The minimum time between swiching-on and switching-off the measuring probe and vice verse is stored in steps of 1 sec.
The probe needs this time to be able to switch again.

MC 1000 - MC 1024 IPLC CONSTANT WORD 1-25 (0-65535)

These machine constants define the 25 word-parameters that can be used as constants in the IPLC program. The IPLC program can use these words to make adjustments in the IPLC program or to configure the IPLC program for different applications.

MC 1025 - MC 1049 IPLC CONSTANT BIT 1-25 (0-1)

These machine constants define the 25 bit-parameters that can be used as constants in the IPLC program. The IPLC program can use these bits to make adjustments in the IPLC program or to configure the IPLC program for different applications.

MC1060 NORMAL TASK GUARD (0-1000%)

This machine constant is for time guarding the IPLC program, the value is a percentage of the normal cycle time (50 ms).
If MC1060 is 0 the adjustable and fixed threshold guarding are switched off.
This means that window marker WIX_ADJ_GUARD is always zero, and WIX_LONGEST_CYCLE is no longer updated. When this machine constant is not zero, the CNC threats this value as the adjustable threshold percentage.

MC 1070 - MC 1079 OFFSET TO IPLC HOME-POSITIONS 1-10

For each axis 3 machine constants are reserved for home positions. To enable positioning of an axis to a home-position with an offset, ten offsets can be defined. The IPLC program orders an axis to move to a certain home-position.

MC 1080 MASK FOR ACTIVE BYTES

The mask for active bytes is the addition of the representation values of the bytes that must be active.

byte number	representation value	bytes to be activated
1	2	2
2	4	
3	8	
4	16	
5	32	32
6	64	64
"	"	
"	"	
15	32768	
		add

Mask for active bytes:98

MC 1081 MASK FOR ACTIVE WORDS

The mask for active words is the addition of the representation values of the words that must be active.

word number	representation value	words to be activated
1	2	2
2	4	
3	8	
4	16	
5	32	32
6	64	64
7	128	
		add

Mask for active words:98

MC 1086 IPLC PROGRAM IN ROM (0) OR IN RAM (1)

The IPLC program can be installed in RAM memory (DEBUG mode) or in ROM memory (EPROM on CPU board).

Assign   0 for IPLC program in ROM
          1 for IPLC program in RAM

MC 1087 IPLC SIZE OF CODE_MEMORY (KBYTE)

When MC1086 is set to IPLC program in RAM, this machine constant defines the numberKilobytes RAM allocated for the IPLC program.
When the IPLC program is in ROM, zero has to be assigned in order to free the memory space for part-programs.

MC 1090 IPLC MARKER BITS CLEARED AT COLD START (0-600)

The number of marker bits stored in this machine constant, will be cleared at cold start of the IPLC program.

MC 1091 IPLC MARKER BYTES CLEARED AT COLD START (0-150)

The number of marker bytes stored in this machine constant, will be cleared at cold start of the IPLC program.

MC 1092 IPLC MARKER WORDS CLEARED AT COLD START (0-150)

The number of marker words stored in this machine constant, will be cleared at cold start of the IPLC program.

MC 1093 IPLC COUNTERS CLEARED AT COLD START (0-50)

The number of counters stored in this machine constant, will be cleared at cold start of the IPLC program.

MC 1094 IPLC TIMERS CLEARED AT COLD START (0-50)

The number of timers stored in this machine constant, will be cleared at cold start of the IPLC program.

MC 1100 - MC1149
IPLC CONSTANT BYTES 1-50 (0-255)

These 50 MC's are reserved for the IPLC. They are not used by the CNC and can freely be used by the IPLC program. The user can define the meaning of these MC's e.g. to adjust parameters in the IPLC program of to configure the IPLC program for different applications.

MC 1200 - MC 1248 (even MC numbers)
LINEAR COMPENSATION POS.1 (INC.TO MZP) - FIRST AXIS

The position of the basepoint is stored in increments with respect to the Machine Zero Point.
When the position is not between the negative and positive software endswitch, the correction value is not taken into account.
Note that in case the Machine Zero Point is shifted by means of altering MC 233 all the compensation positions have to recalculated according to the displacement.

MC 1201 - MC 1249 (odd MC numbers)
LINEAR COMP.CORRECTION (+/-32000INC) - FIRST AXIS

The correction value is stored in increments with respect to the reference point where the correction value is zero.
Make sure that the maximum correction value is 1 increment per movement of 100 increments.
Note that in case the reference point is shifted all the compensation correction values have to be recalculated.

MC 1250 - MC 1298 (even MC numbers)
LINEAR COMPENSATION POS.1 (INC.TO MZP) - SECOND AXIS

The position of the basepoint is stored in increments with respect to the Machine Zero Point.
When the position is not between the negative and positive software endswitch, the correction value is not taken into account.
Note that in case the Machine Zero Point is shifted by means of altering MC 233 all the compensation positions have to recalculated according to the displacement.

MC 1251 - MC 1299 (odd MC numbers)
LINEAR COMP.CORRECTION (+/-32000INC) - SECOND AXIS

The correction value is stored in increments with respect to the reference point where the correction value is zero.
Make sure that the maximum correction value is 1 increment per movement of 100 increments.
Note that in case the reference point is shifted all the compensation correction values have to be recalculated.

MC 1300 - MC 1348 (even MC numbers)
LINEAR COMPENSATION POS.1 (INC.TO MZP) - THIRD AXIS

The position of the basepoint is stored in increments with respect to the Machine Zero Point.
When the position is not between the negative and positive software endswitch, the correction value is not taken into account.
Note that in case the Machine Zero Point is shifted by means of altering MC 233 all the compensation positions have to recalculated according to the displacement.

MC 1301 - MC 1349 (odd MC numbers)
LINEAR COMP.CORRECTION (+/-32000INC) - THIRD AXIS

The correction value is stored in increments with respect to the reference point where the correction value is zero.
Make sure that the maximum correction value is 1 increment per movement of 100 increments.
Note that in case the reference point is shifted all the compensation correction values have to be recalculated.

MC 1350 - MC 1398 (even MC numbers)
LINEAR COMPENSATION POS.1 (INC.TO MZP) - FOURTH AXIS

The position of the basepoint is stored in increments with respect to the Machine Zero Point.
When the position is not between the negative and positive software endswitch, the correction value is not taken into account.
Note that in case the Machine Zero Point is shifted by means of altering MC 233 all the compensation positions have to recalculated according to the displacement.

MC 1351 - MC 1399 (odd MC numbers)
LINEAR COMP.CORRECTION (+/-32000INC) - FOURTH AXIS

The correction value is stored in increments with respect to the reference point where the correction value is zero.
Make sure that the maximum correction value is 1 increment per movement of 100 increments.
Note that in case the reference point is shifted all the compensation correction values have to be recalculated.

MC 1400 - MC 1448 (even MC numbers)
LINEAR COMPENSATION POS.1 (INC.TO MZP) - FIFTH AXIS

The position of the basepoint is stored in increments with respect to the Machine Zero Point.
When the position is not between the negative and positive software endswitch, the correction value is not taken into account.
Note that in case the Machine Zero Point is shifted by means of altering MC 233 all the compensation positions have to recalculated according to the displacement.

MC 1401 - MC 1449 (odd MC numbers)
LINEAR COMP.CORRECTION (+/-32000INC) - FIFTH AXIS

The correction value is stored in increments with respect to the reference point where the correction value is zero.
Make sure that the maximum correction value is 1 increment per movement of 100 increments.
Note that in case the reference point is shifted all the compensation correction values have to be recalculated.

MC 1450 - MC 1498 (even MC numbers)
LINEAR COMPENSATION POS.1 (INC.TO MZP) - SIXTH AXIS

The position of the basepoint is stored in increments with respect to the Machine Zero Point.
When the position is not between the negative and positive software endswitch, the correction value is not taken into account.
Note that in case the Machine Zero Point is shifted by means of altering MC 233 all the compensation positions have to recalculated according to the displacement.

MC 1451 - MC 1499 (odd MC numbers)
LINEAR COMP.CORRECTION (+/-32000INC) - SIXTH AXIS

The correction value is stored in increments with respect to the reference point where the correction value is zero.
Make sure that the maximum correction value is 1 increment per movement of 100 increments.
Note that in case the reference point is shifted all the compensation correction values have to be recalculated.

MC 1500 - MC 1548 (even MC numbers)
CYCLIC COMPENSATION POS.1 (INC.TO MZP) - FIRST AXIS

The position of the basepoint is stored in increments with respect to the Machine Zero Point.
When the position is not between the negative and positive pitch limit, the correction value is not taken into account.
Note that in case the Machine Zero Point is shifted by means of altering MC 233 all the compensation positions have to recalculated according to the displacement.

MC 1501 - MC 1549 (odd MC numbers)
CYCLIC COMP.CORRECTION (+/-32000INC) - FIRST AXIS

The correction value is stored in increments with respect to the reference point where the correction value is zero.

Make sure that the maximum correction value is 1 increment per movement of 100 increments.

Note that in case the reference point is shifted all the compensation correction values have to be recalculated.

MC 1550 - MC 1598 (even MC numbers)
CYCLIC COMPENSATION POS.1 (INC.TO MZP) - SECOND AXIS

The position of the basepoint is stored in increments with respect to the Machine Zero Point.

When the position is not between the negative and positive pitch limit, the correction value is not taken into account.

Note that in case the Machine Zero Point is shifted by means of altering MC 233 all the compensation positions have to recalculated according to the displacement.

MC 1551 - MC 1599 (odd MC numbers)
CYCLIC COMP.CORRECTION (+/-32000INC) - SECOND AXIS

The correction value is stored in increments with respect to the reference point where the correction value is zero.

Make sure that the maximum correction value is 1 increment per movement of 100 increments.

Note that in case the reference point is shifted all the compensation correction values have to be recalculated.

MC 1600 - MC 1648 (even MC numbers)
CYCLIC COMPENSATION POS.1 (INC.TO MZP) - THIRD AXIS

The position of the basepoint is stored in increments with respect to the Machine Zero Point.

When the position is not between the negative and positive pitch limit, the correction value is not taken into account.

Note that in case the Machine Zero Point is shifted by means of altering MC 233 all the compensation positions have to recalculated according to the displacement.

MC 1601 - MC 1649 (odd MC numbers)
CYCLIC COMP.CORRECTION (+/-32000INC) - THIRD AXIS

The correction value is stored in increments with respect to the reference point where the correction value is zero.

Make sure that the maximum correction value is 1 increment per movement of 100 increments.

Note that in case the reference point is shifted all the compensation correction values have to be recalculated.

MC 1650 - MC 1698 (even MC numbers)
CYCLIC COMPENSATION POS.1 (INC.TO MZP) - FOURTH AXIS

The position of the basepoint is stored in increments with respect to the Machine Zero Point.
When the position is not between the negative and positive pitch limit, the correction value is not taken into account.
Note that in case the Machine Zero Point is shifted by means of altering MC 233 all the compensation positions have to recalculated according to the displacement.

MC 1651 - MC 1699 (odd MC numbers)
CYCLIC COMP.CORRECTION (+/-32000INC) - FOURTH AXIS

The correction value is stored in increments with respect to the reference point where the correction value is zero.
Make sure that the maximum correction value is 1 increment per movement of 100 increments.
Note that in case the reference point is shifted all the compensation correction values have to be recalculated.

MC 1700 - MC 1748 (even MC numbers)
CYCLIC COMPENSATION POS.1 (INC.TO MZP) - FIFTH AXIS

The position of the basepoint is stored in increments with respect to the Machine Zero Point.
When the position is not between the negative and positive pitch limit, the correction value is not taken into account.
Note that in case the Machine Zero Point is shifted by means of altering MC 233 all the compensation positions have to recalculated according to the displacement.

MC 1701 - MC 1749 (odd MC numbers)
CYCLIC COMP.CORRECTION (+/-32000INC) - FIFTH AXIS

The correction value is stored in increments with respect to the reference point where the correction value is zero.
Make sure that the maximum correction value is 1 increment per movement of 100 increments.
Note that in case the reference point is shifted all the compensation correction values have to be recalculated.

MC 1750 - MC 1798 (even MC numbers)
CYCLIC COMPENSATION POS.1 (INC.TO MZP) - SIXTH AXIS

The position of the basepoint is stored in increments with respect to the Machine Zero Point.
When the position is not between the negative and positive pitch limit, the correction value is not taken into account.
Note that in case the Machine Zero Point is shifted by means of altering MC 233 all the compensation positions have to recalculated according to the displacement.

MC 1751 - MC 1799 (odd MC numbers)
CYCLIC COMP.CORRECTION (+/-32000INC) - SIXTH AXIS

The correction value is stored in increments with respect to the reference point where the correction value is zero.

Make sure that the maximum correction value is 1 increment per movement of 100 increments.

Note that in case the reference point is shifted all the compensation correction values have to be recalculated.

MACHINE CONSTANTS FOR THE FIRST H-AXIS

MC 1800 FOLLOWING DISTANCE 1 (128-28500INC.)

Defines the position loop gain 1 of the servo drive. Gain 1 is effective until a given nod point is attained. From this point on the lower gain 2 is activated.
Not the gain factor is stored, but the following distance at which the servo output voltage is maximum.
The following distance is stored in increments. The smaller the number of increments, the higher the gain.

MC 1801 NODE POINT (0-28500INC.)

The point at which the gain factor changes, is stored in increments.

MC 1802 FOLLOWING DISTANCE 2 (128-28500INC.)

Gain 2 becomes active after the nod point has been attained.
Not the gain factor is stored, but the following distance at which the servo output voltage is maximum is stored in increments. The smaller the number of increments, the higher the gain.

MC 1803 INPOD DELAY TIME (0-255*15MS)

The "inpod" delay time makes sure that the machine slide is "in position" before the next command is executed.
The delay time is active only after positioning movements, and is started when the "in position"-window is reached.
The value is stored in steps of 15 msec.

Example: The delay time required is 100 msec. By assigning 7, a true delay time of 105 msec (7x15 msec) is achieved.

MC 1804 IN POSITION WINDOW (0-32000INC.)

To define the distance before the command position, where the "inpod"-delay time must start. The distance is stored in increments.
Theoretically, the servo output voltage is 0 V at the end of the positioning movement. In praxis the servo output will show a small positive or negative offset value. Consequently, the following distance display will show an offset value as well.
The "in position"-window must be slightly larger than the displayed following distance.

Example: If the displayed following distance is 4 increments, 10 has to be stored.

MC 1805 COUNT DIRECTION MEASURING SYSTEM (-1,1)

This machine constant defines the count direction of the measuring system. Assigning 1 will give a positive count direction while moving in the orientation direction. Assigning -1 gives a negative count direction in the same direction of movement.

MC 1806 MULTIPL.FACTOR S00/S90 (0=*1,1=*2,2=*4)

Assign   0 for 1 count-pulse decoded per S00/S90 cycle
           1 for 2 count-pulses decoded per S00/S90 cycle
           2 for 4 count-pulses decoded per S00/S90 cycle

When the axis is fitted with a linear measuring-system or when the axis does not exist, this machine-constant has no function.

MC 1807 BACKLASH COMPENSATION (0-32000INC.)

The backlash compensation will be activated the first movement in the opposite direction. The backlash is stored in increments.
Backlash compensation is only employed with rotary measuring-systems. For linear measuring-systems 0 has to be assigned to this machine constant in order to prevent erratic starts of positioning movements.

MC 1808 SOFTW.LIMIT SWITCH POSITIVE (INC.TO RP)

The position of the software limit switch in the positive or negative quadrant is defined. The sign indicates the quadrant, the value is the absolute position in increments with respect to the Machine Reference Point.

MC 1809 SOFTW.LIMIT SWITCH NEGATIVE (INC.TO RP)

The position of the software limit switch in the opposite quadrant of MC 1808 is defined. See MC 1808.

MC 1810 RAPID TRAVERSE RATE (1-320K*100INC/MIN)

The maximum traversing rate is stored in steps of 100 increments per minute.

Example: Required rapid traverse rate is 6 m/min. The value to be stored is 60,000.

MC 1811 JOG FEED RATE (0-320000*100INC/MIN)

The feedrate during jogging at 100% feed-override is stored in steps of 100 increments per minute.

Example: Required jog feed rate is 3 m/min. The value to be stored is 30,000.

MC 1812 RPF DIRECTION APPROACH (-1=NEG,1=POS)

Assign   1 for approaching the "area switch" in positive direction
             -1 for approaching the "area switch" in negative direction

MC 1813 RPF FEED RATE (0-320000*100INC/MIN)

Defines the velocity with which the "area switch" is to be approached.
The feedrate is stored in steps of 100 increments per minute.

Example: The required RPF feedrate is 2 m/min. The value to be stored is 20,000.

MC 1814 RPF CREEP FEED (0-320000*100INC/MIN)

After the "area switch" is actuated the machine slide starts moving in the re-
versed direction for reference point search. The velocity is stored in steps of
100 increments per minute.

The usual velocity for reference point search is 160 mm/min.
The maximum error related to the approaching velocity is:

velocity	value entered	maximum error
80 mm/min	800	0.5 um
160 mm/min	1600	1 um
330 mm/min	3300	2 um
500 mm/min	5000	3 um

MC 1815 RPF OFFSET (INC.BETWEEN MZP AND RP)

The distance between the machine zero-point and the reference point is stored
in increments.

MC 1816 RPF AREA SWITCH (0=OFF,1=ON)

Assign   1 for RPF finding using area-switch and marker signal.

MC 1819 STANDSTILL MONITORING (0-320000INC.)

The servo loops are permanently checked for errors by checking the following
distance. When the following distance increases during standstill, the NC sys-
tem checks whether the standstill monitoring value is exceeded. If so, an error
message is produced.
When setting the number of increments, the drift of the servo system as well as
the displacement in the axis as a result of milling operations are to be taken into
account.

MC 1820 ACC/DEC TIME CONSTANT (0,15-3840MS)

The acc/dec time constant is the time (in msec.) after which the servo drive out-
put voltage must reach 63% of the maximum output voltage, following the e-
curve.

MC 1821 ACC/DEC THRESHOLD(0-1048575*100INC/MIN)

When the acc/dec function is used and the axis responds too slowly when starting and moving into position, a the threshold value can be stored. Below this value the acc/dec is inoperative.
The value must be entered in steps of 40.

MC 1822 ACC/DEC MODE (0=NO,1=FIRST,2=SECOND)

The standard gain control does not satisfy the requirements of all type of machine tools. Some machine tools need a more sophisticated servo drive control.
The CNC offers a so-called acceleration/deceleration (acc/dec) function for the servo drive outputs to meet the requirements of those machine tools.
Activating the acc/dec function the servo output is updated according to an e-factor curve instead of steps.

Assign   0 for no acc/dec
         1 for acc/dec first order only in rapid movements
         2 for acc/dec second order only in rapid movements

MC 1823 AUXILIARY AXIS PRESENT (0=NO,2=YES)

Assign   2 to activate the auxiliary axis.
         Note that MC91 activates this machine constant.

MC 1827 WAIT FOR INPOD (0=NO,1=YES)

Assign   0 for the next movement being started without waiting for the inpod delay time.
         1 for the next movement to be started after the inpod delay time.

MC 1831 RESOLUT.MEAS.SYSTEM (1-3*DECIMALS/INC.)

Define the resolution of the axis by entering the number of digits behind the decimal point. The value 3 gives three decimals behind the decimal point, hence the resolution of the axis is 1 micron. Note that the display is adapted to the setting of this MC.
Note that the display always presents 3 decimals.

MC 1832 DISPLAY MODE (0=NORMAL,1=FOL.DIST.,2-5)

Assign   0 for display of the distance to go (normal display)

1 for display of the following distance

2 for display of the distance to go; without reference point search after power on

3 for display of the following distance; without reference point search after power on

4 for display of the distance to go; without reference point search after power on; error messages are not processed, just briefly displayed; UNCONTROLLED DISPLACEMENTS MAY OCCUR!

5 for display of the following distance; without reference point search after power on; error messages are not processed, just briefly displayed; UNCONTROLLED DISPLACEMENTS MAY OCCUR!

Note: Assigning 2, 3, 4, or 5 is only allowed for commissioning of the system and NEVER for normal operation, since THE SOFTWARE LIMIT SWITCHES ARE INOPERATIVE!

MC 1833 SELECTION OF DEMO-MODE (0=OFF,1=ON)

Assign   0 for normal operation

1 to activate DEMO-mode

Demo-mode means, that the NC system is operating without I/O-signals and without measuring-system signals. The program execution is simulated in the NC-system. The servo outputs allow an X/Y-recorder to be operated.

MC 1836 POSITION FOR "ON THE FLY" BIT 0 (INCR TO MZP)

This constant determines the position at which WIB_08_ON_THE_FLY bit 0 changes state when the axis passes this position.
The different bits in this marker are intended to drive mechanisms like doors and grippers which need to be operated at a particular position while the axis moves past this position. The position is related to the absolute zero-point of the axis.

MC 1837 SWITCH METHOD FOR BIT 0 (1-4)

This constant determines how WIB_08_ON_THE_FLY bit 0 changes state when the auxiliary axis passes the position defined by MC1836.

Distinction has been made between the situation before and after a reference point search. Befor RPF-search the bit can be set in such way that connected equipment is put in a defined state; e.g. transport door open, gripper closed etc.. This pre-defined situation will be maintained until a RPF-search in this axis.

After RPF-search the bit can transmit from 0 to 1 or from 1 to 0 when passing the defined position in positive or negative direction, depending on the entered MC value.

Entering the value "0" into this constant signifies that the "position sensoring" feature is disabled for this bit.

MC VALUE	STATE OF BIT IN WINDOW MARKER BEFORE RPF	STATE OF BIT IN WINDOW MARKER AFTER RPF
1	1	01
2	1	10
3	0	01
4	0	10

MC 1838 POSITION FOR "ON THE FLY" BIT 1 (0-32000INC)

This machine constant has the same meaning as MC1836 but defines the sensoring position for WIB_08_ON_THE_FLY bit 1. For further explanation see the description of MC1836

MC 1839 SWITCH METHOD FOR BIT 1 (1-4)

This machine constant has the same meaning as MC1837 but defines the functioning of WIB_ON_THE_FLY bit 1 in relation to the sensoring position in MC1838.

MC 1840 POSITION FOR "ON THE FLY" BIT 2 (0-32000INC)

This machine constant has the same meaning as MC1836 but defines the sensoring position for WIB_08_ON_THE_FLY bit 2. For further explanation see the description of MC1836

MC 1841 SWITCH METHOD FOR BIT 2 (1-4)

This machine constant has the same meaning as MC1837 but defines the functioning of WIB_ON_THE_FLY bit 2 in relation to the sensoring position in MC1838.

MC 1842 POSITION FOR "ON THE FLY" BIT 3 (0-32000INC)

This machine constant has the same meaning as MC1836 but defines the sensoring position for WIB_08_ON_THE_FLY bit 3. For further explanation see the description of MC1836

MC 1843 SWITCH METHOD FOR BIT 3 (1-4)

This machine constant has the same meaning as MC1837 but defines the functioning of WIB_ON_THE_FLY bit 3 in relation to the sensoring position in MC1838.

MC 1860 DISPLAY LINE NR. AUX AXIS (0=NO, 1-6=LINE NR.)

Since there are only six lines available for the axis position display, a selection must be made of the main axes and auxiliary axes to be displayed during "AUX-AX" display.

Assign 0 for no display of this axis
        1 for this axis to be displayed on the first line
        2 for this axis to be displayed on the second line
           etc.

MC 1861 ROUNDAXIS (0=NO,1=YES,2=OPT,3=LFT,4=RGHT)

Assign 0 in case the aux. axis is no roundaxis.
        1 for the aux. axis is to be treated as a roundaxis and no special options are wanted
        2 for roundaxis with optimised movements. For instance when a movement is programmed from 310° to 15°, the movement will be via the 0° position.
        3 for roundaxis with movements always in the counterclockwise direction.
        4 for roundaxis with movements always in the clockwise direction.

MC 1862 INDEXAXIS (0=NO, 1=YES)

Assign 0 for no indexaxis.
        1 for the aux. axis to be treated as an indexaxis.

MC 1863 RESOLUTION ROTATION AXIS (0-3,600,000)

Defines the resolution of a rotary axis, which is not an indexaxis

MC 1864 NUMBER OF INDEXES (0-255)

Defines the number of indexes for an indexaxis.

MC 1865 MAX.JOGSTEPS (0-5 INDEXES OR INCREMENTS)

Defines the number of increments or indexes the auxiliary axes is allowed to move when pushing a jog button.

MC 1866 TEXT JOG SOFTKEY (1=HOR,2=VER,3=ROT,4={C}CW)

When the axis is displayed during "AUX-AX" display, the axis can be jogged with softkeys.
The softkeys can be provided with the following text:
Assign   1 for horizontal axis; display: LEFT RIGHT
                2 for vertical axis; display: DOWN UP
                3 for rotary axis; display: BACKW FORW
                4 for rotary axis; display: CCW CW
                5 not valid for this version.

MC 1867 INCREMENT LEFT SOFTKEY (1=POS,2=NEG)

When the axis is displayed during "AUX-AX" display the axis can be jogged with softkeys.
Assign   1 for a positive movement when depressing the left softkey
                2 for a negative movement when depressing the left softkey.

MC 1870 HOME POSITION 01 (INC.TO MZP)

The position of home position 1 is stored in increments. The sign indicates the Quadrant, the value is the absolute position with respect to the Machine Zero Point.

MC 1871 HOME POSITION 02 (INC.TO MZP)

The position of home position 2 is stored in increments. The sign indicates the Quadrant, the value is the absolute position with respect to the Machine Zero Point.

MC 1872 HOME POSITION 03 (INC.TO MZP)

The position of home position 3 is stored in increments. The sign indicates the Quadrant, the value is the absolute position with respect to the Machine Zero Point.

MC 1873 HOME POSITION 04 (INC.TO MZP)

The position of home position 4 is stored in increments. The sign indicates the Quadrant, the value is the absolute position with respect to the Machine Zero Point.

MC 1874 HOME POSITION 05 (INC.TO MZP)

The position of home position 5 is stored in increments. The sign indicates the Quadrant, the value is the absolute position with respect to the Machine Zero Point.

MC 1875 HOME POSITION 06 (INC. TO MZP)

The position of home position 6 is stored in increments. The sign indicates the Quadrant, the value is the absolute position with respect to the Machine Zero Point.

MC 1876 HOME POSITION 07 (INC.TO MZP)

The position of home position 7 is stored in increments. The sign indicates the Quadrant, the value is the absolute position with respect to the Machine Zero Point.

MC 1877 HOME POSITION 08 (INC.TO MZP)

The position of home position 8 is stored in increments. The sign indicates the Quadrant, the value is the absolute position with respect to the Machine Zero Point.

MC 1878 HOME POSITION 09 (INC.TO MZP)

The position of home position 9 is stored in increments. The sign indicates the Quadrant, the value is the absolute position with respect to the Machine Zero Point.

MC 1879 HOME POSITION 10 (INC.TO MZP)

The position of home position 10 is stored in increments. The sign indicates the Quadrant, the value is the absolute position with respect to the Machine Zero Point.

MACHINE CONSTANTS FOR THE SECOND H-AXIS

MC 1900 - MC1979

Equal to MC1800-MC1879 but apply to the second auxiliary axis.

MC 2400 - MC 2954 POSITION AT INDEX XXX

These MC's define the index positions for the indexaxes related to the reference point.